® 2025 ANSYS, Inc. or its affiliated companies
Unauthorized use, distribution, or duplication is prohibited.

Maxwell® Help

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
https://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2025 R2
July 2025

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001:2015 companies.

Copyright and Trademark Information

© 1986-2025 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

Ansys, Ansys Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. All other brand, product, service and feature names or trademarks are the property of their respective owners. FLEXlm and FLEXnet are trademarks of Flexera Software LLC.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2015 companies.

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the legal information in the product help files for the complete Legal Notice for Ansys proprietary software and third-party software. If you are unable to access the Legal Notice, please contact ANSYS, Inc.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Table of Contents

Table of Contents **Contents-1**

1 - Welcome to Ansys Maxwell Help **1-1**

User Interface Quick Links 1-1

Modeling Quick Links 1-2

Boundaries/Excitations Quick Links 1-3

Meshing Quick Links 1-3

Analysis Quick Links 1-4

Optimetrics Quick Links 1-4

Results Quick Links 1-5

Finding Information in the Online Help 1-5

Using the Search Function in the Help 1-6

Help Conventions 1-10

Getting Help from Ansys Technical Support 1-12

2 - Getting Started with Ansys Electronics Desktop **2-1**

Ansys Electronics Desktop Student 2-3

HFSS Student Limitations 2-4

Q3D Extractor and 2D Extractor Student Limitations 2-4

Circuit Limitations 2-4

Maxwell and RMxprt Student Limitations 2-5

Icepak Student Limitations 2-5

System Requirements 2-5

Launching Ansys Electronics Desktop 2-5

Windows: 2-5

Linux: 2-6

Launching AEDT from the command line: 2-7

Starting from GNOME desktop: 2-7

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Product Improvement Program 2-7
Ansys Electronics Desktop Overview 2-11
Choosing a Color Scheme [Beta] 2-11
Mouse Button Usage 2-13
Working with Ribbons 2-13
IC Mode Layout Ribbon 2-17
Ansys Electronics Desktop Windows 2-17
Showing and Hiding Windows 2-17
Auto Hiding Windows 2-19
Moving and Resizing Windows 2-19
Project Manager Window 2-21
Working with the Project Tree 2-22
Setting the Project Tree to Expand Automatically 2-23
Viewing Material Definitions 2-24
Viewing Maxwell Design Details 2-25
Design List Window 2-26
Motion Variables Tab 2-27
Message Manager Window 2-28
Setting the Message Manager to Open Automatically 2-29
Showing New Messages 2-30
Automatically Expanding the Message Manager Tree 2-30
Action Messages 2-30
Clearing Messages 2-31
Clearing Messages for the Model 2-31
Copying Messages 2-32
Hiding the Message Manager Window until Messages Appear 2-32
Progress Window 2-32
Stopping or Aborting Simulation Progress 2-33

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stopping and Restarting Time Decomposition Method Analyses 2-33
Viewing Distributed Analysis Subtasks 2-33
Properties Window 2-34
Opening the Properties Window 2-35
Setting the Properties Window to Open Automatically 2-35
Showing and Hiding the Properties Window 2-36
Modifying Object Attributes Using the Properties Window 2-36
Modifying Object Command Properties Using the Properties Window 2-36
Property Window for Schematic and Layout Editors 2-37
Param Values Tab 2-37
Launching Help from the Property Window 2-38
General Tab 2-39
Symbol Tab 2-39
Closing the Property Window 2-41
Properties Dialog Box 2-41
Opening the Properties Dialog Box 2-42
Parameter Values Tab 2-42
General and Symbol Tabs 2-43
Property Displays Tab 2-43
ACT Extensions Window 2-45
Ansys Electronics Desktop Menus 2-50
Menu Bar 2-51
File Menu 2-52
Edit Menu 2-53
Edit Menu for 3D Modeler 2-53
Edit Menu for Report Window 2-56
View Menu 2-57
Basic View Menu 2-57

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3D Modeler View Menu 2-58
Project Menu 2-63
Editor and Design Specific Menus 2-64
Layout Editor Draw Menu 2-66
Tools Menu 2-67
Understanding Registry Tools 2-71
Adding External Tools to the Tools Menu 2-74
Window Menu 2-76
Help Menu 2-79
Context-Sensitive Help 2-80
Obtaining Information about the Software and Release 2-80
Shortcut Menus 2-81
Shortcut Menu in the 3D Modeler Window 2-82
Shortcut Menus in the Project Manager Window 2-83
Customizing Ansys Electronics Desktop Menus 2-85
Ansys Electronics Desktop Design Area 2-95
3D Modeler Window 2-96
Report Window 2-96
Ansys Electronics Desktop History Tree 2-97
Organizing the History Tree 2-99
Controlling the View of Objects in the History Tree 2-100
Group Commands for Modeler Objects 2-101
Accessing Group Commands 2-102
Group Command Descriptions 2-103
Operations on Groups 2-103
Group Properties 2-105
Selecting Objects in the History Tree 2-106
Shortcut Menus in the History Tree 2-109

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

View Item Properties in the History Tree 2-111
Viewing and Editing Commands on History Tree Objects 2-111
Upgrading Version in the History Tree 2-112
Ansys Electronics Desktop Status Bar 2-113
Keyboard Shortcuts 2-114
Desktop Shortcuts 2-114
3D Modeler Window Shortcuts 2-115
Report Shortcuts 2-116
Choosing the View Navigation Options 2-117
Custom Keyboard Shortcuts 2-119
Auto-Complete for Variables and Properties in Electronics Desktop 2-120
Using the Password Manager to Control Access to Resources 2-124
Running Ansys Electronics Desktop from a Command Line 2-125
Command-line Syntax 2-125
Run Commands 2-126
Job Management from the Command Line 2-131
Run Command Examples 2-132
Specifying Project Files 2-132
Options 2-133
-Batchoptions 2-134
-Batchoptions Examples 2-135
Export Options Files 2-135
Examples and Further Explanations of -batchoptions Use 2-136
For -batchoptions Use: Project Directory and Lib Paths 2-138
For -batchoptions Use: TempDirectory 2-139
For -batchoptions Use: Various Desktop Settings 2-139
For -batchoptions Use: Maxwell 2D and Maxwell 3D Options 2-142
Batchoptions: Override Registry Entry 2-145

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Running Ansys Electronics Desktop from a Command Line (Nongraphical) Beta 2-146
Command-line Syntax 2-146
Run Commands 2-146
Run Command Examples 2-151
Specifying Project Files 2-151
Options 2-152
-Batchoptions 2-152
-Batchoptions Examples 2-153
Export Options Files 2-154
Running Ansys Electronics Desktop with a JSON File 2-154
JSON File Format 2-154
Running Ansys Electronics Desktop from a Windows Remote Terminal 2-156
Using Windows HPC Commands 2-157
Adding External Tools 2-157
Customizing Ansys Electronics Desktop with Ansys ACT 2-159
Installing PyAEDT (Beta) 2-159
Debug Logging 2-160
3 - Working with Ansys Electronics Desktop Projects 3-1
Ansys Electronics Desktop Files 3-1
Maxwell Files 3-2
Creating Projects 3-2
Setting Project Options 3-4
Setting General Options 3-6
General Options: Desktop Configuration 3-8
General Options: Project Options 3-10
General Options: Miscellaneous Options 3-11
General Options: User Interface Options 3-12
General Options: Directories Options 3-16

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Options: Desktop Performance 3-17
General Options: Default Units 3-20
General Options: Remote Analysis Options 3-21
General Options: Component Libraries Options 3-22
Setting 3D Modeler Options 3-24
3D Modeler Options: Drawing Options 3-25
3D Modeler Options: Operation Options 3-26
3D Modeler Options: Snap Options 3-27
3D Modeler Options: Display Options 3-29
Display Options: General Options 3-30
Display Options: Rendering Options 3-32
Display Options: History Tree Options 3-34
3D Modeler Options: Group Options 3-35
3D Modeler Options: Discovery Link Options 3-35
3D Modeler Options: Advanced Options 3-38
Setting Maxwell 2D Options 3-39
Setting Maxwell 3D Options 3-40
Setting RMxprt Options 3-41
Setting Reporter Options 3-42
Reporter Options: Report Setup Options 3-42
Reporter Options: Fields Reporter Options 3-43
Reporter Options: Report2D Options 3-44
Report2D Options: Curve 3-44
Report2D Options: Axis 3-45
Report2D Options: Grid 3-45
Report2D Options: Header 3-46
Report2D Options: Note 3-46
Report2D Options: Legend 3-46

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Report2D Options: Marker ... 3-47
Report2D Options: Marker Table ... 3-47
Report2D Options: X-Y Markers ... 3-48
Report2D Options: Stacked ... 3-49
Report2D Options: Digital ... 3-50
Report2D Options: General ... 3-50
Curve Tooltip ... 3-50
Clipboard Option ... 3-51
Format ... 3-51
Report2D Options: Table ... 3-51
Header Row ... 3-51
Format ... 3-52
Copy to Clipboard ... 3-52
Setting HPC and Analysis Options ... 3-52
Configurations Tab ... 3-53
Selecting an Available Configuration ... 3-54
Options Tab ... 3-55
Specifying the Remote Spawn Command (Linux) ... 3-60
Licensing Settings Tool ... 3-61
Using GPU Acceleration for Maxwell 3D Solvers ... 3-64
Enabling GPUs for Maxwell 3D Solvers ... 3-66
Setting Up Non-distributed GPU for Maxwell 3D Solvers ... 3-67
Enabling Distributed GPUs for AC Magnetic Frequency Sweep ... 3-68
Setting Options via Configuration Files ... 3-71
Behavior Examples ... 3-72
Rules for Modifying Option Settings ... 3-72
Configuration File Locations ... 3-73
Products with Multiple Desktop Versions ... 3-73

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Table of Directories and Files 3-73

Setting or Removing Option Values in Configuration Files: UpdateRegistry Command 3-75

UpdateRegistry -Set Command 3-75

UpdateRegistry -Get Command 3-76

UpdateRegistry -GetKeys Command 3-77

UpdateRegistry -Delete Command 3-78

UpdateRegistry -FromFile Command 3-78

UpdateRegistry File Format 3-79

Example Uses for Export Options Features 3-80

Options that Apply to All Users 3-80

Example: Searching for a Registry Key Pathname 3-81

Example: Setting an Installation Default Value 3-81

Example: Setting a Host-Dependent Default Value 3-81

Example: Reverting from a User-Defined Option Value to the Administrator Default 3-82

User Options and the UpdateRegistry Tool 3-83

Example: Removing a Host-Dependent User Option Setting 3-84

Example: Getting a Value from a Specific Configuration File 3-84

Example: Getting a Value Using Precedence Rules 3-85

Example: Adding a Host-Independent User Option Setting 3-85

Example: Setting a RegistryKey not defined in ElectronicsDesktopRegistrySyntax.xml 3-85

Example: Change a Newly Set Registry Key 3-85

Setting the Temporary Directory 3-86

Temporary Directory Configuration File Format 3-88

UpdateRegistry: Setting or Removing Temporary Directory Values in Configuration Files 3-89

Batchoptions Command Line Examples 3-90

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Batchoptions File Format ... 3-90
Example -BatchOptions with -Remote (Windows) ... 3-91
Example -Batchsolve with -Machinelist (Windows) ... 3-92
Example -Batchsolve with -Machinelist (Linux) ... 3-92
Example -Batchsolve for Local (Windows) ... 3-93
Batch Options and Analysis Configurations in the Registry ... 3-94
Managing Projects and Designs ... 3-98
Opening Projects ... 3-99
Opening Recent Projects ... 3-101
Opening Example Projects ... 3-102
Opening Legacy Maxwell Projects ... 3-102
Creating Projects ... 3-104
Creating New 2D Designs from 3D Designs ... 3-105
Creating New 3D Designs from 2D Designs ... 3-106
Creating a Target Design from 3D Designs ... 3-108
Icepak Designs ... 3-109
Mechanical Designs ... 3-110
Saving Projects ... 3-111
Path Name Length Issues for Windows ... 3-111
Changing Auto-Save Settings ... 3-113
Save Before Solving Option ... 3-114
Recovering Project Data from an Auto-Save File ... 3-114
Importing and Exporting Projects and Data ... 3-115
Importing Files ... 3-115
Importing 2D Model (GDSII) Files ... 3-117
Creating GDSII Layer Mapping Files ... 3-119
Tech File Colors with RGB Values ... 3-120
Importing 3D Model Files ... 3-134

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Importing DXF and DWG Format Files ... 3-136
Importing Data Tables ... 3-139
Importing Tabular Arrays ... 3-140
Importing Plot Data ... 3-143
Exporting Files ... 3-144
Exporting 2D Model Files ... 3-144
Exporting 2D AutoCAD Drawing Interchange Format Files ... 3-144
Exporting 3D Model Files ... 3-145
GDSII Files ... 3-146
Parasolid Files ... 3-147
Exporting Graphics Files ... 3-147
Exporting Options Files ... 3-149
Exporting Options Files Using the Desktop UI ... 3-149
Exporting Options Files Using a Script ... 3-149
Validating Projects ... 3-149
Validations Tab ... 3-151
Copying and Pasting a Project or Design ... 3-152
Renaming Projects ... 3-153
Archiving Projects ... 3-153
Restoring an Archived Project ... 3-157
Deleting a Project or Design ... 3-160
Setting Read Only Designs ... 3-160
Updating Design Components ... 3-161
Removing Unused Components from a Project ... 3-162
Undoing and Redoing Commands ... 3-162
Inserting a Documentation File ... 3-163
Working with Design Notes ... 3-163
Printing ... 3-164

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Minerva Remote Storage Environment ... 3-165
Example Projects ... 3-173
Working with PEmag Designs ... 3-173
Creating Maxwell Designs Using PEmag ... 3-174
Modifying Maxwell Designs Using PEmag ... 3-177
Solving Maxwell Designs Generated by PEmag ... 3-178
Working with Variables ... 3-179
Working with Project Variables ... 3-181
Adding a Project Variable ... 3-181
Importing and Exporting Project Variables ... 3-185
Deleting a Project Variable ... 3-185
Editing a Project Variable ... 3-186
Adding a Project Variable Array ... 3-186
List of Intrinsic Variables ... 3-190
List of Constants ... 3-192
Working with Design Variables ... 3-192
Defining Local Variables at the Design Level ... 3-193
Adding a Local Variable ... 3-193
Deleting a Local Variable ... 3-197
Editing a Local Variable ... 3-198
Overriding a Local Variable ... 3-198
Defining Parameter Defaults at the Design Level ... 3-198
Adding a Parameter Default ... 3-198
Deleting a Parameter Default ... 3-200
Editing a Parameter Default ... 3-200
Overriding a Parameter Default ... 3-201
Adding a Design Variable Array ... 3-201
Converting Variables and Parameter Defaults ... 3-206

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adding Variables and Parameter Defaults from Components ... 3-206
Fixed vs. Non-Fixed Variables ... 3-209
Non-Fixed Variable Sweeps ... 3-210
Orphaned Sweeps ... 3-210
Choosing a Variable to Optimize ... 3-211
Choosing a Variable to Tune ... 3-212
Including a Variable in a Sensitivity Analysis ... 3-213
Including a Variable in a Statistical Analysis ... 3-214
Copying and Pasting a Variables List ... 3-216
Working with Datasets ... 3-216
Adding Datasets ... 3-217
Dataset Preview Plot Properties ... 3-221
Importing Datasets ... 3-222
Editing Datasets ... 3-223
Cloning Datasets ... 3-225
Exporting Datasets ... 3-228
Removing Datasets ... 3-229
Changing Dataset Plot Properties ... 3-231
Using SheetScan ... 3-233
SheetScan Menus and Settings ... 3-234
SheetScan View Options ... 3-234
SheetScan Right-click Menu ... 3-235
SheetScan Settings ... 3-236
Loading a Datasheet Picture into SheetScan ... 3-236
Loading a Datasheet Picture Directly ... 3-236
Loading a Datasheet Picture Using the HTML Viewer ... 3-236
Defining a SheetScan Coordinate System ... 3-237
Defining a Characteristic Curve in SheetScan ... 3-238

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Performing Operations on SheetScan Curves ... 3-240
Checking for Monotonicity in X ... 3-241
Importing Characteristic Data into SheetScan ... 3-242
Exporting SheetScan Data ... 3-244
Defining an Expression ... 3-245
Valid Operators for Expressions ... 3-245
Using Intrinsic Functions in Expressions ... 3-246
clp Formula ... 3-247
Using Piecewise Linear Functions in Expressions ... 3-248
Using Dataset Expressions ... 3-249
Handling Delta Temperature Units in Expressions ... 3-251
Defining Mathematical Functions ... 3-253
Maxwell ACT Extensions Wizards ... 3-253
Using the Maxwell Machine Toolkit Wizard (Windows only) ... 3-254
Interactive Solve ... 3-254
Delayed Solve ... 3-255
Solve Using LS-DSO ... 3-256
Resume an Interrupted Parametric Sweep Run ... 3-258
Generate Efficiency Map Only ... 3-258
Create Efficiency Maps Based on Reduced-Order Model ... 3-262
Defining Machine Toolkit Parameters ... 3-263
Using the Maxwell Eccentricity Wizard ... 3-269
For 2D Designs ... 3-271
For 3D Designs ... 3-272
4 - Setting Up a Maxwell Design ... 4-1
Inserting a Maxwell Design ... 4-1
Creating New 2D Designs from 3D Designs ... 4-2
Converting a 3D Model to 2D ... 4-2

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating New 3D Designs from 2D Designs ... 4-3
Converting 2D XY Model to 3D ... 4-3
Converting 2D RZ Model to 3D ... 4-3
Setting Units of Measurement for the Model ... 4-4
Design Settings for the Maxwell Application ... 4-5
Set Material Threshold Tab ... 4-7
Model Depth Settings ... 4-8
Set Material Override Tab ... 4-9
Inductance Type Settings for 3D Magnetostatic Solutions ... 4-9
Inductance Computation Tab Settings for 2D Magnetostatic Solutions ... 4-10
Matrix Computation Tab Settings for Transient Solutions ... 4-11
Validations Tab ... 4-12
Setting a Symmetry Multiplier ... 4-14
Export Options Tab ... 4-16
Model Settings Tab ... 4-17
User-Defined Skew Type Computations ... 4-22
Background Material Tab ... 4-23
Preserve Transient Solution Tab ... 4-23
Advanced Product Coupling Tab ... 4-24
5 - Specifying the Solver Type ... 5-1
Magnetostatic Solver ... 5-2
AC Magnetic (Eddy Current) Solver ... 5-2
Transient Solver ... 5-4
Maxwell 3D Transient A-Phi Solution ... 5-4
Comparison of T-Omega and A-Phi Solvers ... 5-5
Limitations ... 5-5
Boundaries and Excitations ... 5-5
Winding Excitations for A-Phi Formulation ... 5-7

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coil Order ... 5-7
Voltage Excitations for A-Phi Formulation ... 5-9
Current Excitations for A-Phi Formulation ... 5-12
Rules/Limitations of Excitations on Conduction Paths ... 5-14
Winding Excitations ... 5-14
Current Excitations ... 5-14
Voltage Excitations ... 5-15
Matrix Setup ... 5-15
Postprocessing Support ... 5-17
Electrostatic Solver ... 5-19
Meshing Recommendations for the Electrostatic Solver ... 5-19
2D AC Conduction ... 5-21
3D AC Conduction ... 5-21
DC Conduction Solver ... 5-22
Electric Transient Solver ... 5-22
Maxwell Solvers Material Properties ... 5-23
Disconnected Solution Domains ... 5-27
6 - Assigning Materials ... 6-1
Solving Inside or on the Surface ... 6-7
Searching for Materials ... 6-7
Searching by Material Property ... 6-10
Adding New Materials ... 6-12
Assigning Material Property Types ... 6-15
Relative Permeability for a Maxwell or RMxprt Material ... 6-16
Specifying B-H Curves for Nonlinear Relative Permeability ... 6-17
Defining Anisotropic Relative Permeability Tensors ... 6-28
Eddy or Frequency Domain Solutions Only ... 6-28
Magnetostatic, Eddy, and Transient Solutions Only ... 6-28

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General ... 6-28
Temperature-Dependent B-H Curves ... 6-29
Dielectric Loss Tangent for a Maxwell Material ... 6-30
Defining Anisotropic Dielectric Loss Tangent Tensors ... 6-30
Relative Permittivity for a Maxwell Material ... 6-31
Defining Anisotropic Relative Permittivity Tensors ... 6-31
Eddy or Frequency Domain Solutions Only ... 6-31
Specifying a D-E Curve for Nonlinear Relative Permittivity ... 6-32
Bulk Conductivity for a Maxwell Material ... 6-34
Defining Anisotropic Conductivity Tensors ... 6-35
Specifying a J-E Curve for Nonlinear Bulk Conductivity ... 6-35
Defining Anisotropic Tensors ... 6-37
Assigning Anisotropic Tensors ... 6-38
Cylindrical Anisotropic Material Properties: Tensor Conversion from Cylindrical to Cartesian CS ... 6-41
Spherical Anisotropic Material Properties: Tensor Conversion from Spherical to Cartesian CS ... 6-45
Magnetic Loss Tangent ... 6-49
Imaginary Permeability ... 6-49
AC Magnetic ... 6-49
Defining Magnetic Loss Tangent Tensors ... 6-50
Defining Variable Material Properties ... 6-51
Specifying Thermal Modifiers ... 6-51
Defining Material Properties as Expressions ... 6-59
Defining Functional Material Properties ... 6-60
Material Coordinate Systems ... 6-60
Lamination Modeling ... 6-61
Stacking Direction ... 6-63

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Litz Wire Modeling ... 6-63
Core Loss Model for a Maxwell Material ... 6-67
Additional Considerations ... 6-70
DC-Biased Effects ... 6-70
Hysteresis Model-Based Core Loss Computation Approach ... 6-70
Additional Core Loss Due to Flux Normal to Laminations ... 6-71
Additional Core Loss Due to Manufacturing Processes ... 6-71
Cut Edge Effects on Core Loss for Lamination Steels ... 6-72
Calculating Properties for Core Loss (BP Curve) ... 6-76
Core Loss Coefficients for Electrical Steel ... 6-79
Core Loss Coefficient Extraction from Single-Frequency Loss Curve ... 6-80
Core Loss Coefficient Extraction from Multi-Frequency Loss Curves ... 6-81
Core Loss Parameter Extraction for Power Ferrite Materials ... 6-82
Importing a B-P Curve From a File ... 6-82
Manually Specifying B-P Curve Coordinates ... 6-83
Temperature-Dependent Core Loss Curves ... 6-84
Extracting Equivalent Cut Depth Based on Multiple Curves ... 6-86
Parameter Identification Based on Single-Sheet Tester ... 6-87
Assigning Materials from the Object Properties Window ... 6-89
Viewing and Modifying Material Attributes ... 6-90
Setting the Material Threshold ... 6-93
Validating Materials ... 6-93
Copying Materials ... 6-93
Removing Materials ... 6-94
Exporting Materials to a Library ... 6-94
Sorting Materials ... 6-94
Filtering Materials ... 6-95
Working with Material Libraries ... 6-96

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Editing Libraries ... 6-97
Editing Material Definitions ... 6-97
Setting a Thermal Modifier ... 6-98
Setting a Spatial Modifier ... 6-99
Importing and Converting Materials ... 6-102
Setting the Temperature of Objects ... 6-102
Reverting Objects to Initial Temperature ... 6-107
Setting Deformation of Objects for Stress Feedback ... 6-107
Reverting Objects to Zero Displacement ... 6-110
Permanent Magnets ... 6-111
Nonlinear vs. Linear Permanent Magnets ... 6-111
Compute Remanent Br from B-H Curve ... 6-111
Temperature-Dependent Nonlinear Permanent Magnets ... 6-112
Irreversible Demagnetization Due to Temperature Change ... 6-120
Characteristics and Main Parameters of Permanent-Magnetic Materials ... 6-122
Demagnetization Curve ... 6-123
Recoil Lines ... 6-124
Recoil Magnetic Permeability ... 6-125
Inflection Point ... 6-125
Calculating the Properties for a Permanent Magnet ... 6-126
Calculating the Properties for a Nonlinear Permanent Magnet ... 6-126
Creating a Nonlinear B-H Curve with Four Parameters ... 6-129
Calculating Properties Using a Hysteresis Loop in Maxwell ... 6-130
Magnetostriction and Inverse Magnetostriction ... 6-134
Calculating the Properties for Magnetostriction ... 6-135
Calculating the Properties for Inverse Magnetostriction ... 6-142
Perfect Conductors ... 6-149
7 - Drawing a Model ... **7-1**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Units of Measurement for the Model 7-1
Drawing Objects 7-3
Drawing a Line Segment 7-5
Drawing a Three-Point Arc Line 7-10
Drawing a Center-Point Arc Line 7-12
Drawing a Spline 7-16
Drawing a Polyline 7-19
Assigning a Cross Section and Dimensions to a Polyline 7-21
Inserting Line Segments 7-23
Drawing an Equation-Based Curve 7-24
Functions, Operators and Quantities for the Edit Equation Dialog Box 7-28
Drawing a Circle 7-29
Drawing an Ellipse 7-32
Drawing a Rectangle 7-36
Drawing a Regular Polygon 7-39
Drawing an Equation-Based Surface 7-42
Drawing a Sphere 7-46
Drawing a Cylinder 7-50
Drawing a Box 7-54
Drawing a Regular Polyhedron 7-58
Drawing a Cone 7-61
Drawing a Torus 7-64
Drawing a Helix 7-67
Drawing a Segmented Helix with Polygon Cross-Section Using a User Defined Primitive 7-71
Drawing a Segmented Helix with Rectangular Cross-Section Using a User Defined Primitive 7-72
Drawing a Spiral 7-73

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Drawing a Spiral Using User-Defined Primitives 7-76
Drawing a Geodesic Sphere Using a User-Defined Primitive 7-76
Drawing a Bondwire 7-77
Drawing a Point 7-78
Drawing a Plane 7-79
Creating Segmented Geometry 7-80
Segmented Objects 7-81
Drawing Non-Model Objects 7-82
Selecting Non-Model Drawing Mode 7-83
Changing an Object to Non-Model 7-83
Drawing a Region 7-83
Drawing a SubRegion 7-86
Reassigning a SubRegion 7-92
Group Commands for Modeler Objects 7-95
Accessing Group Commands 7-97
Group Command Descriptions 7-97
Operations on Groups 7-98
Group Properties 7-99
Model Analysis 7-100
Analysis Options Dialog Box 7-101
Analyzing the Surface Mesh 7-102
Model Analysis Dialog Box 7-103
Objects Tab 7-104
Surface Mesh (Single/Pairs) Tab 7-106
Object Misalignment Tab 7-106
Mesh Feedback Tab 7-107
Model Preparation Commands 7-112
Simplify Command 7-113

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Heal ... 7-120
Validation and Healing Options for Import File ... 7-122
Healing Stage One: Validation Check ... 7-122
Healing Stage Two: Fix Intersections Between Objects ... 7-123
Healing Stage Three: Fix Object Pair Intersections ... 7-123
Healing Stage Four: Analysis Phase ... 7-124
Healing Non-Manifold Objects ... 7-124
Setting the Healing Options ... 7-126
Stitch Sheets ... 7-127
Align Selected Faces ... 7-127
Remove Selected Faces ... 7-129
Remove Selected Edges ... 7-129
Perform Explicit Subtractions ... 7-130
User Customization through User-Defined Primitives (UDPs) ... 7-131
Creating a User-Defined Primitive ... 7-132
Specifying UDP Parameters ... 7-133
Updating the User-Defined Primitives Menu ... 7-134
User-Defined Primitives (UDPs) for Motor Geometries ... 7-134
Stator Geometries ... 7-135
Rotor Geometries ... 7-136
Pole Geometries ... 7-137
Parameters for RMxprt User-Defined Primitives ... 7-138
RMxprt/Band ... 7-138
RMxprt/ClawPoleCore ... 7-138
RMxprt/ConCoil ... 7-139
RMxprt/DCMCore ... 7-140
RMxprt/DiskCoil ... 7-142
RMxprt/DiskPMCore ... 7-143

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RMxprt/DiskSlotCore ... 7-143
RMxprt/DoubleCage ... 7-144
RMxprt/IPMCore ... 7-145
RMxprt/LapCoil ... 7-145
RMxprt/LinearMCore ... 7-146
RMxprt/NonSalientPoleCore ... 7-147
RMxprt/PMCore ... 7-147
RMxprt/PMDamperCore ... 7-148
RMxprt/RacetrackSlotCore ... 7-149
RMxprt/SalientPoleCore ... 7-150
RMxprt/SlotCore ... 7-151
RMxprt/SquirrelCage ... 7-151
RMxprt/SRMCore ... 7-152
RMxprt/SynRMCore ... 7-152
RMxprt/TransCoil ... 7-153
RMxprt/TransCore ... 7-153
RMxprt/UnivMCore ... 7-154
RMxprt/VentSlotCore ... 7-155
RMxprt/WaveCoil ... 7-156
User-Defined Model (UDM) ... 7-157
UDM Compared to User-Defined Primitives ... 7-158
Insert UDM Command on Draw Menu ... 7-159
UDM Properties ... 7-159
UDM Parameters ... 7-160
UDM Part Edits ... 7-160
Library of Models for CAD Integration ... 7-161
3D Modeler Discovery Link ... 7-163
Importing Discovery Geometry into Ansys Electronics Desktop ... 7-163

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Preserving IDs During Parametric Variations 7-166
Connecting to an Active Discovery Session 7-167
Property Window of Discovery Link UDM 7-168
Archive a Project Containing Discovery Link 7-170
Break Connection with Discovery Link 7-170
SpaceClaim Integration 7-171
3D Component Library 7-171
Creating a 3D Component from an Existing Model 7-173
Creating a 3D Component with an External Circuit 7-177
Create 3D Component: Model Tab 7-181
Create 3D Component Boundaries Tab 7-182
Create 3D Component Excitations Tab 7-183
Create 3D Component Mesh Tab 7-184
Create 3D Component Coordinate Systems Tab 7-185
Create 3D Component Parameters Tab 7-186
Non-Geometry Variables in 3D Components 7-187
Dependent Variables in 3D Components 7-187
Create 3D Component Encryption Tab 7-188
Set Passwords 7-191
Hiding Component Content 7-192
Inserting Encrypted Components 7-193
Editing Encrypted Components 7-193
Opening a 3D Component File 7-194
Copying and Pasting Encrypted Components 7-194
Create 3D Component Licensing Tab 7-195
Create 3D Component Image Tab 7-195
Create 3D Component Info Tab 7-198
Save 3D Component File 7-200

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using the Component Libraries Window 7-201
Component file organization on the disk 7-205
Inserting a 3D Component into a Design 7-205
Replace Selection by 3D Component 7-213
Viewing Properties of 3D Components 7-217
3D Components in the Project Manager Window 7-220
3D Components in the History Tree 7-221
Editing 3D Component Properties 7-223
Editing a 3D Component Definition 7-227
Verifying Updates to Components 7-230
Updating Definitions 7-230
Replacing 3D Components 7-231
Updating a 3D Component Definition 7-232
Human Body Exteriors 7-233
Qi Wireless Power Transfer System 7-236
Legacy Projects with 3D Components 7-239
Modifying Objects 7-239
Modify Object Appearance 7-239
Modify Object Dimensions 7-239
Modify Object Location or Orientation 7-240
Surface or Edge Operations for Objects 7-240
Boolean Operations on Objects 7-240
Assigning Color to Objects 7-241
Setting the Default Color of Objects 7-241
Assigning Transparency to an Object 7-242
Setting the Default Transparency of Objects 7-242
Copying and Pasting Objects 7-243
Import a Model from the Clipboard 7-244

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copy Image 7-245
Deleting Objects 7-246
Deleting Start Points and Endpoints 7-246
Delete Last Operation 7-247
Cutting Objects 7-247
Moving Objects 7-247
Rotating Objects 7-249
Changing the Orientation of an Object 7-251
Mirroring Objects 7-251
Offsetting Objects 7-253
Duplicating Objects 7-254
Duplicating Objects Along a Line 7-255
Duplicating Objects Around an Axis 7-256
Duplicating and Mirroring Objects 7-258
Scaling Objects 7-258
Sweeping Objects 7-260
Sweeping Around an Axis 7-260
Sweeping Along a Vector 7-262
Sweeping Along a Path 7-264
Best Practices 7-264
Sweeping Faces Along Normal 7-266
Draft Type Options 7-266
Thicken Sheet 7-269
Wrap Sheet Command 7-270
Covering Lines 7-275
Covering Faces 7-275
Uncovering Faces 7-276
Detaching Faces 7-276

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Detaching Edges 7-276
Creating a Cross-Section 7-277
Connecting Objects 7-277
Project Sheet Object 7-278
Moving Faces or Edges 7-282
Moving Faces Along the Normal 7-282
Moving Faces Along a Vector 7-283
Moving Edges Along Normal 7-285
Uniting Objects 7-286
Subtracting Objects 7-286
Creating Objects from Intersections 7-288
Creating an Object from a Face 7-289
Creating an Object from an Edge 7-290
Splitting Objects 7-291
Separating Bodies 7-295
Converting Polyline Segments 7-295
Rounding the Edge of Objects (Fillet Command) 7-296
Flattening the Edge of Objects (Chamfer Command) 7-298
Imprinting an Object 7-301
Imprint Projection Commands 7-304
Purging Object History 7-307
Generate History to Reproduce Portions of Model 7-307
Selecting Items in the 3D Modeler Window 7-308
Selecting Objects 7-310
Selecting Several Objects 7-311
Selecting Objects by Name or Material 7-312
Selection by Object Name 7-313
Selection by Object Material 7-313

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select By Area 7-314
Setting the Default Color and Transparency of Selected Objects 7-316
Setting the Default Color of Highlighted Objects 7-316
Creating an Object Selection 7-317
Reassigning Objects to Another Object Selection 7-318
Using or Viewing Selection Objects 7-318
Selecting Faces 7-319
Selecting All Faces of an Object 7-321
Selecting Faces by Name 7-321
Selecting Faces by Plane 7-322
Creating a Face Selection 7-323
Selecting Edges 7-324
Selecting All Edges of an Object or Face 7-326
Selecting Vertices 7-327
Selecting Multi 7-328
Controlling the Selection in Multi Mode 7-330
Selecting Groups and Submodels 7-330
Clearing a Selection 7-331
Selecting the Face, Edge, Vertex, or Object Behind 7-331
Selecting Objects and Surfaces That Lie Inside Other Objects 7-332
Selecting Objects by Variable 7-334
Assigning Coordinates to New Objects 7-334
Setting the Reference Point 7-335
Defining Cartesian Coordinates 7-335
Defining Cylindrical Coordinates 7-336
Defining Spherical Coordinates 7-336
Using Absolute Coordinates 7-337
Using Relative Coordinates 7-338

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Choosing the Movement Mode 7-338
Moving the Cursor In Plane 7-339
Moving the Cursor Out of Plane 7-341
Moving the Cursor in 3D Space 7-342
Moving the Cursor Along the X-Axis 7-343
Moving the Cursor Along the Y-Axis 7-344
Moving the Cursor Along the Z-Axis 7-345
Measure Modes 7-345
Setting Coordinate Systems 7-349
Setting the Working Coordinate System 7-351
Creating a Relative Coordinate System 7-351
Creating an Offset Relative CS 7-352
Creating a Rotated Relative CS 7-354
Creating an Offset and Rotated Relative CS 7-356
Creating a Face Coordinate System 7-358
Automatically Creating Face Coordinate Systems 7-361
Creating an Object Coordinate System 7-361
Creating an Offset Object CS 7-362
Creating a Rotated Object CS 7-363
Creating an Object CS that is Both Offset and Rotated 7-364
Move CS to End Command for History Tree 7-365
Modifying Coordinate Systems 7-366
Expressing Cartesian, Cylindrical, or Spherical Coordinates 7-367
Deleting Coordinate Systems 7-368
Setting the Drawing Plane 7-368
Cleaning Up Model History 7-369
8 - Modifying the 3D Model View 8-1
Change the View 8-1

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Show or Hide Design Objects ... 8-2
Change Object Visualization ... 8-2
Change the Background ... 8-3
View Options: 3D UI Options ... 8-3
Panning the View ... 8-8
Zooming in and Out ... 8-9
Zoom In and Zoom Out Commands ... 8-11
Zoom Previous Command ... 8-12
Zooming into a Rectangular Area ... 8-14
Zooming in or out Using a Mouse Wheel ... 8-15
Zoom to a Selected Boundary or Excitation ... 8-15
Modifying the View Orientation ... 8-16
Apply an Orientation to the Current View ... 8-18
Using the Orientation Gadget ... 8-20
Controlling Visibility of the Orientation Gadget ... 8-24
Rotating or Spinning the View ... 8-25
Changing the Model View with Alt+Double-Click Areas ... 8-32
Applying a Default View Orientation ... 8-33
Applying a New View Orientation ... 8-35
Removing an Orientation ... 8-36
Fitting Objects in the View Window ... 8-37
Fitting All Objects in a View Window ... 8-38
Fitting Selected Objects in a View Window ... 8-38
Hiding Objects from View ... 8-39
Showing Objects ... 8-41
Showing Only Selected Objects in All or Active Views ... 8-44
Active View Visibility Dialog Box ... 8-46
Rendering Objects as Wireframes or Solids ... 8-48

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Default View Rendering Mode ... 8-48
Setting the Enhanced Display of Material Color and Transparency ... 8-49
Setting the Curved Surface Visualization ... 8-51
Using Clip Planes ... 8-52
Modifying the Lighting ... 8-56
Setting the Projection View ... 8-58
Setting the Background Color ... 8-59
Modifying the Coordinate System Axes View ... 8-60
Showing or Hiding the Axes ... 8-61
Show the Axes for Selected Objects ... 8-61
Enlarging or Shrinking the Axes ... 8-62
Showing or Hiding the Triad Axes ... 8-62
Choosing Grid Settings ... 8-62
Setting the Grid Type ... 8-63
Setting the Grid Style ... 8-64
Setting the Grid Density and Spacing ... 8-65
Setting the Grid's Visibility ... 8-67
Setting the Grid Plane ... 8-67
9 - Maxwell User-Defined Primitives ... 9-1
Maxwell User-Defined Primitives ... 9-1
Launching a UDP from Maxwell Desktop ... 9-1
RMxprt UDPs ... 9-4
UDPs for Bands and Regions ... 9-4
Band UDP ... 9-4
Creating a Band ... 9-6
Creating an Outer Region ... 9-7
Creating a Tool ... 9-8
Creating 3D Independent and Dependent Boundaries ... 9-13

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating Core Regions ... 9-15
Creating an Inner Core Region ... 9-15
Creating an Outer Core Region ... 9-16
UDPs for Slot Cores and Coils ... 9-16
SlotCore UDP ... 9-17
Creating an Inner or Outer Slot Core ... 9-20
Slots with Different Fillet and Bottom Types ... 9-21
VentSlotCore UDP ... 9-22
Creating an Inner or Outer Vent Slot Core ... 9-25
RacetrackSlotCore UDP ... 9-25
Creating an Outer Racetrack Slot Core ... 9-27
NonSalientPoleCore UDP ... 9-27
Creating an Inner Non-Salient Pole Core ... 9-30
Creating a Non-Salient Pole Core with a Q-Axis Damper ... 9-30
Creating a Non-Salient Pole Core with a Uniform-Distributed Damper ... 9-36
LapCoil UDP ... 9-39
Creating a Whole Lap Winding ... 9-42
Creating a Single Lap Coil and its Terminals ... 9-42
Creating a 3D Open Insulation ... 9-43
Creating a Bended Coil ... 9-44
Creating a Tooth Coil ... 9-45
ConCoil UDP ... 9-47
Creating a Single Concentric Coil and its Terminals ... 9-52
WaveCoil UDP ... 9-52
Creating a Whole Wave Winding ... 9-55
Creating a Single Wave Coil ... 9-55
SquirrelCage UDP ... 9-55
Creating a Whole Squirrel-Cage Winding ... 9-58

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating Bars Only (Squirrel Cage) ... 9-58
Creating Two Rings Only ... 9-58
DoubleCage UDP ... 9-59
Creating a Whole Double-Cage Winding ... 9-61
Creating Bars Only (Double Cage) ... 9-62
Creating Two Rings Only (Double Cage) ... 9-62
CupCoil UDP ... 9-63
Creating a Coil and its Terminals (CupCoil) ... 9-65
HairpinCoil UDP ... 9-68
Parameter Considerations ... 9-69
Creating Closed Three-Phase Hairpin Windings (HairpinCoil) ... 9-72
Creating One Phase Half-Coiled Windings (HairpinCoil) ... 9-74
Creating One Phase Whole-Coiled Windings (HairpinCoil) ... 9-76
Creating Two Hairpin Coils Connected with End-Tip and Leads ... 9-77
Creating a Terminal for a Hairpin Coil ... 9-79
UDPs for Pole Cores and Coils ... 9-80
SalientPoleCore UDP ... 9-81
Creating an Inner or Outer Salient Pole Core ... 9-85
Creating a Coil and its Terminals (Salient Pole) ... 9-86
Creating a Damper (Salient Pole) ... 9-86
Creating Poles or the Yoke ... 9-88
Creating Inner Poles with Shoe Slope ... 9-90
DCMCore UDP ... 9-91
Creating an Outer DC Machine Core and All Coils ... 9-100
Creating Main Poles, Frame and Commutating Poles Separately ... 9-101
Creating a Shunt Coil and its Terminals ... 9-101
Creating a Series Coil and its Terminals ... 9-101
Creating a Commutating Coil and its Terminals ... 9-102

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ShadedPoleCore2 UDP 9-102
Creating an Outer Shaded Pole Core 9-105
Creating a Coil and its Terminals (Shaded Pole Core) 9-106
Creating a Jointed or a Separated Shaded Conductor and Its Terminals 9-108
Creating a Core with Poly Holes 9-110
UnivMCore UDP 9-116
Creating an Outer Universal Motor Core 9-118
Creating All Coils (UnivMCore) 9-119
Creating a Coil and its Terminal (UnivMCore) 9-120
SRMCore UDP 9-121
Creating an Inner or Outer Core for Switched Reluctance Machines 9-124
Creating Core and All Coils (SRMCore) 9-125
Creating a Coil and its Terminal (SRMCore) 9-126
Creating Core with Unparallel Pole Edges (SRMCore) 9-127
SRM Core with Slot Corner Fillet and V-Shape Bottom 9-128
SynRMCore UDP 9-129
Creating an Inner Core for a Synchronous Reluctance Machine 9-135
Creating a SynRMCore with Different Barriers 9-136
ClawPoleCore UDP 9-143
Creating an Inner Claw Pole Core 9-145
Creating a Coil and its Terminal (Claw Pole) 9-145
Creating a Magnet (Claw Pole) 9-147
PMCore UDP 9-147
Creating an Inner or Outer PM Core 9-149
Creating All Magnets (PMCore) 9-151
Creating a Magnet (PM Core) 9-151
Creating an Insert Surface PM Core 9-152
IPMCore UDP 9-153

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating an Inner IPM Core ... 9-155
Creating All Magnets (IPM Core) ... 9-155
Creating One-Pole Magnets ... 9-156
PMDamperCore UDP ... 9-157
Creating an Inner PM Damper Core ... 9-160
Creating a Damper (PM Damper Core) ... 9-161
Creating All Magnets (PM Damper Core) ... 9-162
Creating a Magnet (PM Damper Core) ... 9-163
StepMCore UDP ... 9-166
Parameter Definitions ... 9-167
Creating an Inner or Outer Core for a Variable Reluctance Stepper Motor ... 9-170
Creating an Inner Rotor Core for a Hybrid Stepper Motor ... 9-172
Creating a Coil and Coil Terminals for a Stepper Motor ... 9-173
Creating a Stator Core with Non-Uniform Pole Pitch ... 9-174
Square Yoke UDP ... 9-175
Parameter Definitions ... 9-176
Creating a Stator Yoke to Match Stator Poles of Stepper Motors ... 9-178
UPMCore UDP ... 9-180
Creating a U-Type IPM Core ... 9-181
Creating Magnets (UPM Core) ... 9-182
Creating PM Ducts (UPM Core) ... 9-184
VPMCore UDP ... 9-185
Creating a V-Type IPM Core ... 9-187
Creating Magnets (VPM Core) ... 9-189
Creating PM Ducts (VPM Core) ... 9-190
VentCore UDP ... 9-191
Creating a VentCore ... 9-195
Creating a Shaft (VentCore) ... 9-197

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating Vent Ducts (VentCore) ... 9-198
Creating Notches (VentCore) ... 9-198
UDPs for Other Cores and Coils ... 9-199
DiskSlotCore UDP ... 9-199
Creating an Upper or Lower Axial-FluxSlot Core (DiskSlot Core) ... 9-203
Creating a Squirrel-Cage Winding ... 9-204
DiskCoil UDP ... 9-204
Creating a Whole Winding (Upper or Lower) ... 9-207
Creating a Lap Coil and its Terminals ... 9-208
DiskPMCore UDP ... 9-208
Creating an Upper or Lower Axial-Flux PM Core ... 9-210
Creating All Magnets (Disk PM Core) ... 9-211
Creating One Magnet Only (Disk PM Core) ... 9-211
LinearMCore UDP ... 9-212
Creating a Linear Machine Core ... 9-214
Creating a Core and All Coils (Linear Machine Core) ... 9-215
Creating One Coil Only (Linear Machine Core) ... 9-215
TransCore UDP ... 9-216
Creating a Three-Leg Transformer Core ... 9-219
Creating Three Legs Only ... 9-219
Creating Two Yokes Only ... 9-220
TransCoil UDP ... 9-220
Creating Coils for All Three Legs ... 9-223
Creating One-Leg Coil Only ... 9-223
TransTwist UDP ... 9-224
Creating a Core and All Twist Coils ... 9-230
Creating a Twist Wire and its Terminals ... 9-231
Creating a Twist Insulator of All Coils ... 9-232

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UDPs for Other Machine Parts ... 9-233
PressPlate UDP ... 9-234
Housing UDP ... 9-237
Housing FinType ... 9-241
Housing DuctType ... 9-242
EndCap UDP ... 9-242
Helix UDPs ... 9-243
PolygonHelix UDP ... 9-243
Creating a Right-Handed or Left-Handed Polygon Helix ... 9-248
RectHelix UDP ... 9-249
Creating a Right-Handed or Left-Handed Rectangle Helix ... 9-251
Information about Coding UDPs ... 9-252
Intrinsic Head Files ... 9-252
Head File for Intrinsic Structure Definition ... 9-252
Head File for Intrinsic Function Declaration ... 9-254
Code Implementation ... 9-263
Define UDP Parameters and Default Values ... 9-263
Function Inside Band UDP ... 9-263
Initialize Parameters and Check Validity ... 9-264
Parameter Validation Functions ... 9-265
GetParameters() ... 9-265
PutParameters() ... 9-265
ValidateParameters() ... 9-266
Implement Main Function to Create Primitive ... 9-267
Map Parameters between Different Versions ... 9-269
Source Code Files ... 9-270
UDPBase.h ... 9-270
UDPBase.cpp ... 9-273

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Band.h ... 9-283
Band.cpp ... 9-284
10 - Assigning Boundaries and Excitations for 3D Designs ... 10-1
Selecting Objects and Faces in the Geometry ... 10-2
Defining Boundary Conditions ... 10-2
Defining Excitations ... 10-3
Viewing Boundaries and Excitations ... 10-4
Setting the Visibility for 3D Boundaries and Excitations ... 10-4
Functional Boundaries and Excitations ... 10-5
Assigning a Thin Layer Boundary ... 10-5
Setting Eddy Effects and Displacement Current in 3D ... 10-6
Displacement Current for the 3D AC Magnetic A-Phi ... 10-7
Setting Core Loss for Transient and AC Magnetic Solvers ... 10-7
Specifying a Custom Core Loss ... 10-8
Required Files ... 10-8
Examples and Templates ... 10-9
Defining the coreloss_user.data File ... 10-10
Writing Your Own Formulations ... 10-13
Using Your Control Program with a Maxwell Design ... 10-14
Frequently Asked Questions ... 10-15
Permanent Magnet Field Excitations for 2D and 3D Transient and Magnetostatic Solvers ... 10-16
Setting Demagnetization/Magnetization Computations for Source Designs ... 10-20
General Procedure for Setting Demagnetization/Magnetization Computations for Source Designs ... 10-21
Using the Hysteresis Model-Based Demagnetization/Magnetization Approach ... 10-22
Using the Hysteresis Model-Based Magnetization Approach ... 10-23
Classic Magnetization Approach ... 10-24

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Hysteresis Model-Based Magnetization Approach 10-25
First Design: Set up a Hysteresis-Based Magnetization Computation Design (source design) 10-26
Third Design: Set up the Target Design 10-30
Showing and Verifying Conduction Paths 10-31
Recalculating Conduction Paths 10-32
Modifying Boundary Conditions and Excitations 10-32
Duplicating Boundaries and Excitations 10-32
Showing and Hiding Boundaries and Excitations 10-33
Showing and Hiding Boundaries and Excitations in Every View Window 10-33
Setting Default Values for Boundaries and Excitations 10-33
Reassigning Excitations 10-34
Deleting All Boundaries and/or Excitations 10-36
Reprioritizing Boundaries 10-36
Viewing and Editing Boundary or Excitation Properties 10-37
Reviewing Boundaries and Excitations in the Solver View 10-37
Set Default Boundary/Excitation Base Names 10-38
Excitation Name Ordering Conventions 10-39
Magnetostatic Boundaries and Excitations 10-39
Magnetostatic Boundaries 10-40
Default Boundary Conditions for a Magnetostatic Solver 10-41
Assigning a Magnetic H Field for a Magnetostatic Solver 10-42
Assigning an Integrated Zero Tangential H Field Boundary 10-43
Assigning an Insulating Boundary for a Magnetostatic Solver 10-43
Assigning a Symmetry Boundary Condition for a Magnetostatic Solver 10-44
Assigning an Independent Boundary for a Magnetostatic Solver 10-44
Assigning a Dependent Boundary for a Magnetostatic Solver 10-45
Assigning a Thin Layer (Air Gap) Boundary for a Magnetic Solver 10-47

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Getting Better Convergence with the Thin Layer Model 10-49
Assigning a Resistive Sheet Boundary for the 3D Magnetostatic Solver 10-49
Resistive Sheet Output 10-54
Magnetostatic Excitations 10-55
Assigning a Voltage Excitation for a Magnetostatic Solver 10-56
Assigning a Voltage Drop Excitation for a Magnetostatic Solver 10-56
Assigning a Current Density Excitation for a Magnetostatic Solver 10-57
Assigning a Current Excitation for a Magnetostatic Solver 10-58
Assigning a Winding Setup for a Magnetostatic Solver 10-58
Assigning a Coil Terminal for a Magnetostatic Solver 10-61
Electrostatic Boundaries and Excitations 10-62
Electrostatic Boundaries 10-62
Default Boundary Conditions for an Electrostatic Solver 10-63
Assigning a Symmetry Boundary Condition for an Electrostatic Solver 10-63
Assigning an Independent Boundary for an Electrostatic Solver 10-64
Assigning a Dependent Boundary for an Electrostatic Solver 10-64
Assigning an Insulating Boundary for an Electrostatic Solver 10-66
Electrostatic Excitations 10-66
Assigning a Voltage Excitation for an Electrostatic Solver 10-67
Assigning a Charge Excitation for an Electrostatic Solver 10-67
Assigning a Floating Excitation for an Electrostatic Solver 10-68
Assigning a Volume Charge Density Excitation for an Electrostatic Solver 10-68
3D AC Conduction Boundaries and Excitations 10-69
3D AC Conduction Boundaries 10-69
Default Boundary Conditions for a 3D AC Conduction Solver 10-70
Assigning a Symmetry Boundary Condition for a 3D AC Conduction Solver 10-70
Assigning an Independent Boundary for a 3D AC Conduction Solver 10-71
Assigning a Dependent Boundary for a 3D AC Conduction Solver 10-72

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning an Insulating Boundary for the 3D AC Conduction Solver 10-73
Assigning a Thin Layer Boundary for a 3D AC Conduction Solver 10-74
3D AC Conduction Excitations 10-75
Assigning a Voltage Excitation for a 3D AC Conduction Solver 10-76
Assigning a Current Excitation for a 3D AC Conduction Solver 10-76
Assigning a Sink Excitation for a 3D AC Conduction Solver 10-77
DC Conduction Boundaries and Excitations 10-77
DC Conduction Boundaries 10-77
Default Boundary Conditions for a DC Conduction Solver 10-78
Assigning an Insulating Boundary for a DC Conduction Solver 10-78
Assigning a Symmetry Boundary Condition for a DC Conduction Solver 10-79
Assigning a Independent Boundary for a DC Conduction Solver 10-80
Assigning a Dependent Boundary for a DC Conduction Solver 10-81
Assigning a Thin Layer Boundary for a 3D DC Conduction Solver 10-82
DC Conduction Excitations 10-83
Assigning a Voltage Excitation for a DC Conduction Solver 10-84
Assigning a Current Excitation for a DC Conduction Solver 10-84
Assigning a Sink Excitation for a DC Conduction Solver 10-84
DC Conduction + Electrostatic Boundaries and Excitations 10-85
AC Magnetic T-Omega Boundaries and Excitations 10-85
AC Magnetic T-Omega Boundaries 10-86
Default Boundary Conditions 10-87
Assigning a Magnetic H Field 10-88
Assigning an Integrated Zero Tangential H Field Boundary 10-89
Assigning an Insulating Boundary 10-89
Assigning a Symmetry Boundary 10-90
Assigning an Independent Boundary 10-90
Assigning a Dependent Boundary 10-91

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning a Radiation Boundary 10-93
Assigning an Impedance Boundary 10-94
Impedance Boundary 10-99
When to Use Impedance Boundaries 10-100
Assigning a Resistive Sheet Boundary 10-101
Assigning a Thin Layer Boundary 10-104
Getting Better Convergence with the Thin Layer Model 10-105
AC Magnetic T-Omega Excitations 10-106
Assigning a Current Density Excitation 10-107
Assigning a Current Density Terminal Excitation 10-108
Assigning a Current Excitation 10-108
Assigning a Coil Terminal 10-109
Assigning a Winding Setup 10-110
Using AC Magnetic Excitation to Generate Harmonic Losses 10-112
Validation 10-116
External Circuit Setup 10-116
AC Magnetic A-Phi Boundaries and Excitations 10-121
AC Magnetic A-Phi Boundaries 10-121
AC Magnetic A-Phi Excitations 10-121
Assigning a Current Excitation 10-122
Excitation Models 10-123
Closed Loop Excitations 10-126
Assigning a Voltage Excitation 10-127
Excitation Models 10-128
Closed Loop Excitations 10-133
Transient T-Omega Boundaries and Excitations 10-135
Transient T-Omega Boundaries 10-136
Default Boundary Conditions 10-138

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning a Zero Tangential H Field Boundary 10-138
Assigning an Integrated Zero Tangential H Field Boundary 10-138
Assigning an Insulating Boundary 10-139
Assigning a Symmetry Boundary 10-139
Assigning a Independent Boundary 10-140
Assigning a Dependent Boundary 10-141
Assigning a Resistive Sheet Boundary 10-142
Assigning an Impedance Boundary 10-144
Assigning a Touching Boundary 10-149
Assigning a Thin Layer (Air Gap) Boundary for a 3D Transient Solver 10-150
Getting Better Convergence with the Thin Layer Model 10-152
Transient T-Omega Excitations 10-152
Assigning a Coil Terminal 10-153
Assigning a Winding Setup 10-154
Setting Up a Y Connection 10-157
Setting Up an SVPWM Excitation 10-158
Setting Up an External Circuit Connection for an External Winding Type 10-166
Using the AC_Model Component to Speed Up Analysis 10-169
Setting Up an External Circuit Using Simulink 10-171
Setting Up Motion for Transient Projects 10-176
Assigning a Band of Motion 10-178
Modifying Motion Setup Properties 10-182
Non-Cylindrical Rotational Motion 10-183
Reassigning a Band of Motion 10-184
Adding and Deleting Objects from a Band of Motion 10-185
Showing and Hiding Motion Objects 10-185
Setting the Visibility for Motion Objects 10-185
Modeling Maxwell Motion Eccentricity 10-185

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using the Maxwell Eccentricity Wizard 10-186
For 2D Designs 10-188
For 3D Designs 10-190
Transient A-Phi Formulation Boundaries and Excitations 10-191
Transient A-Phi Formulation Boundaries 10-192
Default Boundary Conditions for a Transient A-Phi Solver 10-193
Assigning a Flux Tangential Boundary for a Transient A-Phi Solver 10-193
Transient A-Phi Formulation Excitations 10-194
Electric Transient Solver Boundaries and Excitations 10-195
Electric Transient Excitations 10-195
Assigning a Voltage Excitation for an Electric Transient Solver 10-196
Assigning a Charge Excitation for an Electric Transient Solver 10-196
Assigning a Floating Excitation for an Electric Transient Solver 10-196
Assigning a Volume Charge Density Excitation for an Electric Transient Solver 10-197
Assigning a Current Excitation for an Electric Transient Solver 10-197
Assigning a Sink Excitation for an Electric Transient Solver 10-197
11 - Assigning Boundaries and Excitations for 2D Designs 11-1
Selecting Objects and Faces in the 2D Geometry 11-1
Defining Boundary Conditions in 2D 11-1
Defining 2D Excitations 11-2
Viewing 2D Boundaries and Excitations 11-3
Setting the Visibility for 2D Boundaries and Excitations 11-3
Functional Boundaries and Excitations in 2D 11-4
Setting Eddy Effects and Displacement Current in 2D 11-4
Modifying 2D Boundary Conditions and Excitations 11-5
Duplicating 2D Boundaries and Excitations 11-5
Showing and Hiding Boundaries and Excitations in 2D 11-6
Showing and Hiding 2D Boundaries and Excitations in Every View Window 11-6

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Default Values for 2D Boundaries and Excitations ... 11-6
Reassigning 2D Boundaries ... 11-7
Reassigning 2D Excitations ... 11-7
Deleting All 2D Boundaries and/or Excitations ... 11-8
Viewing and Editing 2D Boundary or Excitation Properties ... 11-8
2D Magnetostatic Boundaries and Excitations ... 11-8
2D Magnetostatic Boundaries ... 11-8
Default Boundary Conditions ... 11-9
Assigning a Vector Potential ... 11-10
Assigning a Symmetry Boundary Condition ... 11-10
Assigning a Balloon Boundary Condition ... 11-11
Assigning an Independent Boundary ... 11-11
Assigning a Dependent Boundary ... 11-11
2D Magnetostatic Excitations ... 11-12
Assigning a Current Density Excitation ... 11-12
Assigning a Current Excitation ... 11-13
Assigning a Winding Setup ... 11-14
Assigning a Coil ... 11-16
2D Electrostatic Boundaries and Excitations ... 11-16
2D Electrostatic Boundaries ... 11-17
Default Boundary Conditions ... 11-18
Assigning a Symmetry Boundary Condition ... 11-18
Assigning a Balloon Boundary Condition ... 11-18
Assigning a Independent Boundary ... 11-19
Assigning a Dependent Boundary ... 11-19
2D Electrostatic Excitations ... 11-20
Assigning a Voltage Excitation ... 11-20
Assigning a Charge Excitation ... 11-21

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning a Floating Excitation ... 11-21
Assigning a Charge Density Excitation ... 11-22
2D AC Conduction Boundaries and Excitations ... 11-22
2D AC Conduction Boundaries ... 11-22
Default Boundary Conditions ... 11-23
Assigning a Symmetry Boundary Condition ... 11-23
Assigning a Balloon Boundary Condition ... 11-24
Assigning an Independent Boundary ... 11-24
Assigning a Dependent Boundary ... 11-25
2D AC Conduction Excitations ... 11-25
Assigning a Voltage Excitation ... 11-26
2D DC Conduction Boundaries and Excitations ... 11-26
2D DC Conduction Boundaries ... 11-26
Default Boundary Conditions ... 11-27
Assigning a Symmetry Boundary Condition ... 11-28
Assigning a Balloon Boundary Condition ... 11-28
Assigning a Resistance Boundary Condition ... 11-28
Assigning an Independent Boundary ... 11-29
Assigning a Dependent Boundary ... 11-30
2D DC Conduction Excitations ... 11-31
Assigning a Voltage Excitation ... 11-31
2D AC Magnetic Boundaries and Excitations ... 11-31
AC Magnetic Boundaries ... 11-32
Default Boundary Conditions ... 11-32
Assigning a Vector Potential ... 11-33
Assigning an Impedance Boundary ... 11-33
2D Impedance Boundary ... 11-34
When to Use Impedance Boundaries in 2D ... 11-35

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning a Symmetry Boundary ... 11-36
Assigning a Balloon Boundary Condition ... 11-36
Assigning an Independent Boundary ... 11-36
Assigning a Dependent Boundary ... 11-37
AC Magnetic Excitations ... 11-38
Assigning a Current Excitation ... 11-38
Assigning a Parallel Current Excitation ... 11-39
Assigning a Current Density Excitation ... 11-40
Assigning a Coil ... 11-40
Assigning an End Connection ... 11-41
Assigning a Winding Setup ... 11-42
External Circuit Setup ... 11-44
2D Transient Boundaries and Excitations ... 11-48
2D Transient Boundaries ... 11-48
Default Boundary Conditions ... 11-49
Assigning a Vector Potential ... 11-49
Assigning a Symmetry Boundary ... 11-49
Assigning a Balloon Boundary Condition ... 11-50
Assigning an Independent Boundary ... 11-50
Assigning a Dependent Boundary ... 11-51
2D Transient Excitations ... 11-51
Assigning a Current Excitation ... 11-52
Assigning a Current Density Excitation ... 11-52
Assigning a Coil ... 11-53
Assigning an End Connection ... 11-54
Assigning a Winding Setup ... 11-54
Setting Up a Y Connection ... 11-56
Setting Up an SVPWM Excitation ... 11-57

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Up an External Circuit ... 11-66
Setting Up an External Circuit Using Simulink ... 11-70
Setting Up Motion for Transient Projects ... 11-70
Assigning a Band of Motion ... 11-72
Modifying Motion Setup Properties ... 11-76
Non-Cylindrical Rotational Motion ... 11-77
Reassigning a Band of Motion ... 11-78
Adding and Deleting Objects from a Band of Motion ... 11-79
Showing and Hiding Motion Objects ... 11-79
Setting the Visibility for Motion Objects ... 11-79
Modeling Maxwell Motion Eccentricity ... 11-79
Using the Maxwell Eccentricity Wizard ... 11-80
For 2D Designs ... 11-82
For 3D Designs ... 11-84
12 - Defining Mesh Operations ... 12-1
Specifying Initial Mesh Settings ... 12-3
Length Based Mesh Refinement ... 12-9
Assigning Length-Based Mesh Refinement on Object Faces ... 12-11
Assigning Length-Based Mesh Refinement Inside Objects ... 12-12
Assigning Skin Depth-Based Mesh Refinement on Object Faces ... 12-13
Visualization for Skin Depth Refinement Mesh Results ... 12-18
Assigning Edge Cut Mesh Operation ... 12-20
Assigning Cylindrical Gap Treatment ... 12-21
3D Clone Mesh Generation ... 12-22
Band Mapping Angle ... 12-28
TAU2D: Band Mapping Angle Mesh (Clone+Symmetry) ... 12-30
Layers in Band Mesh (Maxwell 3D Magnetostatic and Transient) ... 12-35
Example ... 12-35

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Considerations and Limitations ... 12-37
Clone Mesh Density ... 12-38
Skin Depth Layers Setting for 2D ... 12-40
Modifying Surface Approximation Settings ... 12-42
Surface Approximation Settings ... 12-46
Guidelines for Modifying Surface Approximation Settings ... 12-47
Applying Curvilinear Elements ... 12-47
Modifying the Model Resolution ... 12-48
Use Model Resolution Length ... 12-50
Reverting to the Initial Mesh ... 12-53
Generating Mesh without Solving ... 12-53
TAU Flex Meshing ... 12-54
Steps for Using TAU Flex ... 12-55
TAU Flex Warning Messages ... 12-58
Reviewing TAU Flex Mesh Warnings ... 12-58
Addressing TAU Flex Mesh Issues ... 12-60
Specifying Surface Priority for TAU Mesher ... 12-60
TAU 2D Mesh Generation ... 12-61
TAU 2D Mesh Generation for Rotational Models ... 12-62
TAU 2D Clone Mesh Generation ... 12-62
TAU 2D Rotational Sweep Mesh Generation ... 12-64
Maxwell 2D Symmetry Mesh ... 12-65
TAU 2D Regular Mesh Generation (Non-Rotational Models) ... 12-67
TAU 2D Skin Depth Layer Mesh Generation ... 12-69
TAU 2D Mesh Assembly ... 12-70
TAU2D: Mesh Density Control Over Selected Objects ... 12-71
Layer Mesh Generation for 3D Rotational Models ... 12-77
Edge Cut Layer Mesh Generation ... 12-78

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rotational Layer Mesh Generation 12-79
Edge Cut + Rotational Layer Mesh Generation 12-82
Changing Properties and Reassigning Mesh Operations 12-83
Deleting Previously Assigned Mesh Refinements 12-83
Viewing List of Mesh Operations 12-84
Reassigning a Mesh Operation 12-85
Viewing Mesh Properties 12-86
13 - Definition Libraries 13-1
Working with Definition Libraries 13-1
System, User, and Personal Library Directories 13-1
Changing the Locations of Libraries 13-2
Library and Project Definitions 13-2
Updating Project Definitions from Library Definitions 13-3
Exporting and Importing Definition Archives 13-3
Exporting Definition Archives 13-3
Importing Definition Archives 13-6
Managing Library Files 13-8
Library Search Precedence 13-9
Component Creation Sequence 13-9
Using the Component Editor 13-11
Creating and Editing Components 13-11
Editing an Existing Component 13-12
Creating a New Component 13-12
The Edit Component Window 13-12
Components General Tab 13-13
Components Miscellaneous Tab 13-14
Components Terminals Tab 13-16
Components Solver On Demand Tab 13-17

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HFSS 3D Layout as Solver On Demand 13-20
Edit Component Properties window 13-21
Parameter Statistics Display 13-23
Reserved Component Parameters 13-24
Component CosimDefinition Property 13-25
Netlist String Syntax 13-25
Using the Symbol Editor 13-28
Creating a New Symbol 13-29
Editing an Existing Symbol 13-30
Symbol Editor Operations 13-30
Symbol Draw Menu 13-31
Symbol Menu 13-37
Import File (Symbol) 13-39
Editing Pin Properties 13-40
Edit Symbol Pin Locations 13-41
Bus Pin Editor 13-41
Pin Location Editor 13-43
Pin List 13-44
Symbol Property Display Setup 13-46
Symbol Grid Setup window 13-47
Using the Footprint Editor 13-48
Opening the Footprint Editor 13-49
Creating a New Footprint 13-49
Editing an Existing Footprint 13-50
Load a Footprint into the Current Project 13-50
Footprint Editor Operations 13-50
Footprint Edit Menu 13-51
Footprint Shortcut Menu 13-52

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Footprint View Menu 13-53
Footprint Draw Menu 13-53
Defining Footprint Handles 13-57
Using the Padstack Editor 13-58
Creating a New Padstack 13-58
Editing a Padstack 13-59
Editing a Polygon Pad 13-64
Pad Behavior in Vias and Pins 13-66
Editing Via and Pin Padstacks in Layout 13-67
Initial Padstack Definition Layers 13-68
Pin and Net Padstack Operations 13-69
Using the Models Editor 13-71
Using the Material Editor 13-72
Using Vendor Components 13-72
Using RF Vendor Library Components 13-73
Encrypted Libraries 13-74
Creating and Managing Encrypted Libraries 13-74
Password Assignment Types 13-74
Setting Up Passwords and Encrypted Libraries 13-76
Generate a New Password Protected Resource 13-77
Using the Library Editor 13-84
Starting the Library Editor 13-84
Edit Libraries Window 13-85
Exporting Hierarchical Components 13-87
14 - Adding Solution Parameters 14-1
Assigning a Force Parameter 14-1
Assigning a Torque Parameter 14-2
Assigning a Matrix 14-3

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning a Reduce Matrix for 2D and 3D AC Magnetic Designs 14-5
Reassigning Parameters 14-7
Viewing Parameter List 14-8
Deleting All Parameters 14-8
Showing and Hiding Solution Parameters 14-8
Setting the Visibility for Parameters 14-9
Changing the Parameter Setup 14-9
15 - Specifying Solution Settings 15-1
Available Tabs by Solution Type 15-1
Analysis Parameters 15-3
Setting Analysis Parameters for Non-Transient Solutions 15-5
Defining Settings on the General Tab for Non-Transient Solutions 15-5
Setup Name for Non-Transient Solutions 15-5
Adaptive Setup for Non-Transient Solutions 15-5
Parameter Settings 15-6
Defining Settings on the Convergence Tab for Non-Transient Solutions 15-6
Defining Expressions for Non-Transient Solutions 15-7
Defining Settings on the Solver Tab for Non-Transient Solutions 15-10
Magnetostatic Solver Settings 15-10
Permeability Options for Magnetostatic Solutions 15-13
Using the Advanced Permeability Option 15-16
Advanced Permeability Option File Formats 15-19
Per Mesh Element Centroid Location 15-21
AC Magnetic Solver Settings 15-21
Including DC Fields in a 3D AC Magnetic T-Omega Solution 15-25
Electrostatic Solver Settings 15-36
AC Conduction Solver Settings 15-36
DC Conduction Solver Settings 15-38

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Import Mesh for Non-Transient Solutions 15-39
Setup Link for Non-Transient Solutions 15-41
Enabling the Iterative Solver 15-44
Defining Settings on the Frequency Sweep Tab for AC Magnetic and AC Conduction Solutions 15-44
Defining Settings on the Defaults Tab for Non-Transient Solutions 15-46
Setting Analysis Parameters for Transient Solutions 15-46
Defining Settings on the General Tab for Transient Solutions 15-47
Setup Name for Transient Solutions 15-47
Transient Setup (2D) 15-47
Adaptive Time Step 15-47
Non-Adaptive Time Step 15-48
Transient Setup (3D) 15-48
Defining Settings on the Save Fields Tab for Transient Solutions 15-48
Save Fields on Object/Face Named Selection 15-50
Defining Settings on the Advanced Tab for Transient Solutions 15-53
Using a Control Program in Maxwell 2D and 3D Transient Solutions 15-53
Activating a Control Program in Transient Solutions 15-55
Control Program File Formats in Transient Solutions 15-56
User Control File Format 15-57
User Control Fields Extraction Format 15-59
Solution and Previous Control File Formats 15-63
Start/Continue from a Previously Solved Setup 15-64
Import Mesh for Transient Solutions 15-65
Setup Link for Transient Solutions 15-65
Using an AC Magnetic Solution as the Initial Condition for a Transient Solution 15-67
Defining Expressions for Transient Solutions 15-69
Speeding Up a Maxwell 2D Parametric Transient Design that Uses Expression 15-71

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cache
Defining Settings on the Solver Tab for Transient Solutions 15-74
Automatic Detection of Reaching Steady State for Transient Simulations 15-81
Time Decomposition Method for Maxwell Transient Designs 15-84
Using TDM 15-85
Limitations to Using TDM 15-86
Defining Settings for Output Variables for Transient Solutions 15-87
Defining Settings on the Defaults Tab for Transient Solutions 15-88
Setting Analysis Parameters for Electric Transient Solutions 15-88
Defining Settings on the General Tab for Electric Transient Solutions 15-88
Setup Name for Electric Transient Solutions 15-88
Timestep Setup for Electric Transient Solutions 15-89
Defining Expressions for Electric Transient Solutions 15-89
Defining Settings on the Solver Tab for Electric Transient Solutions 15-91
Setup Link for Electric Transient Solutions 15-91
Defining Settings on the Defaults Tab for Electric Transient Solutions 15-92
Resetting Analysis to Time Zero 15-92
Changing Memory Settings 15-92
Specifying the Desired RAM Limit 15-93
Specifying the Hard Memory Limit 15-94
Setting View Context for Transient Solutions 15-96
Speed and Position Display Settings 15-97
Renaming a Solution Setup 15-97
Copying a Solution Setup 15-97
Disabling and Enabling an Analysis Setup 15-98
Setup Link Window 15-99
Clearing Linked Data 15-99
Setting the Report Updating During Analysis Options 15-100

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16 - Using a 3D Layout Component in Maxwell 101
Adding a 3D Layout Component 103
Using the Layout Component's Reference CS for Placement 110
Editing the Placement of a Layout Component 114
Inserting Additional Layout Component Instances 119
Editing a Layout Component Definition 120
Visualization for Layout Components 127
Using Show Layout for Layout Components 128
Object Attributes Control of Visualization for Layout Components 129
Visualizing Terminals 134
Visualization for RLC boundaries and Component Groups Categorized as Circuit Elements 138
Parameter Support for Layout Components 140
Show Nets for Layout Components 142
Replacing or Editing Layout Components 152
Edit/Replace Procedure 153
Meshing with a Layout Component 154
Mesh Linking 156
Simulating a 3D Layout Component with an A-Phi Solver 157
Creating a Layout Component in HFSS 3D Layout 157
Importing a Layout Component to Maxwell 3D 160
Limitations for the 3D Layout Component Feature in the Maxwell Solver 169
Flex PCB Component Support in Maxwell 170
Field Plots on Layout Components Layer or Net 179
Using Fields Calculator with Layout Component in a Design 184
17 - Coupling with the Maxwell Application 17-1
Maxwell to Ansys Thermal Coupling 17-2
Maxwell-Specific Considerations 17-2

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Maxwell 2D Coupling Considerations 17-3
Convergence of Maxwell-Mechanical Two-Way Thermal Coupling 17-3
Coupling Maxwell Designs with Ansys Thermal via Workbench 17-5
Maxwell to Ansys Stress Feedback Coupling 17-12
Maximum Force Density (AC) Components Calculation 17-13
Coupling Maxwell Designs with Ansys Structural via Workbench 17-13
Coupling Maxwell with Ansys Thermal and Structural via Workbench 17-16
Example 1: One Iteration 17-17
Example 2: Manual Iteration 17-20
Example 3: Revert Maxwell Solution 17-22
Example 4: Only Stress Feedback 17-24
Maxwell Multiple RPM Harmonic Force Coupling 17-27
Harmonic Force Coupling for Inverter-Fed Machines 17-29
Generate Inverter-Fed Harmonic Force Results 17-29
Limitations 17-34
Harmonic Force Interpolation in Frequency-Speed Plane 17-36
For Electrical Machines 17-36
For Inverter-Fed Motors 17-39
Maxwell Harmonic Force Calculations and Coupling 17-41
Maxwell to Ansys Object-Based Harmonic Force Coupling 17-41
Inverter Feedback Tab 17-47
Force Densities in Maxwell 17-47
Force Density Calculation in Maxwell 17-48
Harmonic Force Calculation from Maxwell 17-48
Window Function Options for DFT of Harmonic Forces 17-50
Harmonic Force Calculation in Maxwell 2D with Skew Model 17-52
Skew Model Setup 17-52
Harmonic Force Calculation Requirements for Maxwell 2D Slice Model 17-54

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Transient Report of Object-Based Harmonic Force Components 17-54
Maxwell to Ansys Element-Based Harmonic Force Coupling 17-57
Element-Based Harmonic Force Calculations and Coupling for Transient Designs 17-57
Element-Based Harmonic Force Coupling for AC Magnetic Designs 17-62
Export Transient/Harmonic Force from Maxwell 17-63
Maxwell to LMS Virtual.Lab Coupling 17-65
Maxwell to Romax Coupling 17-67
Maxwell to Third-Party Coupling by UNV File 17-69
Maxwell One-Way Coupling with Ansys Motion 17-71
Maxwell One-Way Coupling with Ansys Mechanical 17-73
Maxwell Partial Model Coupling with Workbench 17-76
[Beta] Element-Based Multiphysics Coupling for 2D Multislice Skew Model 17-79
Near Field Coupling between Maxwell and HFSS 17-83
Magnetic Bias Source Coupling between Maxwell and HFSS 17-84
Coupling Maxwell with Ansys Motion (Stand-Alone) 17-85
Running an Ansys Motion-Maxwell Cosimulation on Linux 17-91
Coupling Maxwell with Ansys Motion in Workbench 17-93
Including Transient Data in Maxwell-Motion Workbench Coupling 17-103
Coupling Maxwell to a Twin Builder Component 17-107
Coupling Maxwell to a Twin Builder Excitation Component 17-110
Coupling Magnetostatic and AC Magnetic Designs with Ansys Fluent 17-112
Sample Input.txt File Format 17-113
Determining the Zone ID in Ansys Fluent 17-114
Determining the Face ID in Ansys Fluent 17-114
Determining the Object ID in Maxwell 17-114
Magnetostriction in Maxwell Magnetostatic to Workbench Mechanical Coupling 17-115
One-Way Coupling 17-115

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Two-Way Coupling 17-115
Magnetostriction Stress Coupling Between Maxwell 2D/3D Transient and Workbench Mechanical 17-116
Launching Motor-CAD from Ansys Electronics Desktop 17-121
System Coupling Analyses Using Maxwell 17-122
Supported Capabilities and Limitations 17-123
Using HPC Distribution Settings for Coupled Analyses 17-124
Variables Available for System Coupling 17-124
System Coupling Related Settings 17-125
System Settings 17-125
Maxwell Settings 17-125
Creating a System Coupling Setup 17-126
Generating a System Coupling Participant Setup File 17-127
Licensing Considerations when using System Coupling 17-127
Modeling an Electric Arc in a Static Device 17-128
Maxwell Setup 17-129
Fluent Setup 17-133
System Coupling Setup 17-135
Modeling an Electric Arc in a Device with Motion 17-142
Motion Setup 17-143
Maxwell Setup 17-144
Fluent Setup 17-150
System Coupling Setup 17-152
18 - High Performance Computing 18-1
Remote Analysis 18-1
Prerequisites for Remote and Distributed Analysis 18-2
Configuring the Local Machine to Solve Remotely 18-3
Remote Analysis Options 18-3

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Running Remote Analysis 18-4
Troubleshooting 18-5
Remote Solve Node = Windows 18-7
Remote Solve Node = Linux 18-8
Distributed Analysis 18-9
Beta Feature: Parallel Component Mesh Adapt for 3D Component Array 18-10
Configure a Distributed Analysis 18-10
Distributed Analysis Configuration 18-10
Distributed Analysis Configuration - Machines Tab 18-13
Manually Adding a Machine 18-14
Importing a Machine List 18-14
Machine List Details 18-15
Testing Machines 18-16
Distributed Analysis Configuration - Job Distribution Tab 18-16
Distribution Levels 18-19
Single-level Distributions 18-19
Two-Level Distributions 18-20
Distributed Analysis Configuration - Options Tab 18-21
Relation to Batchoptions 18-22
Adding Configurations or Accepting Edits 18-23
Enabling GPU 18-23
Selecting Optimal Configurations for Distributed Analysis 18-26
Two-Level Distribution Guidelines 18-27
Distributed Direct Solver Guidelines for HPC Configuration 18-28
Distributed Iterative Solver Guidelines for HPC Configuration 18-28
Large Scale DSO for Parametric Analysis 18-29
Prerequisites for Large Scale DSO 18-30
Job Management Interface for Large Scale DSO 18-31

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Large Scale DSO Command Line Syntax 18-37
Large Scale DSO Job Outputs 18-41
Output Location and Organization 18-41
CSV File Contents 18-41
Postprocessing Large Scale DSO Dataset Solutions 18-42
Importing a Large Scale DSO Solution 18-42
Viewing Imported Solutions 18-43
Creating a Dataset Report 18-44
Cloning a Dataset Solution 18-44
Large Scale DSO Results Database Organization 18-44
Large Scale DSO Job Monitoring 18-45
Additional Resources for Large Scale DSO Monitoring 18-45
Large Scale DSO Deployment/Configuration 18-46
Large Scale DSO Tutorial Example 18-48
Prepare the Model for Large Scale DSO Analysis 18-48
Submit the Large Scale DSO Job: Examples 18-53
Large Scale DSO Example: Post Process the Results 18-55
Large Scale DSO Theory 18-57
Large Scale DSO Known Issues/Troubleshooting 18-60
Node Order 18-60
Parallel Task Limitation for LS-DSO Parametric Variations 18-61
Job Restart 18-61
Linux-Only Issues 18-62
Interactive Scheduler Jobs 18-62
Specifying Options for Interactive Scheduler Jobs 18-63
Batchoptions for Interactive Scheduler Jobs 18-63
DSO Configuration for Interactive Scheduler Jobs 18-65
Analysis Configuration - Machines Tab 18-66

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Analysis Configuration - Job Distribution Tab ... 18-67
Analysis Configuration - Options Tab ... 18-68
Design Type Options for Interactive Scheduler Jobs ... 18-69
Distribution Command Line Options ... 18-70
High Performance Computing (HPC) Integration ... 18-72
Scheduler Terminology ... 18-73
What a Scheduler Does ... 18-74
Configuring Electronics Installation for HPC ... 18-75
Firewall Configuration ... 18-76
Installation Directory Examples ... 18-76
MPI Vendor and MPI Version ... 18-76
Integration with Microsoft Windows® HPC Scheduler ... 18-78
Submitting and Monitoring Ansys EM HPC Jobs ... 18-79
Submitting and Monitoring Jobs for Windows HPC ... 18-81
Windows® HPC Job Templates ... 18-99
Selecting Computation Resource Units (Job Unit Type) ... 18-99
Windows® HPC Non-Exclusive Jobs ... 18-100
Windows® HPC Job Credentials ... 18-112
Integration with Grid Engine (GE) ... 18-113
GE Job Management ... 18-113
Installation of Ansys EM Tools on GE ... 18-115
GE Commands for Information about Jobs and Cluster Configuration ... 18-117
Monitoring GE Serial and Parallel Batch Jobs ... 18-120
Ansys EM Desktop -monitor Command Line Option for SGE ... 18-120
Example SGE qsub Command Lines ... 18-120
Recommended Practices for GE Clusters ... 18-122
Issue with qrsh (SGE) ... 18-127
Issue with MainWin Core Services for SGE ... 18-127

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Integration with Platform's Load Sharing Facility (LSF) 18-129
LSF Job Management 18-129
LSF Job Submission Guidelines 18-130
Integration of Ansys EM Products with LSF 18-130
Installing Ansys EM Tools on LSF Cluster 18-131
Using the bsub Command to Submit Batch Jobs 18-133
bsub Arguments 18-134
Example LSF bsub Command Lines (Linux Only) 18-135
Serial Job 18-135
Serial Job Requiring a Minimum of 4GB 18-135
Multi-processing Job using 4 Cores 18-135
Distributed Processing Job using 4 Engines 18-135
Distributed Processing and Multi-processing Job using 4 Cores, with 2 Cores for Multi-processing 18-136
Shell Script (~/projects/OptimTee.csh): 18-136
Monitoring LSF Batch Jobs 18-136
Terminating LSF Batch Jobs 18-137
LSF Known Issues and Workarounds 18-138
LSF Troubleshooting 18-138
Integration with PBS (Portable Batch System) 18-139
PBS Job Management 18-139
Non Standard Installations for PBS 18-140
PBS Limitations 18-140
Submitting Ansys EM PBS Batch Jobs 18-142
Monitoring PBS Batch Jobs 18-143
qsub Arguments 18-144
Example PBS qsub Command Lines 18-145
Integration with SLURM Scheduler (Linux Utility) 18-147

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Configuration of the generic scheduler with SLURM 18-147
Environment Variables 18-148
Configuration File 18-149
Persistence Mode 18-149
Remote RSM Service Support 18-150
Using the generic scheduler with SLURM 18-150
Using MPI Tight Integration 18-151
Using the Command Line to Submit HPC Jobs 18-151
Distributed Jobs 18-151
Serial Job on a Single Processor 18-152
Distributed Job Using Four Processors 18-152
Multiprocessing Job Using Four Cores 18-152
Distributed Analysis and Multi-Processing in the Same Job 18-153
Integrating Ansys EM Tools with Third-Party Schedulers 18-153
Introduction 18-153
Serial Jobs 18-153
Parallel Jobs 18-154
Common Requirements for Running Jobs 18-154
Using a Shared Library (Linux) or a DLL (Microsoft Windows) 18-155
Build Information for Scheduler Proxy Library 18-155
Implementation Details for Custom Scheduler Integration 18-156
IsProductLaunchedInYourEnvironment 18-157
GetTempDirectory 18-157
GetMachineListAvailableForDistribution 18-158
GetMessageStringToRegisterForSigTerm 18-159
LaunchProcess 18-160
GetUseRsmForEngineLaunch 18-161
GetThisJobID 18-162

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

GetSchedulerDisplayName 18-163
Extending Scheduler Integration with Tight-Integration for MPI 18-164
Support for Open MPI on Linux [Beta] 18-167
Scheduler Proxy Interfaces 18-171
Testing Scheduler Integration 18-177
Testing IsProductLaunchedInYourEnvironment 18-177
Testing GetSchedulerDisplayName and GetThisJobID 18-178
Testing GetTempDirectory 18-178
Testing GetMachineListAvailableForDistribution 18-178
Testing LaunchProcess 18-178
Testing GetUseRsmForEngineLaunch 18-179
Troubleshooting Custom Scheduler Integration 18-179
None of the Proxy Functions are Called 18-180
Troubleshooting IsProductLaunchedInYourEnvironment Function 18-180
Troubleshooting GetSchedulerDisplayName 18-180
Troubleshooting GetThisJobID 18-181
Troubleshooting GetTempDirectory 18-181
Troubleshooting GetMachineListAvailableForDistribution 18-181
Troubleshooting LaunchProcess 18-182
Troubleshooting GetUseRsmForEngineLaunch 18-182
Using an IronPython Program for Integration with a Scheduler 18-183
GetName [IronPython] 18-184
GetDescription [IronPython] 18-185
IsProductLaunchedInYourEnvironment [IronPython] 18-185
GetSchedulerDisplayName [IronPython] 18-186
GetThisJobID [IronPython] 18-186
GetUseRsmForEngineLaunch [IronPython] 18-187
GetTempDirectory [IronPython] 18-187

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

GetMessageStringToRegisterForSigTerm [IronPython] 18-188
GetMachineListAvailableForDistribution [IronPython] 18-188
LaunchProcess [IronPython] 18-189
Ansys HPC Platform Services Support for HPC Job Management 18-190
Prerequisites for Using Ansys HPC Platform Services with Ansys Electronics Desktop 18-192
Job Management UI for Ansys HPC Platform Services 18-192
Ansys HPC Platform Services-Specific Settings: 18-192
Compute Resources Tab 18-192
Multistep Submissions 18-198
Aborting an Analysis 18-202
Cluster Specific Considerations 18-202
Option Settings for Ansys HPC Platform Services 18-203
Selecting a Scheduler 18-203
Submitting a Job 18-205
Using Advanced Job Submission Options 18-210
Batchoptions 18-211
Environment Variables 18-212
Batch Extract 18-213
Customize Job Submission 18-214
Using the Command Line to Submit HPC Jobs 18-214
Distributed Jobs 18-214
Serial Job on a Single Processor 18-214
Distributed Job Using Four Processors 18-215
Multiprocessing Job Using Four Cores 18-215
Distributed Analysis and Multi-Processing in the Same Job 18-215
Job Import and Export 18-216
Scripting 18-216

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Multi-Step Job Submission 18-216
Windows to Linux Job Submission 18-221
Prerequisites for Job Submission 18-222
Prerequisites for Job Monitoring 18-223
Supported Schedulers 18-223
Job Submission Scripting 18-229
Monitoring Jobs 18-230
Monitor Job Window 18-230
Monitoring Ansys Cloud Burst Jobs 18-233
Monitoring Large Scale DSO Jobs 18-233
Web Client for Batch Solve Monitoring and Reporting 18-234
Aborting an Analysis 18-240
GPU Acceleration 18-241
License Options 18-241
Enable/Disable GPU Acceleration from the Ansys Electronics Desktop Solver User Interface 18-241
Integration with Ansys Remote Simulation Management (RSM) 18-246
Changing the AnsoftRSMService Listening Port 18-261
AnsoftRSMService Configuration 18-262
Ansoft Electromagnetics Desktop Configuration 18-263
Running HPC Diagnostics 18-265
Site-Specific Diagnostics Job 18-268
Changing Solution Priority for System Resources 18-270
desktopproxy Utility to Abort or Cleanstop a BatchSolve 18-271
19 - Ansys Workbench Integration Overview 19-1
Integrating Ansys EM Suite with Ansys Workbench 19-1
Workbench Data Integration Overview 19-3
Adding New Analysis Systems 19-5

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Importing Existing Ansys EM Projects into Workbench 19-7
Editing Ansys EM Models in Workbench 19-8
Analyzing Ansys EM Models in Workbench 19-9
Performing Parameter Studies in Workbench 19-11
Scripting in Workbench 19-15
Ansys EM - Ansys Multiphysics Coupling 19-16
Multiphysics Coupling on Workbench with Ansys Thermal 19-16
Multiphysics Coupling on Workbench with Ansys Structural 19-17
Multiphysics Coupling between Ansys EM Field Systems on Workbench 19-18
Ansys EM CAD Integration through Workbench 19-19
CAD Integration Functionality 19-22
CAD Integration and Geometry Sharing 19-24
Bi-Directional CAD Integration 19-27
CAD Integration Model Edits 19-28
Multiple Geometry Links for CAD Integration 19-29
Creating Dynamic Links to EDT Designs 19-30
Healing with CAD Integration 19-31
Important Geometry Options for CAD Integration 19-31
Ansys EM to Ansys Geometry Transfer 19-33
Material Assignment Transfer 19-34
Geometry Transfer through Ansys DesignModeler (DM) 19-34
Workbench Material Data Transfer 19-35
Detailed Behavior 19-35
User-Defined Model (UDM) 19-36
UDM Compared to User-Defined Primitives 19-37
Insert UDM Command on Draw Menu 19-39
UDM Properties 19-39
UDM Parameters 19-40

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UDM Part Edits 19-40
Library of Models for CAD Integration 19-41
Feedback Iterator 19-42
The Feedback Iterator System 19-44
Feedback Iterator in Use 19-45
Feedback Iterator Component Properties 19-46
Feedback Iterator GUI Operations 19-48
Resetting the Feedback Iterator 19-49
Resetting via the Ansys EM User Interface 19-50
Resetting via a Workbench Journal Script 19-51
Callback Interface 19-52
Callback and State API 19-52
Utility Functions 19-53
Output/Debugging Functions 19-54
Example Scenarios for Feedback Iterator 19-54
Setting up Iteration with Feedback Iterator 19-54
Breaking Iteration Control 19-55
Starting an Iterative Update 19-56
Running a Single Iteration 19-57
Interrupting an Iterative Loop 19-57
Resuming an Interrupted Iterative Loop 19-57
Modifying any of the Systems Involved in Iteration (Coupled Clients) 19-58
Maxwell in a Thermal Deformation Use Case 19-58
Iterating to Convergence 19-58
20 - Scripting 20-1
21 - Exporting Equivalent Circuit Data 21-1
Exporting a Circuit From a Parametric Solution 21-1
Types of Equivalent Circuit Models 21-3

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Exporting a Circuit From an Imported Table ... 21-4
Setting Up Current Variables ... 21-6
Current Variables Represent Ampere-Turns ... 21-6
Current Variables Represent Amperes ... 21-7
Setting Inputs and Outputs in the Table Window ... 21-7
Setting Coil and Mechanical Terminals ... 21-9
22 - Running Simulations ... 22-1
Local Analysis ... 22-2
Solving a Single Setup or Sweep ... 22-3
Running More than One Simulation ... 22-4
Monitoring Queued Simulations ... 22-5
Monitoring the Solution Process ... 22-5
Changing a Solution Priority for System Resources ... 22-7
Aborting an Analysis ... 22-7
Re-solving after Modifying a Design ... 22-8
Re-solving after Ansys Workbench Thermal Feedback ... 22-8
23 - Network Data Explorer ... 23-1
Network Data Explorer Overview ... 23-2
NDE Ribbon ... 23-3
Network Data Selection Pane ... 23-4
Cell and Frequency Selection Pane ... 23-4
Data View Pane ... 23-5
Loading Data into Network Data Explorer ... 23-5
Exporting Data from Network Data Explorer ... 23-6
Exporting SYZ Data ... 23-6
Exporting Macro Model ... 23-9
Comparing Original S-Parameters with Exported S-Parameters ... 23-16
Creating an NPort Model ... 23-16

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scripting for Network Data Explorer 23-17
Network Data Explorer Ribbon 23-17
Network Data Explorer Data Sources 23-18
Network Data Explorer Display Format 23-18
Network Data Explorer Display Full Port Names 23-19
Network Data Explorer Save or Reset Default Settings 23-20
Smoothing 23-20
Cell Filtering 23-21
Changing Port Properties and Reducing Matrix Size 23-22
Displaying Mixed-Mode Parameters using Differential Pairs 23-23
Reset All Port Properties 23-25
Data View Pane Context Menus 23-25
Multiple Frequency Statistics 23-25
Highlight Min/Max 23-26
Select Transpose 23-26
Color Legend Attributes 23-27
Matrix Entries Plot Menu 23-28
Exploring Network Data and Modifying the Display 23-32
Viewing a Matrix Table 23-33
Viewing a Color-Coded Matrix Plot 23-34
Displaying a Cell Graph Across All Frequencies 23-35
Displaying Matrix Statistics by Frequency 23-36
Displaying Individual Statistics for All Frequencies 23-37
Creating a Statistics Plot 23-38
Comparing Network Data 23-39
Causality Checking and Plots 23-40
Multithreading 23-47
Multithreading Technical Notes 23-48

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24 - Postprocessing and Generating Reports **24-1**
Viewing Solution Data 24-1
Viewing Convergence Data 24-2
Viewing the Number of Completed Passes 24-4
Viewing Output Variable Convergence 24-5
Plotting Convergence Data 24-5
Viewing a Solution Profile 24-5
Viewing Matrix Data 24-10
Exporting a Circuit Model 24-13
Coupling Coefficient Matrix 24-15
AC Loss Effects of Litz Wire on Impedance Matrix 24-15
Viewing Parameter Data 24-16
Viewing Mesh Statistics 24-17
Viewing Winding Results for AC Magnetic T-Omega Solutions 24-19
Viewing End Connection Results (2D AC Magnetic) 24-19
Viewing Loss Results (AC Magnetic) 24-20
Deleting Solution Data 24-21
Creating Animations 24-22
Creating Geometry Animations 24-24
Creating Field Plot Animations 24-27
Creating Parametric Animations 24-28
Controlling the Animation's Display 24-32
Exporting Animations 24-33
Exporting Ensight from the Select Animation Dialog Box 24-33
Exporting from the Animation Control Dialog Box 24-35
Terminal Current/Voltage Output 24-37
Creating a Quick Report 24-38
Quick Report Categories 24-39

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating a New Report 24-40
Using Families Tab for Reports 24-46
Filtering Quantity Selections for the Reporter 24-49
Creating a Report from a Report Data File 24-50
Modifying Reports 24-50
Modify Report: Using the Edit Sweep Dialog Box 24-55
Zooming and Fitting Reports 24-56
Modifying the Background Properties of a Report 24-56
Modifying the Legend in a Report 24-59
Creating Custom Report Templates and Defaults 24-62
Selecting the Report Type 24-63
Selecting the Display Type 24-64
Creating 2D Rectangular Plots 24-66
Creating a 2D Rectangular Stacked Plot 24-67
Creating 3D Rectangular Plots 24-71
Creating 3D Rectangular Bar Plots 24-77
Plotting a Quantity Versus Adaptive Pass 24-87
Creating Data Tables 24-88
Creating Rectangular Contour Plots 24-90
Delta Markers in 2D Reports 24-92
Working with Traces 24-92
Editing Trace Properties 24-94
Editing the Display Properties of Traces 24-95
Adding Data Markers to Traces 24-98
Y Markers in Stacked XY Plots 24-100
Discarding Report Values Below a Specified Threshold 24-107
Adding Characteristics to a Trace 24-108
Adding a Recently Used Trace Characteristic 24-108

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adding a Trace Characteristic from Favorites ... 24-108
Adding Trace Characteristics to your Favorites ... 24-109
Adding Characteristics Using the Add Trace Characteristics Window ... 24-111
Removing All Trace Characteristics ... 24-113
Removing Traces ... 24-113
Copy and Paste of Report and Trace Definitions ... 24-113
Copy and Paste of Report and Trace Data ... 24-114
Limit Lines in Cartesian Plots ... 24-115
Sweeping a Variable in a Report ... 24-122
Sweeping Values Across a Distance ... 24-123
Selecting a Function for a Plot ... 24-123
Selecting Solution Quantities to Plot ... 24-130
Selecting a Field Quantity to Plot ... 24-132
Plotting Imported Solution Data ... 24-132
Postprocessed Quantities ... 24-132
Per-Winding Loss Outputs for Maxwell 2D/3D Transient and AC Magnetic Designs ... 24-133
Terminal Current/Voltage Output ... 24-135
Setting a Range Function ... 24-135
Perform FFT on a Report ... 24-137
FFT Window Functions ... 24-137
Apply FFT to Report Functions ... 24-139
Perform TDR on Report ... 24-141
TDR Windowing Functions ... 24-142
Spectral Domain Plots in Transient (FFT) ... 24-144
Time Domain Plots in AC Magnetic (IFFT) ... 24-144
Specifying Output Variables ... 24-145
Building an Expression Using Existing Quantities ... 24-148

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Function List for Output Variables 24-149
Defining Settings for Output Variables for Transient Solutions 24-149
Report Data 24-150
Opening Reports 24-150
Updating Reports (Post-Processing Data) 24-151
Deleting Reports 24-151
Exporting Reports 24-151
Importing 2D Plot Data 24-152
Report File Formats 24-153
Creating Custom Report Templates and Defaults 24-154
Exporting Reports as Graphics 24-155
Copying Reports to the Clipboard as Images 24-157
Plotting Field Overlays 24-158
Plotting Standard Fields Overlays 24-159
Plotting Standard Field Quantities 24-163
Force Density Quantities 24-165
Loss Quantities 24-167
Demagnetization Coefficient Plots 24-169
Demag_Coef Plot over the Geometry of Nonlinear Permanent Magnets: 24-170
Demag_Coef Curve as a Function of Time at Specific Locations: 24-171
Demagnetization Percentage Statistics Curve as a Function of Time over All Magnet Volumes 24-173
Plotting Time Averaged Fields Overlays 24-174
Plotting Noise Vibration Overlays (Transient designs only) 24-177
Plotting Field Line Trace Overlays for E Fields 24-182
Inception Voltage Evaluation 24-183
Named Expressions 24-187
Plotting Derived Field Quantities 24-188

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating 2D Reports from Named Expressions 24-189
Creating Scalar Field Plots 24-190
Working with Scalar Field Plot Markers 24-190
Adding a Field Plot Marker 24-191
Deleting a Field Plot Marker 24-192
Exporting a Field Plot Marker Table 24-193
Clearing All Field Plot Markers 24-193
Editing Field Plot Markers 24-193
Showing Field Plot Markers 24-195
Creating Vector Field Plots 24-195
Modifying Field Plots 24-198
Setting Field Overlay Attributes 24-200
Modifying Field Plot Colors 24-211
Color Key Settings 24-216
Modifying the Field Plot Scale 24-217
Modifying Vector Field Plot Arrows 24-220
Setting the Mesh Visibility on Field Plots 24-221
Modifying Scalar Field Plot Isovalues 24-222
Modifying Scalar Field Plot Shading 24-226
Mapping Scalar Field Plot Transparency to Field Values 24-228
Modifying Markers on Point Plots 24-229
Viewing Data Markers 24-229
Modifying Line Plots 24-230
Setting a Plot's Visibility 24-231
Saving a Field Overlay Plot 24-231
Opening a Field Overlay Plot 24-231
Deleting a Field Overlay Plot 24-232
Setting Field Overlay Plot Defaults 24-232

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Renaming a Plot 24-233
Exporting Field Plots 24-233
Field Plot Data Export Format (*.aedtplt) 24-235
File Format Description 24-237
Using the Fields Calculator 24-239
Support Expression Input in Fields Calculator 24-240
Scripting Support 24-240
Preliminary Considerations for Using the Fields Calculator 24-240
Field Convergence and Accuracy 24-240
Fast Sweep and Dispersive Models 24-241
Inputs/Excitations 24-241
Units 24-242
Opening the Fields Calculator 24-242
Context Area 24-243
The Calculator Stack 24-245
Stack Commands 24-246
Pop 24-247
RlUp 24-247
RlDn 24-247
Exch 24-247
Clear 24-247
Undo 24-247
Algebraic Mode 24-247
Names 24-249
Geometry 24-252
Registers 24-253
Enlarging the Register Display Area 24-256
Units of Measure 24-256

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Input Commands 24-256
Quantity Command 24-258
Geometry Command 24-258
Background Geometry 24-262
Constant Command 24-263
Number Command 24-264
Function Command 24-265
Geometry Settings 24-266
Reading an External Input 24-267
General Commands 24-267
Steps for Calculating the Complex Vector Electric Field 24-270
Scalar Commands 24-271
Vec? Command 24-272
1/x (Inverse) Command 24-272
Pow Command 24-272
Square Root Command 24-273
Trig 24-273
d/d? (Partial Derivative) Command 24-273
Integral Command 24-274
Min Command 24-274
Max Command 24-275
Gradient Command 24-275
Ln Command 24-275
Log Command 24-276
Std Command 24-276
Vector Commands 24-276
Scal? Command 24-277
Matl Command 24-277

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mag Command 24-278
Dot Command 24-278
Cross Command 24-278
Divg Command 24-278
Curl Command 24-278
Tangent Command 24-279
Normal Command 24-279
Unit Vec Command 24-280
X Form 24-280
Output Commands 24-282
Value Command 24-282
Eval Command 24-284
Write Command 24-285
Export Command 24-287
Calculating Derived Field Quantities Using Named Expressions 24-292
Named Expression Library 24-294
Exiting the Fields Calculator 24-296
Plotting the Mesh 24-296
Example of Mesh Plots for an HFSS Design with Layout Components 24-300
Setting Mesh Plot Attributes 24-305
Export Layout Field Data 24-308
Export Grid Max Field Data 24-309
Export Via Average Current Density 24-311
Export Via Current 24-313
Export Layer-Induced Eddy Current 24-315
User-Defined Outputs (UDOs) 24-322
Named Probes and Properties in User-Defined Outputs 24-323
Computation of Traces Based UDO Calculations 24-325

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dimensions Reduction by UDO Calculations ... 24-325
Dynamic Probes ... 24-326
User-Defined Outputs: Python Script API ... 24-326
UDO Extension Implementation ... 24-327
Import Statements ... 24-327
UDOExtension Class ... 24-327
Validate ... 24-327
IUDOPluginExtension Abstract Class ... 24-328
GetUDSName ... 24-329
GetUDSDescription ... 24-329
GetUDSSweepNames ... 24-330
GetCategoryNames ... 24-330
GetQuantityNames ... 24-331
GetQuantityInfo ... 24-331
GetInputUDSParams ... 24-332
GetDynamicProbes ... 24-335
Compute ... 24-335
Data Types Used in UDO Python Scripts ... 24-338
Constants ... 24-338
Abstract Classes ... 24-338
IUDSInputData ... 24-338
GetDoubleProbeData ... 24-339
GetSweepsDataForProbe ... 24-339
GetComplexProbeData ... 24-340
GetSweepNamesForProbe ... 24-341
GetRequiredQuantities ... 24-341
GetVariableValues ... 24-342
GetInterpolationOrdersData ... 24-342

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

IUDSOutputData ... 24-343
SetSweepsData ... 24-343
SetDoubleQuantityData ... 24-344
SetComplexQuantityData ... 24-344
Working With Properties for UDO ... 24-345
IPropertyList Abstract Class ... 24-345
IProperty Abstract Class ... 24-346
INumberProperty Abstract Class ... 24-346
ITextProperty Abstract Class ... 24-346
IMenuProperty Abstract Class ... 24-347
Other Application-Specific Classes Used in Python Scripts ... 24-348
Constants Class ... 24-348
UDSProbeParams Class ... 24-348
UDSDynamicProbes Class ... 24-349
QuantityInfo Class ... 24-349
IProgressMonitor Abstract Class ... 24-350
SetTaskName ... 24-351
SetSubTaskName ... 24-351
BeginTask ... 24-352
SetTaskProgressPercentage ... 24-352
CheckForAbort ... 24-352
EndTask ... 24-353
Using .NET Collection Classes and Interfaces in Python Scripts ... 24-353
User-Defined Outputs: Messaging Methods ... 24-355
AddErrorMessage ... 24-360
AddInfoMessage ... 24-361
AddWarningMessage ... 24-361
User-Defined Outputs: Script Organization ... 24-361

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using Script Libraries ... 24-362
Using Additional .NET Assemblies ... 24-362
Toolkit ... 24-363
Toolkit Examples ... 24-363
Automotive Cable Bundle Toolkit ... 24-363
Oil-Gas Cable Bundle Toolkit ... 24-366
User-Defined Documents (UDDs) ... 24-367
Create User-Defined Document Inputs ... 24-368
UDD Document Creation and Display ... 24-371
Managing Documents Listed in the Project Window Under Results ... 24-372
Documents Folder's Context Menu ... 24-373
Document Folder's Property Window ... 24-373
Viewing UDDs with an HTML Web Browser ... 24-374
UDD Script Libraries ... 24-375
User-Defined Documents: Python Script API ... 24-376
Import Statements ... 24-376
Data Types Used in UDD Python Scripts ... 24-377
Constants ... 24-377
Abstract Classes ... 24-377
IProgressMonitor Abstract Class ... 24-377
SetTaskName ... 24-378
SetSubTaskName ... 24-378
BeginTask ... 24-379
SetTaskProgressPercentage ... 24-379
CheckForAbort ... 24-380
EndTask ... 24-380
UDDExtension Class ... 24-381
UDDInputData Class ... 24-381

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UDDInputBool ... 24-382
UDDInputDouble ... 24-382
UDDInputSolution ... 24-382
UDDInputText ... 24-383
UDDInputTrace ... 24-383
UDDInputParams Class ... 24-383
UDD Input Interfaces ... 24-384
Python Script to Define Document ... 24-386
Sample Script ... 24-388
Document Generator Interfaces ... 24-391
Chip Model Analyzer (CMA) ... 24-395
PinToPinUtility ... 24-396
25 - Optimetrics ... 25-1
Parametric Overview ... 25-2
Setting Up a Parametric Analysis ... 25-2
Adding a Variable Sweep Definition ... 25-4
Specifying Variable Values for a Sweep Definition ... 25-5
Synchronizing Variable Sweep Definitions ... 25-6
Modifying a Variable Sweep Definition Manually ... 25-6
Overriding a Variable's Current Value in a Parametric Setup ... 25-7
Specifying a Solution Setup for a Parametric Setup ... 25-8
Specifying the Solution Quantity to Evaluate for Parametric Analysis ... 25-8
Setting up Calculations for Optimetrics ... 25-9
Specifying a Solution Quantity's Calculation Range ... 25-11
Using Distributed Analysis ... 25-12
Solving a Parametric Analysis on Another Machine ... 25-12
Adding a Parametric Sweep from a File ... 25-13
Viewing Results for Parametric Solution Quantities ... 25-14

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plotting Solution Quantity Results vs. a Swept Variable 25-15
Optimization Overview 25-16
Choosing an Optimizer 25-16
Legacy Optimizers 25-18
Quasi Newton (Gradient) 25-18
Pattern Search (Search-Based) 25-21
Sequential Nonlinear Programming (Gradient) 25-23
Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete) 25-24
Genetic Algorithm (Random Search) 25-25
Multi-Objective Genetic Algorithm (MOGA) 25-26
MOGA Workflow 25-26
MOGA Steps 25-27
MOGA Steps to Generate a New Population 25-28
Convergence Criteria in MOGA-Based Multi-Objective Optimization 25-30
MATLAB Optimizer 25-31
Screening (Search Based) Optimizer Setup 25-37
Mixed-Integer Sequential Quadratic Programming (Gradient and Discrete) Optimizer Setup 25-40
Nonlinear Programming by Quadratic Lagrangian (Gradient) Optimizer Setup 25-46
Adaptive Multiple-Objective (Random Search) Optimizer Setup 25-50
Adaptive Single-Objective (Gradient) Optimizer Setup 25-56
Optimization Variables and the Design Space 25-61
Setting Up an Optimization Analysis 25-61
Quasi Newton (Gradient) Optimizer Setup 25-63
Pattern Search (Search-based) Optimizer Setup 25-65
Merit-Based Sequential Quadratic Programming(Gradient) Optimizer Setup 25-67
Sequential Nonlinear Programming (Gradient) Optimizer Setup 25-68
Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete) 25-70

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Optimizer Setup
Genetic Algorithm (Random Search) Optimizer Setup 25-72
MATLAB Optimizer Setup 25-73
Screening (Search Based) Optimizer Setup 25-75
Multi-Objective Genetic Algorithm(Random Search) Optimizer Setup 25-78
Nonlinear Programming by Quadratic Lagrangian (Gradient) Optimizer Setup 25-83
Mixed-Integer Sequential Quadratic Programming (Gradient and Discrete) Optimizer Setup 25-88
Adaptive Multiple-Objective (Random Search) Optimizer Setup 25-93
Adaptive Single-Objective (Gradient) Optimizer Setup 25-98
Setting the Maximum Iterations for an Optimization Analysis 25-103
Cost Function 25-104
Acceptable Cost 25-105
Cost Function Noise 25-105
Adding a Cost Function 25-105
Adding or Editing a Cost Function Calculation 25-108
Specifying a Solution Quantity for a Cost Function Goal 25-109
Setting the Calculation Range of a Cost Function Goal 25-110
Setting a Goal Value 25-111
Specifying a Single Goal Value 25-111
Specifying an Expression as a Goal Value 25-111
Specifying a Variable-Dependent Goal Value 25-112
Goal Weight 25-112
Modifying the Starting Variable Value for Optimization 25-114
Setting the Min. and Max. Variable Values for Optimization 25-114
Edit Calculation Range 25-115
Overriding the Min. and Max. Variable Values for a Single Optimization Setup 25-117
Changing the Min. and Max. Variable Values for Every Optimization Setup 25-118

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Step Size 25-118
Setting the Min. and Max. Step Sizes 25-119
Setting the Min and Max Focus 25-120
Equalizing the Influence of Different Optimization Variables 25-120
Setting the Min and Max Focus Values 25-121
Solving a Parametric Setup Before an Optimization 25-121
Solving a Parametric Setup During an Optimization 25-122
Automatically Updating a Variable's Value after Optimization 25-122
Changing the Cost Function Norm 25-122
Explanation of L1, L2, and Max Norms in Optimization 25-123
Example of a More Complex Cost Function 25-125
Advanced Genetic Algorithm Optimizer Options 25-126
Sensitivity Analysis Overview 25-129
Selecting a Primary Output 25-129
Setting Up a Sensitivity Analysis 25-129
Setting the Maximum Iterations Per Variable 25-131
Setting Up an Output Parameter 25-132
Specifying a Solution Quantity for an Output Parameter 25-132
Setting the Calculation Range of an Output Parameter 25-134
Modifying the Starting Variable Value for Sensitivity Analysis 25-134
Setting the Min. and Max. Variable Values 25-135
Overriding the Min. and Max. Variable Values for a Single Sensitivity Setup 25-135
Changing the Min. and Max. Variable Values for Every Sensitivity Setup 25-136
Setting the Initial Displacement 25-136
Solving a Parametric Setup before a Sensitivity Analysis 25-136
Solving a Parametric Setup during a Sensitivity Analysis 25-137
Performing Worst Case Analysis 25-137
Viewing Output Parameter Results for a Sensitivity Analysis 25-141

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plotting Output Parameter Results for a Sensitivity Analysis 25-142
Statistical Analysis Overview 25-142
Setting Up a Statistical Analysis 25-143
Setting the Maximum Iterations for a Statistical Analysis 25-144
Specifying an Initial Seed Value for a Statistical Analysis 25-145
Specifying the Solution Quantity to Evaluate for Statistical Analysis 25-145
Setting the Solution Quantity's Calculation Range 25-146
Setting the Distribution Criteria 25-147
Overriding the Distribution Criteria for a Single Statistical Setup 25-147
Changing the Distribution Criteria for Every Statistical Setup 25-148
Statistical Cutoff Probability 25-149
Edit Distribution 25-150
Modifying the Starting Variable Value for Statistical Analysis 25-152
Solving a Parametric Setup During a Statistical Analysis 25-153
Viewing Distribution Results for a Statistical Analysis 25-153
Plotting Distribution Results for a Statistical Analysis 25-154
Using Design of Experiments 25-154
Setting Up Design of Experiments in Ansys Electronics Desktop 25-157
Add the Design of Experiments Setup 25-157
Design of Experiments Tab 25-160
Optimal Space Filling Design (OSF) 25-163
Central Composite Design (CCD) 25-164
Box Behnken Design (CCD) 25-165
Custom DOE Type 25-166
Latin Hypercube Sampling 25-166
Table Tab for Design of Experiments 25-168
Response Surface Tab for Design of Experiments 25-169
Variables Tab for Design of Experiments 25-171

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Viewing Analysis Result for Design of Experiments 25-173
Min-Max Search View for Design of Experiments Result 25-176
Refinement Points Table 25-176
Performing a Manual Refinement 25-177
Response Points Table 25-177
Response Curve 25-178
Response Curve (2D Slices) 25-181
Verification Points Table 25-183
Goodness of Fit (Predicted vs Observed Chart) 25-184
Response Surface Results 25-185
Local Sensitivity Charts 25-188
Local Sensitivity Curves 25-189
Using the Fast Calculation-Update Algorithm 25-191
optiSLang Integration with Electronics Desktop 25-194
Prerequisites 25-195
optiSLang User Workflow 25-196
Parametrization for optiSLang Integration 25-198
Inputs 25-198
Outputs 25-199
Parameterization with optiSLang Analysis Goals in Mind 25-199
Results and Reports for optiSLang Integration 25-200
Creating an optiSLang Setup in AEDT 25-202
Setting up Calculations 25-203
Setting Options 25-205
Finishing Setup 25-206
Example optiSLang Setups 25-206
Example 1: Solving a User-created, Pre-existing optiSLang Setup 25-207

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example 2: Increasing Performance by Avoiding Wasteful Solution of the Nominal Design 25-209
Example 3: Avoiding Redundant Computations while Exploiting AEDT Features around Postprocessing Variables 25-210
Example 4: Parametrization of an AEDT Design for optiSLang Integration 25-210
Solving an optiSLang Setup 25-210
Considerations for Using the Analyze All Command 25-211
Considerations for Distributed Analysis 25-211
optiSLang Menu Options in AEDT 25-212
Viewing optiSLang Postprocessing Results 25-213
Tuning Overview 25-214
Tuning a Variable 25-214
Applying a Tuned State to a Design 25-217
Saving a Tuned State 25-217
Resetting Variable Values after Tuning 25-218
Reverting to a Saved Tuned State 25-218
Saving Field Solutions for Optimetrics Analyses 25-218
Saving Field Solutions for a Parametric Setup 25-219
Saving Field Solutions for an Optimization Setup 25-220
Saving Field Solutions for a Sensitivity Setup 25-221
Saving Field Solutions for a Tuning Analysis 25-222
Saving Field Solutions for a Statistical Setup 25-222
Saving Field Solutions for a Design of Experiments Setup 25-223
Copying Meshes in Optimetrics Sweeps 25-224
Adding an Expression in the Output Variables Window 25-224
Excluding a Variable from an Optimetrics Analysis 25-225
Modifying the Value of a Fixed Variable 25-225
Linear Constraints 25-226

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting a Linear Constraint 25-227
Modifying a Linear Constraint 25-228
Deleting a Linear Constraint 25-228
Running an Optimetrics Analysis 25-228
Viewing Analysis Results for Optimetrics Solutions 25-229
Viewing Solution Data for an Optimetrics Design Variation 25-230
Viewing an Optimetrics Solution's Profile Data 25-230
Viewing Cost Results for an Optimization Analysis 25-231
Plotting Cost Results for an Optimization Analysis 25-231
Link to DesignXplorer 25-231
26 - Getting Started with RMxprt 26-1
Getting Started with RMxprt 26-2
Creating a Project and Inserting a New RMxprt Design 26-2
RMxprt Files 26-3
Saving Project Notes in RMxprt 26-3
The RMxprt Desktop 26-4
RMxprt Title Bar 26-5
Working with the RMxprt Menu Bar 26-5
Working with the RMxprt Shortcut Menus 26-5
Working with the RMxprt Machine Editor Windows 26-6
Setting the RMxrpt Window View 26-7
Printing in RMxprt 26-7
Viewing RMxprt Design Details 26-7
Quick Start for RMxprt 26-8
RMxprt Example Part 1: Create a New Project 26-8
RMxprt Example Part 2: Select a Machine 26-8
RMxprt Example Part 3: Input Design Data 26-10
RMxprt Example Part 4: Analyze the Design 26-18

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RMxprt Example Part 5: Create Reports and View Output 26-19
RMxprt Example Part 6: Output Design Data for Further Analysis 26-24
Setting Up RMxprt Projects 26-27
Setting Up A Machine Model 26-29
Maxwell Model Wizard 26-31
Creating a Maxwell Model 26-34
Changing the Machine Type 26-35
Design Settings in RMxprt 26-35
Setting the Material Threshold in RMxprt 26-36
RMxprt Export Options 26-37
Setting User-Defined Data for a Design 26-37
Specifying Calibration Factors via User-Defined Data 26-38
Specifying Data Based on Design Template Files 26-39
Validating RMxprt Projects 26-44
Setting General Options in RMxprt 26-44
Setting RMxprt Options 26-45
Configuring a Wire Specification Library 26-46
Editing Wire Data 26-47
Setting RMxprt Machine Model Units 26-49
Edit AC Windings 26-50
Enable Winding Editor 26-51
Edit Winding Configuration 26-53
View Winding Connections 26-54
Working with the Slot Editor 26-55
The Slot Editor Window 26-56
Slot Editor Data Editing View 26-56
Editing Symmetric Slots 26-57
Editing Antisymmetric Slots 26-58

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Editing Half Slots 26-59
The New Slot Dialog Box 26-59
Slot Editor Graphical View 26-60
Editing Slot Segments 26-60
The Edit Slot Segment Dialog Box 26-61
Working with Materials in RMxprt 26-66
Material Library Management for RMxprt 26-66
Adding New Materials to an RMxprt Project 26-66
Soft-Magnetic Materials 26-67
Relative Permittivity for an RMxprt Material 26-68
Bulk Conductivity for an RMxprt Material 26-68
Dielectric Loss Tangent for RMxprt Material 26-69
Magnetic Loss Tangent for RMxprt Material 26-69
Magnetic Coercivity for Maxwell and RMxprt Materials 26-70
Core Loss Model for an RMxprt Material 26-70
Calculating Properties for Core Loss in RMxprt (BP Curve) 26-71
Electrical Steel Core Loss from a Single-Frequency Loss Curve 26-71
Computation of Core Loss from a Single-Frequency Loss Curve 26-72
Electrical Steel Core Loss from Multi-Frequency Loss Curves 26-73
Computation of Core Loss from Multi-Frequency Loss Curves 26-74
Power Ferrite Core Loss from Multi-Frequency Loss Curves 26-75
Computation of Power Ferrite Core Loss from Loss Curves 26-76
Mass Density for RMxprt Material 26-76
Composition for RMxprt Material 26-76
Permanent Magnet Materials in RMxprt 26-77
Compute Remanent Br from B-H Curve 26-77
Calculating the Properties for a Nonlinear Permanent Magnet in RMxprt 26-77
Calculating the Properties for a Linear Permanent Magnet 26-81

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using Demagnetization Curves 26-82
Hysteresis Loop 26-82
Demagnetization Curve 26-83
Recoil Lines 26-84
Recoil Magnetic Permeability 26-85
Inflection Point 26-86
Curve Fitting of Demagnetization Curves 26-86
Three-Parameter Curve Fitting 26-86
Four-Parameter Curve Fitting 26-88
Conductor Data 26-90
Editing Conductivity Properties in RMxprt 26-90
Specifying RMxprt Solution Settings 26-91
Generating a Custom Design Sheet for RMxprt 26-92
Key Words in Output Data for RMxprt 26-93
Creating RMxprt Customized Design Sheet Template 26-94
Design Template of Microsoft Excel Worksheet in Preferred Styles 26-95
Resort to Keywords in Design Output 26-96
Set Boundary for Data Imported into Worksheet for the RMxprt App 26-97
Insert Figures into Template for RMxprt 26-98
Use Different Languages for RMxprt Design Sheets 26-99
Postprocess Data for RMxprt 26-100
Running an RMxprt Simulation 26-102
Aborting RMxprt Analyses 26-102
Re-solving an RMxprt Problem 26-102
Postprocessing and Generating Reports in RMxprt 26-103
Viewing RMxprt Solution Data 26-103
Browse Solutions in RMxprt 26-105
Exporting a Simplorer Model or Customized Design Sheet 26-105

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating a Maxwell Design from RMxprt 26-106
Speed Adjustment to Achieve to Rated Output Power 26-107
Using the Initial Mechanical Transient Setup by RMxprt 26-108
Adjusting Speed Manually for a Constant Speed Setup 26-111
Determine Solution at RMxprt Setting Speed 26-111
First Iteration – Determine Solution at First adjusted Speed 26-111
Second Iteration – Determine Solution at Second Adjusted Speed 26-112
Creating an Icepak Design 26-113
Creating Reports in RMxprt 26-114
Modifying Reports in RMxprt 26-114
Opening All Reports in RMxprt 26-114
Deleting All Reports in RMxprt 26-115
Selecting the Display Type in RMxprt 26-115
Creating 2D Rectangular Plots in RMxprt 26-115
Creating 3D Rectangular Plots in RMxprt 26-117
Creating 3D Rectangular Bar Plots 26-123
Creating 3D Polar Plots in RMxprt 26-132
Creating Data Tables in RMxprt 26-137
Working with Traces in RMxprt 26-139
Removing Traces in RMxprt 26-139
Replacing Traces in RMxprt 26-139
Adding Blank Traces in RMxprt 26-140
Sweeping a Variable in a Report in RMxprt 26-140
Selecting a Function in RMxprt 26-141
Selecting a Parameter, Variable, or Quantity to Plot in RMxprt 26-147
Creating Quick Reports in RMxprt 26-148
RMxprt Quick Report Categories 26-148
Specifying RMxprt Winding Data 26-149

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Winding Type 26-149
Winding Types Available for Machines 26-150
Enable the Winding Editor 26-151
Edit Winding Configuration 26-153
Setting the Number of Winding Layers 26-153
Connecting and Disconnecting Windings 26-153
Polyphase Winding Editor 26-153
Enabling the Winding Editor 26-154
Windings Basic Terminology 26-156
Conductor 26-156
Strands 26-156
Coil 26-156
Coil Pitch 26-156
Coil Set 26-157
Winding 26-157
Polyphase AC Winding 26-157
Whole-Coiled Windings 26-157
Half-Coiled Windings 26-158
Single-Layer Windings 26-158
Chain-Type Windings 26-159
Lap-Type Windings 26-160
Half-Coiled Lap-Type Winding 26-160
Whole-Coiled Lap-Type Winding 26-160
Concentric-Type Windings 26-161
Half-Coiled Concentric-Type Winding 26-161
Whole-Coiled Concentric-Type Winding 26-162
Double-Layer Windings 26-162
Fractional-Pitch Winding 26-164

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Auto-Arrangement of AC Windings 26-165
Phase Spread 26-166
Coil Arrangement 26-167
Coil Connections 26-172
Connection of Double-Pole, Dual-Speed Windings 26-176
DC Windings 26-177
Wave Winding 26-178
Frog-Leg Winding 26-178
Virtual Slots 26-179
Equipotential Connectors 26-180
Equipotential Connector Class A of Simplex-Lap Winding 26-180
Equipotential Connector Class B of Multiplex-Wave Winding 26-180
Equipotential Connector of Multiplex-Lap Winding 26-180
Pole Windings 26-181
Limited Space for Wire Arrangement 26-182
Round Wire Winding 26-183
Cylinder Coil 26-184
Edgewise Coil 26-184
Pole Winding with Half Turns 26-185
Exporting Winding Data 26-186
RMxprt Machine Types 26-188
Three-Phase Induction Motors 26-188
Analysis Approach for Three-Phase Induction Motors 26-188
Defining a Three-Phase Induction Motor 26-190
Defining the General Data for a Three-Phase Induction Motor 26-191
General Data for Three-Phase Induction Motors 26-192
Defining the Stator Data for a Three-Phase Induction Motor 26-192
Stator Data for Three-Phase Induction Motors 26-193

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining the Stator Slots for a Three-Phase Induction Motor 26-194
Stator Slot Data for Three-Phase Induction Motors 26-194
Defining the Stator Windings for a Three-Phase Induction Motor 26-195
Stator Winding Data for Three-Phase Induction Motors 26-199
Winding Editor for a Three-Phase Induction Motor 26-201
Defining Different Size Wires for a Three-Phase Induction Motor 26-201
Stator Vent Data for Three-Phase Induction Motors 26-202
Defining the Rotor Data for a Three-Phase Induction Motor 26-203
Rotor Data for Three-Phase Induction Motors 26-204
Defining the Rotor Slots for a Three-Phase Induction Motor 26-205
Rotor Slot Data for Three-Phase Induction Motors 26-205
Defining the Rotor Winding for a Three-Phase Induction Motor 26-205
Rotor Winding for Three-Phase Induction Motors 26-206
Rotor Vent Data for Three-Phase Induction Motors 26-206
Defining the Shaft Data for a Three-Phase Induction Motor 26-207
Access Shaft Data for Three-Phase Induction Motors 26-207
Setting Up Analysis Parameters for a Three-Phase Induction Motor 26-208
Solution Data for Three-Phase Induction Motors 26-209
Single-Phase Induction Motors 26-209
Analysis Approach for Single-Phase Induction Motors 26-209
Defining a Single-Phase Induction Motor 26-211
Defining the General Data for a Single-Phase Induction Motor 26-212
General Data for Single-Phase Induction Motors 26-214
Defining the Stator Data for a Single-Phase Induction Motor 26-215
Stator Data for Single-Phase Induction Motors 26-216
Defining the Stator Slots for a Single-Phase Induction Motor 26-216
Stator Slot Data for Single-Phase Induction Motors 26-217
Defining the Stator Windings for a Single-Phase Induction Motor 26-218

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stator Winding Data for Single-Phase Induction Motors 26-224
Winding Editor for a Single-Phase Induction Motor 26-225
Defining Different Size Wires for a Single-Phase Induction Motor 26-225
Defining the Rotor Data for a Single-Phase Induction Motor 26-226
Rotor Data for Single-Phase Induction Motors 26-227
Defining the Rotor Slots for Single-Phase Induction Motors 26-227
Rotor Slot Data for Single-Phase Induction Motors 26-228
Defining the Rotor Windings for Single-Phase Induction Motors 26-228
Rotor Winding Data for Single-Phase Induction Motors 26-229
Defining the Shaft Data for a Single-Phase Induction Motor 26-229
Shaft Data for Single-Phase Induction Motors 26-229
Adding or Removing a Vent from a Single-Phase Induction Motor 26-230
Setting Up Analysis Parameters for a Single-Phase Induction Motor 26-230
Solution Data for Single-Phase Induction Motors 26-231
Adjust-Speed Synchronous Machines 26-232
Analysis Approach for Adjust-Speed Synchronous Machines 26-232
Stator Winding Connected to a Sinusoidal AC Source 26-232
Stator Winding Fed by a DC to AC Inverter 26-235
Defining an Adjustable-Speed Synchronous Machine 26-237
Defining the General Data for an Adjust-Speed Synchronous Machine 26-238
General Data for Adjust-Speed Synchronous Machines 26-239
Defining the Circuit Data for an Adjust-Speed Synchronous Machine 26-239
Circuit Data for Adjust-Speed Synchronous Machines 26-240
Defining the Stator Data for an Adjust-Speed Synchronous Machine 26-240
Defining the Stator Dimensions and Slots 26-241
Stator Data for Adjust-Speed Synchronous Machines 26-242
Stator Slot Data for Adjust-Speed Synchronous Machines 26-242

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining the Stator Windings and Conductors for an Adjust-Speed Synchronous Machine 26-243
Winding Editor for an Adjustable-Speed Synchronous Machine 26-250
Stator Winding Data for Adjust-Speed Synchronous Machines 26-251
Defining the Rotor Data for an Adjust-Speed Synchronous Machine 26-253
Rotor Data for Adjust-Speed Synchronous Machines 26-254
Defining the Rotor Pole for an Adjust-Speed Synchronous Machine 26-254
Rotor Pole Data for Adjust-Speed Synchronous Machines 26-255
Defining the Shaft Data for an Adjust-Speed Synchronous Machine 26-256
Shaft Data for Adjust-Speed Synchronous Machines 26-256
Setting Up Analysis Parameters for an Adjust-Speed Synchronous Machine 26-256
Solution Data for Adjust-Speed Synchronous Machines 26-258
Permanent-Magnet DC Motors 26-258
Analysis Approach for PMDC Motors 26-259
Defining a Permanent-Magnet DC Motor 26-260
Defining the General Data for PMDC Motors 26-260
General Data for PMDC Motors 26-261
Defining the Stator Data for a PMDC Motor 26-261
Stator Data for PMDC Motors 26-262
Defining the Stator Pole for a PMDC Motor 26-262
Stator Pole Data for PMDC Motors 26-263
Defining the Rotor Data for a PMDC Motor 26-264
Rotor Data for PMDC Motors 26-264
Defining the Rotor Slots for a PMDC Motor 26-265
Rotor Slot Data for PMDC Motors 26-266
Defining the Rotor Windings and Conductors for a PMDC Motor 26-266
Defining Different Size Wires for a PMDC Motor 26-270
Rotor Winding Data for PMDC Motors 26-271

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining the Commutator and Brush for a PMDC Motor 26-273
Commutator and Brush Data for PMDC Motors 26-274
Defining the Shaft Data for a PMDC Motor 26-275
Shaft Data for PMDC Motors 26-275
Setting Up Analysis Parameters for a PMDC Motor 26-275
Solution Data for PMDC Motors 26-276
Three-Phase Synchronous Machines 26-277
Analysis Approach for Three-Phase Synchronous Machines 26-277
Main Features 26-279
Defining a Three-Phase Synchronous Machine 26-280
Defining the General Data for a Three-Phase Synchronous Machine 26-281
General Data for Three-Phase Synchronous Machines 26-281
Defining the Stator for a Three-Phase Synchronous Machine 26-281
Stator Data for Three-Phase Synchronous Machines 26-282
Defining Stator Slots for a Three-Phase Synchronous Machine 26-283
Stator Slot Data for Three-Phase Synchronous Machines 26-284
Defining Stator Windings and Insulation for a Three-Phase Synchronous Machine 26-284
Stator Winding and Insulation for Three-Phase Synchronous Machines 26-290
Winding Editor for a Three-Phase Synchronous Machine 26-292
Defining Different Size Wires for a Three-Phase Synchronous Machine 26-292
Stator Vent Data for Three-Phase Synchronous Machines 26-293
Defining the Rotor for a Three-Phase Synchronous Machine 26-294
Rotor, Rotor Pole, and Insulation for Three-Phase Synchronous Machines 26-296
Defining the Rotor Pole for a Three-Phase Synchronous Machine 26-298
Defining the Rotor Winding Data for a Three-Phase Synchronous Machine 26-299
Rotor Winding Data for Three-Phase Synchronous Machines 26-301
Defining the Rotor Damper Data 26-301

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Damper Data for Three-Phase Synchronous Machines 26-302
Defining the Shaft Data for a Three-Phase Synchronous Machine 26-303
Shaft Data for Three-Phase Synchronous Machines 26-303
Setting Up Analysis Parameters for a Three-Phase Synchronous Machine 26-303
Solution Data for Three-Phase Synchronous Machines 26-304
Brushless Permanent-Magnet DC Motors 26-305
Analysis Approach for Brushless PMDC Motors 26-305
Defining a Brushless Permanent-Magnet DC Motor 26-307
Defining the General Data for a Brushless PMDC Motor 26-308
General Data for Brushless PMDC Motors 26-308
Defining the Circuit Data for a Brushless PMDC Motor 26-309
Circuit Data for Brushless PMDC Motors 26-310
Defining the Stator Data for a Brushless PMDC Motor 26-311
Stator Data for Brushless PMDC Motors 26-312
Defining the Stator Slots for a Brushless PMDC Motor 26-312
Stator Slot Data for Brushless PMDC Motors 26-313
Defining the Stator Windings and Conductors for a Brushless PMDC Motor 26-313
Winding Editor for a Brushless DC Motor 26-319
Defining Different Size Wires for a Brushless DC Motor 26-320
Stator Winding Data for Brushless PMDC Motors 26-320
Defining the Rotor Data for a Brushless PMDC Motor 26-322
Rotor Data for Brushless PMDC Motors 26-323
Defining the Rotor Pole for a Brushless PMDC Motor 26-323
Rotor Pole Data for Brushless PMDC Motors 26-325
Defining the Shaft Data for a Brushless PMDC Motor 26-325
Shaft Data for Brushless PMDC Motors 26-325
Setting Up Analysis Parameters for a Brushless PMDC Motor 26-326
Analyses Offered 26-326

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution Data for Brushless PMDC Motors 26-327
Switched Reluctance Motors 26-328
Analysis Approach for Switched Reluctance Motors 26-328
Defining a Switched Reluctance Motor 26-330
Defining the General Data for a Switched Reluctance Motor 26-331
General Data for Switched Reluctance Motors 26-332
Defining the Circuit Data for a Switched Reluctance Motor 26-332
Circuit Data for Switched Reluctance Motors 26-334
Defining the Stator Data for a Switched Reluctance Motor 26-334
Stator Data for Switched Reluctance Motors 26-335
Defining the Stator Winding Data for a Switched Reluctance Motor 26-335
Defining Different Size Wires for a Switched Reluctance Motor 26-338
Stator Winding Data for Switched Reluctance Motors 26-339
Defining the Rotor Data for a Switched Reluctance Motor 26-339
Rotor Data for Switched Reluctance Motors 26-340
Defining the Shaft Data for a Switched Reluctance Motor 26-340
Shaft Data for Switched Reluctance Motors 26-341
Setting Up Analysis Parameters for a Switched Reluctance Motor 26-341
Solution Data for Switched Reluctance Motors 26-342
Line-Start Permanent-Magnet Synchronous Motors 26-342
Analysis Approach for Line-Start PM Synchronous Motors 26-342
Defining a Line-Start Permanent Magnet Synchronous Motor 26-344
Defining the General Data for a Line-Start PM Synchronous Motor 26-345
General Data for Line-Start PM Synchronous Motors 26-345
Defining the Stator Data for a Line-Start PM Synchronous Motor 26-346
Stator Data for Line-Start PM Synchronous Motors 26-346
Defining the Stator Slots for a Line-Start PM Synchronous Motor 26-347
Stator Slot Data for Line-Start PM Synchronous Motors 26-348

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining the Stator Windings and Conductors for a Line-Start PM Synchronous Motor 26-348
Winding Editor for a Line-Start Synchronous Motor 26-351
Defining Different Size Wires for a Line-Start Synchronous Motor 26-352
Stator Winding Data for Line-Start PM Synchronous Motors 26-353
Optional Vent for Line-Start PM Synchronous Motor Stator 26-355
Defining the Rotor Data for a Line-Start PM Synchronous Motor 26-355
Rotor Data for Line-Start PM Synchronous Motors 26-356
Defining the Rotor Pole for a Line-Start PM Synchronous Motor 26-356
Rotor Pole Data for Line-Start PM Synchronous Motors 26-357
Optional Rotor Damper for Line-Start PM Synchronous Motor 26-358
Defining the Shaft Data for a Line-Start PM Synchronous Motor 26-358
Shaft Data for Line-Start PM Synchronous Motors 26-359
Setting Up Analysis Parameters for a Line-Start PM Synchronous Motor 26-359
Solution Data for Line-Start PM Synchronous Motors 26-360
Universal Motors 26-360
Analysis Approach for Universal Motors 26-360
Defining a Universal Motor 26-362
Defining the General Data for a Universal Motor 26-363
General Data for Universal Motors 26-363
Defining the Stator Data for a Universal Motor 26-363
Stator Data for Universal Motors 26-364
Defining the Stator Pole for a Universal Motor 26-364
Stator Pole Data for Universal Motors 26-366
Defining the Stator Windings and Conductors for a Universal Motor 26-366
Defining Different Size Wires for a Universal Motor Stator Winding 26-368
Stator Winding Data for Universal Motors 26-369
Defining the Rotor Data for a Universal Motor 26-369

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rotor Data for Universal Motors 26-370
Defining the Rotor Slots for Universal Motors 26-371
Rotor Slot Data for Universal Motors 26-371
Defining the Rotor Windings and Conductors for a Universal Motor 26-372
Defining Different Size Wires for a Universal Motor Rotor Winding 26-376
Rotor Winding Data for Universal Motors 26-377
Defining the Commutator and Brush for a Universal Motor 26-379
Commutator and Brush Data for Universal Motors 26-380
Defining the Shaft Data for a Universal Motor 26-381
Shaft Data for Universal Motors 26-381
Setting Up Analysis Parameters for a Universal Motor 26-381
Solution Data for Universal Motors 26-382
General DC Machines 26-382
Analysis Approach for General DC Machines 26-383
DC Machine Operating as a Motor 26-384
DC Machine Operating as a Generator 26-384
Defining a General DC Machine 26-385
Defining the General Data for a General DC Machine 26-386
General Data for General DC Machines 26-386
Defining the Stator Data for a General DC Machine 26-387
Stator Data for General DC Machines 26-388
Defining the Stator Pole for a General DC Machine 26-388
Stator Pole Data for General DC Machines 26-389
Defining the Stator Field Data for a General DC Machine 26-390
Stator Field Data for General DC Machines 26-390
Shunt Data for General DC Machines 26-390
Series Data for General DC Machines 26-391
Compensating Data for General DC Machines 26-392

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Commutating Data for General DC Machines 26-393
Winding Data for Commutating 26-394
Defining the Rotor Data for a General DC Machine 26-395
Rotor Data for General DC Machines 26-396
Defining the Rotor Slots for a General DC Machine 26-396
Rotor Slot Data for General DC Machines 26-397
Defining the Rotor Windings and Conductors for a General DC Machine 26-398
Defining Different Size Wires for a General DC Machine Rotor Winding 26-401
Rotor Winding Data for General DC Machines 26-402
Vent Data for General DC Machines 26-404
Defining the Commutator and Brush for a General DC Machine 26-405
Commutator and Brush Data for General DC Machines 26-406
Defining the Shaft Data for a General DC Machine 26-407
Access Shaft Data for General DC Machines 26-407
Setting Up Analysis Parameters for a General DC Machine 26-408
Solution Data for General DC Machines 26-408
Claw-Pole Alternators 26-409
Analysis Approach for Claw-Pole Alternators 26-410
Rotor Equipped with an Excitation Winding 26-410
Rotor Equipped with a Permanent Magnet Only 26-411
Power and Efficiency 26-411
Defining a Claw-Pole Alternator 26-412
Defining the General Data for a Claw-Pole Alternator 26-413
General Data for Claw-Pole Alternators 26-413
Defining the Stator Data for a Claw-Pole Alternator 26-414
Stator Data for Claw-Pole Alternators 26-415
Defining the Stator Slot Data for a Claw-Pole Alternator 26-415
Stator Slot Data for Claw-Pole Alternators 26-416

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining the Stator Winding Data for a Claw-Pole Alternator 26-417
Winding Editor for a Claw-Pole Alternator 26-419
Defining Different Size Wires for a Claw-Pole Alternator 26-420
Stator Winding Data for Claw-Pole Alternators 26-421
Defining the Rotor Data for a Claw-Pole Alternator 26-423
Rotor Data for Claw-Pole Alternators 26-423
Defining the Rotor Pole for a Claw-Pole Alternator 26-424
Rotor Pole Data for Claw-Pole Alternators 26-424
Defining the Shaft Data for a Claw-Pole Alternator 26-425
Shaft Data for Claw-Pole Alternators 26-425
Setting Up Analysis Parameters for a Claw-Pole Alternator 26-425
Solution Data for Claw-Pole Alternators 26-426
Three-Phase Non-Salient Synchronous Machines (NSSM) 26-427
Analysis Approach for Three-Phase Non-Salient Synchronous Machines 26-427
Defining Three-Phase Non-Salient Synchronous Machines 26-430
Defining the General Data for a Three-Phase NSSM 26-430
Defining the Stator for Three-Phase NSSM 26-431
Defining Stator Slots for a Three-Phase NSSM 26-432
Defining Stator Windings and Insulation for a Three-Phase NSSM 26-433
Define Wires, Conductors and Windings of NSSM Stator 26-433
Define End Windings and Insulation of NSSM Stator 26-433
Winding Editor 26-434
Stator Vent Data for Three-Phase NSSM 26-434
Define NSSM Rotor Data 26-435
Define NSSM Rotor Slot 26-435
Define NSSM Rotor Winding 26-436
Rotor Vent Data for NSSMs 26-437
Define NSSM Shaft Data 26-438

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Analysis Setup for Three-Phase Non-Salient Synchronous Machines 26-438
Validate NSSM Solution Setup 26-439
Design Output for Non-Salient Synchronous Machines 26-440
View Performance 26-440
View Design Sheet 26-440
View Curves 26-441
Create Reports 26-444
Transient FEA of the Non-Salient Synchronous Machines 26-445
Create Maxwell 2D Design 26-446
Review Maxwell 2D Design Setups 26-446
Solution Type Setup 26-446
Model Setup 26-446
Boundary Setup 26-447
Material Assignment 26-448
Excitation Setup 26-448
Mesh Operation Setup 26-451
Solution Setup 26-451
Analyze Maxwell 2D Design 26-451
Generic Rotating Machines 26-453
Defining a Generic Rotating Machine 26-454
Induction Machines 26-455
Synchronous Machines 26-456
Electronic Commutating Machine 26-456
Defining an Inner-Rotor Induction Machine 26-456
Analysis Approach for Three-Phase Induction Motors 26-457
Defining an Outer-Rotor Induction Machine 26-459
Analysis Approach for Three-Phase Induction Motors 26-460
Defining a Solid-Rotor Induction Machine 26-461

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining a Wound-Rotor Induction Machine 26-462
Defining a Double-Fed Induction Generator 26-463
Analysis Approach for a Doubly Fed Induction Generator 26-464
DFIG Operating as a Generator 26-464
Defining an Axial-Flux Induction Machine 26-467
Defining a Hysteresis Machine 26-468
Defining a Rotating-Field Synchronous Machine 26-469
Defining a Rotating-ArmatureSynchronous Machine 26-470
Defining a Reluctance Synchronous Machine 26-471
Defining an IPM Synchronous Machine 26-472
Defining an Axial-Flux PM Synchronous Machine 26-473
Defining an IPM Brushless DC Motor 26-474
Defining an Axial-Flux Brushless DC Motor 26-475
Generic Rotating Machine Core Types 26-476
Defining Data for the SLOT_AC Core Type 26-477
Defining Data for the SLOT_CAGE Core Type 26-479
Defining Data for the SOLID Core Type 26-481
Defining Data for the AXIAL_AC Core Type 26-482
Defining Data for the AXIAL_CAGE Core Type 26-484
Defining Data for the SALNT_POLE Core Type 26-486
Defining SALNT_POLE Stator and Rotor Field Data for a Generic Rotating Machine. 26-490
Defining SALNT_POLE Stator and Rotor Field Winding Data for a Generic Rotating Machine. 26-490
Defining Data for the NONS_RELU Core Type 26-493
NONS_RELU Rotor Core Pole Types 26-495
Defining Data for the PM_INTERIOR Core Type 26-496
PM_INTERIOR Rotor Core Pole Types 26-498

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining Data for the AXIAL_PM Core Type ... 26-500
Defining the Stator and Rotor Windings for a Generic Rotating Machine ... 26-501
Stator and Rotor Winding Data for Generic Rotating Machines ... 26-506
Winding Editor for a Generic Rotating Machine ... 26-508
Defining Different Size Wires for a Generic Rotating Machine ... 26-509
Stator and Rotor Circuit Data for Generic Rotating Machines ... 26-511
Defining the Axial AC Rotor Brush for a Generic Rotating Machine ... 26-512
Vent Data for Generic Rotating Machines ... 26-512
Defining the Shaft Data for a Generic Rotating Machine ... 26-513
Access Shaft Data for General DC Machines ... 26-514
Setting Up Analysis Parameters for a Generic Rotating Machine ... 26-514
Solution Data for Generic Rotating Machines ... 26-516
Adding Machine Housings ... 26-517
Housing Data ... 26-518
Housing Duct Data ... 26-519
Housing Fin Data ... 26-520
Housing Flange Data ... 26-523
Housing Foot Data ... 26-524
Housing Front Cap and Rear Cap Data ... 26-525
Stator Vent Data ... 26-526
Rotor Vent Data ... 26-527
27 - Using the Maxwell Circuit Editor ... 27-1
Maxwell-Circuit Coupling ... 27-2
Circuit Voltage Equations ... 27-2
Discrete Formula for Capacitors ... 27-3
Error Analysis ... 27-4
Schematic Editor ... 27-5
The Schematic Editor Window ... 27-5

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Schematic Editor Options ... 27-7
Schematic Editor Options: General ... 27-7
Schematic Editor Options: Fonts ... 27-8
Schematic Editor Options: Colors ... 27-8
Schematic Editor Options: Wiring Tab ... 27-9
Schematic Editor Options: Multiple Placement ... 27-9
Schematic Editor Options: Symbol Editor ... 27-10
The Component Libraries Window ... 27-10
The Components Tab ... 27-11
Using the Favorites and Most Recently Used Folders ... 27-12
Help for Components ... 27-13
Placing Components on a Schematic ... 27-13
The Search Tab ... 27-13
Copying and Pasting Properties ... 27-15
To Copy and Paste Selected Properties for Components ... 27-15
To Copy and Paste Common Properties for Primitive Drawing Elements ... 27-16
Components ... 27-18
Dedicated Elements ... 27-18
AC_Model ... 27-19
Commutator Bar and Commutator Bar Model ... 27-20
Winding ... 27-20
ECE3 Model ... 27-21
ECE6 Model ... 27-23
Transformation between the Six-Phase and dq0 Systems ... 27-26
Circuit Model and Look-up Table ... 27-27
ECEDCM Model ... 27-31
ECEF Model ... 27-35
ECEIM Model ... 27-36

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ECEL Model ... 27-45
ECER Model ... 27-47
ECESRM Model ... 27-49
Look-Up Table Processing ... 27-52
ECET Model ... 27-56
ECEW Model ... 27-58
Passive Elements ... 27-59
Diode & Diode Model ... 27-60
Capacitor (CAP) ... 27-61
Resistor (Res) ... 27-61
Inductor (IND) ... 27-62
Mutual Inductance (IndM) ... 27-62
Current Controlled Switch (SW_I) ... 27-62
Voltage Controlled Switch (SW_V) ... 27-64
Transformer ... 27-65
Probes ... 27-66
Ammeter ... 27-66
Voltmeter ... 27-66
Voltmeter with One Pin Grounded (VoltmeterG) ... 27-67
Current and Voltage Sources ... 27-67
DC Current Source ... 27-68
Exponential Current Source ... 27-68
Pulse Current Source ... 27-70
Piecewise Linear Current Source ... 27-71
PWM Current Source ... 27-74
Frequency-Modulated Sinusoidal Current Source ... 27-75
Sinusoidal Current Source ... 27-76
Additional Considerations ... 27-77

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DC Voltage Source ... 27-78
Exponential Voltage Source ... 27-78
Pulse Voltage Source ... 27-79
Piecewise Linear Voltage Source ... 27-81
PWM Voltage Source ... 27-84
Frequency-Modulated Sinusoidal Voltage Source ... 27-85
Sinusoidal Voltage Source ... 27-86
Placing Components in the Maxwell Circuit Editor Schematic ... 27-88
Assigning Component Properties in Maxwell Circuit Editor ... 27-88
Callback Scripting Using PropHost Object ... 27-89
Setting Up an External Circuit Using the Maxwell Circuit Editor ... 27-90
Add the Circuit Elements ... 27-91
Connect the Circuit Elements in Series ... 27-92
Export the Netlist ... 27-92
Save the Maxwell Circuit Editor Project ... 27-92
Assign the External Circuit ... 27-92
Renaming a Source in Maxwell Circuit Editor ... 27-93
Applying the Commutating Bar Element ... 27-93
28 - Maxwell 3D Technical Notes ... 28-1
Maxwell 3D Solutions ... 28-1
Finite Element Analysis ... 28-2
Types of Solutions and Degrees of Freedom (DOFs) ... 28-3
Size of Mesh Vs. Accuracy ... 28-3
The Mesh Generation Process ... 28-4
Seeding the Mesh ... 28-5
Guidelines for Seeding the Mesh ... 28-6
Length-Based Mesh Refinement ... 28-6
Restrict Length of Elements ... 28-7

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Skin Depth-Based Mesh Refinement 28-10
Surface Approximation Settings 28-11
Guidelines for Modifying Surface Approximation Settings 28-12
Meshing Region Vs. Problem Region 28-13
Model Resolution 28-13
Electric Field Calculation 28-14
Symmetry for an Electric Field Solution 28-18
Even Symmetry (Flux Tangential) for an Electric Field Solution 28-18
Odd Symmetry (Flux Normal) for an Electric Field Solution 28-19
Matching Boundary for an Electric Field Solution 28-19
Independent Boundary for an Electric Field Solution 28-20
Dependent Boundary for an Electric Field Solution 28-20
When to Use Matching Boundaries for an Electric Field Solution 28-20
Solution Process for an Electric Field Solution 28-21
Electric Field Energy for an Electric Field Solution 28-21
Capacitance Matrix for an Electric Field Solution 28-22
Capacitance in Terms of Charge and Voltage 28-22
Capacitance in Terms of Current and Time Varying Voltage 28-23
Matrix Elements for an Electric Field Solution 28-23
Diagonal Elements for an Electric Field Solution 28-24
Off-Diagonal Elements for a Capacitance Matrix 28-24
Symmetry for a Capacitance Matrix 28-24
Solution Process for a Capacitance Matrix 28-24
Lumped Capacitance 28-25
Conductance Matrix for a 3D DC Conduction Solution 28-25
Lorentz Force for an Electrostatic Calculation 28-27
Lorentz Torque for an Electrostatic Calculation 28-28
Virtual Force for an Electrostatic Calculation 28-28

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Virtual Torque for an Electrostatic Calculation 28-29
Ohmic Loss for DC Conduction 28-30
Surface Charge Density 28-30
Surface Current Density 28-30
Magnetostatic Field Calculation 28-31
Symmetry for a Magnetostatic Field Solution 28-37
Odd Symmetry (Flux Tangential) for a Magnetostatic Field Solution 28-37
Even Symmetry (Flux Normal) for a Magnetostatic Field Solution 28-38
Matching Boundary for a Magnetostatic Field Solution 28-39
Independent Boundary for a Magnetostatic Field Solution 28-39
Dependent Boundary for a Magnetostatic Field Solution 28-39
When to Use Matching Boundaries for a Magnetostatic Field Solution 28-39
Nonlinear Materials 28-40
Solution Process for a Magnetostatic Field Solution 28-41
Magnetic Field Energy for a Magnetostatic Field Solution 28-41
Magnetic Coenergy for a Magnetostatic Field Solution 28-42
Magnetic Apparent Energy for a Magnetostatic Field Solution 28-43
Inductance Matrix for a Magnetostatic Field Solution 28-44
Types of Inductance 28-44
Inductance in Terms of Flux Linkage and Current 28-46
Inductance in Terms of Voltage and Time-Varying Current 28-46
Matrix Elements for an Inductance Matrix 28-47
Diagonal Elements for an Inductance Matrix 28-47
Off-Diagonal Elements for an Inductance Matrix 28-48
Symmetry for an Inductance Matrix 28-48
Solution Process for an Inductance Matrix 28-48
Grouping Inductance Matrix Elements 28-49
Lorentz Force 28-50

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lorentz Torque ... 28-50
Virtual Force ... 28-51
Virtual Torque ... 28-51
AC Magnetic (Eddy Current) T-Omega Solver ... 28-52
Real and Imaginary Components ... 28-54
Solution Process for an AC Magnetic T-Omega Solution ... 28-55
Nonlinear Field Simulation (3D) ... 28-56
Skin Depth for an AC Magnetic (Eddy Current) T-Omega Solution ... 28-57
Magnetic Field Energy ... 28-58
Hysteresis Loss ... 28-59
Ohmic Loss ... 28-60
Dielectric Loss ... 28-61
Impedance Matrix ... 28-61
Matrix Elements for an Impedance Matrix ... 28-62
Diagonal Elements for an Impedance Matrix ... 28-62
Off-Diagonal Elements for an Impedance Matrix ... 28-62
Symmetry for an Impedance Matrix ... 28-63
Solution Process for an Impedance Matrix ... 28-63
Inductance for an Impedance Matrix ... 28-64
Resistance for an Impedance Matrix ... 28-64
Line Impedance ... 28-64
AC Inductance and Resistance for an Impedance Matrix ... 28-65
Lorentz Force ... 28-66
Lorentz Torque ... 28-66
Virtual Force ... 28-67
Virtual Torque ... 28-67
Average Force ... 28-67
AC Magnetic (Eddy Current) A-Phi Solver (Beta) ... 28-68

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Nonlinear Solution ... 28-70
Skin Effect ... 28-70
Field Definitions for the AC Magnetic A-Phi Solver ... 28-72
Parameters Extraction ... 28-73
Force and Torque ... 28-73
RL and GC Matrices ... 28-74
RL Matrix ... 28-76
GC Matrix ... 28-76
Multiphysics Coupling ... 28-77
3D AC Conduction Solver ... 28-79
Governing Equations ... 28-80
Constitutive Relations ... 28-80
Continuity Equation Including Lossy Domains ... 28-81
Time-Harmonic Field Assumptions ... 28-81
Nonlinear Solution - 3D AC Conduction Solver ... 28-81
Field Definitions for the 3D AC Conduction Solver ... 28-82
Losses Calculation - 3D AC Conduction Solver ... 28-84
Virtual Force ... 28-84
Virtual Torque ... 28-84
Admittance - 3D AC Conduction Solver ... 28-84
Admittance Matrix - 3D AC Conduction Solver ... 28-85
Phasor Notation ... 28-86
Iterative Matrix Solver Technical Details ... 28-87
3D Transient Solver ... 28-88
Coil Terminals ... 28-90
Solution Process ... 28-93
Inductance Computation for 2D and 3D Transient Solutions ... 28-93
Flux Linkage ... 28-94

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3D Transient A-Phi Formulation 28-95
Complex Field Behavior for a Transient Solution 28-97
Special Mesh Structure for a Transient Solution 28-97
Overcoming Challenges for a Transient Solution 28-97
Boundary Conditions for a Transient Solution 28-98
Symmetry for a Transient Solution 28-98
Odd Symmetry (Magnetic Flux Tangential) 28-98
Even Symmetry (Magnetic Flux Normal) 28-99
Matching Boundary for a Transient Solution 28-100
Insulating Boundary for a Transient Solution 28-101
Meshing and Band Setting Recommendations for 3D Transient Applications With Motion 28-101
Improving Reliability of Results for Air-Gap Modeling of Electric Motors (3D) 28-104
Bondwires 28-105
Partial Simulation for Full Rotational Machine Models 28-106
Background Information 28-106
Workflow for Partial Simulation for a Full Rotational Machine Model 28-111
Healing Models 28-111
Error Types 28-112
Detecting Errors 28-113
Healing and Meshing 28-114
Kernel Errors 28-114
Mixed Dimensionality 28-114
Intersecting Objects 28-114
Small Features and Misalignment 28-115
Detecting and Addressing Model Problems to Improve Meshing 28-115
Stage One: Healing During Geometry Import 28-116
Stage Two: Healing after Geometry Import 28-117

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stage Four: Removing Small Features 28-118
Stage Five: Aligning Objects 28-119
Stage Six: Troubleshooting if Meshing Still Fails 28-120
Handling Complicated Models 28-121
Interface Options for Complicated Models 28-121
Geometry Imperfections and Complicated Models 28-122
Modeler > Import 28-122
Modeler > Validation Settings 28-122
Object Overlap Settings for Complicated Models 28-122
Postprocessing for Complicated Models 28-122
Materials 28-123
Relative Complex Permeability 28-123
Relative Complex Permittivity 28-124
Bulk Conductivity 28-124
Dielectric Loss Tangent 28-125
Magnetic Loss Tangent 28-125
Imaginary Permeability 28-126
AC Magnetic 28-126
Ferrite Materials 28-127
Magnetic Saturation 28-127
Lande G Factor 28-128
Delta H 28-128
Anisotropic Materials 28-128
Anisotropic Relative Permeability Tensors 28-128
Anisotropic Relative Permittivity Tensors 28-129
Anisotropic Conductivity Tensors 28-130
Anisotropic Dielectric Loss Tangent Tensors 28-130
Anisotropic Magnetic Loss Tangent Tensors 28-131

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Anisotropic Materials and Ports ... 28-132
Frequency-Dependent Material Properties ... 28-132
Magnetostriction Modeling of Magnetic Material ... 28-133
29 - Maxwell 2D Technical Notes ... 29-1
Solvers ... 29-1
Axisymmetric Field Simulation ... 29-2
Electrostatic Field Simulation ... 29-4
Electrostatic Theory ... 29-4
Capacitance ... 29-5
Capacitance in Terms of Charges and Voltages ... 29-5
Capacitance in Terms of Currents and Time Varying Voltages ... 29-7
Computing Capacitance ... 29-8
Virtual Forces (Electrostatic) ... 29-8
Virtual Torques (Electrostatic) ... 29-9
Flux Linkage (Electrostatic) ... 29-10
Magnetostatic Field Simulation ... 29-10
Magnetostatic Theory ... 29-10
Inductance ... 29-12
Inductance in Terms of Flux Linkage and Currents ... 29-13
Inductance in Terms of Voltages and Time Varying Currents ... 29-14
Computing an Inductance Matrix ... 29-15
Virtual Forces (Magnetostatic) ... 29-15
Virtual Torques (Magnetostatic) ... 29-16
Flux Linkage (Magnetostatic) ... 29-17
AC Magnetic Field Simulation ... 29-17
AC Magnetic Solver Theory ... 29-17
Components of Current Density ... 29-18
Integrating the Current Density ... 29-19

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AC Magnetic Assumptions ... 29-19
Deriving the AC Magnetic Equation ... 29-19
Maxwell's Equations ... 29-20
Relationship of Magnetic and Electric Field ... 29-21
Relationship of Current and Current Density ... 29-21
Eddy Currents and Skin Depth ... 29-22
Impedance Matrix ... 29-23
Computing an Impedance Matrix ... 29-24
Matrix Inductance ... 29-24
Matrix Resistance ... 29-25
Inductance and Resistance in Impedance Computations ... 29-25
Virtual Forces ... 29-26
Virtual Torques ... 29-27
Current Flow ... 29-28
Nonlinear AC Magnetic Field Simulation ... 29-28
Nonlinear AC Magnetic Theory ... 29-28
Sinusoidal B ... 29-29
DC Conduction Field Simulation ... 29-29
DC Conduction Theory ... 29-30
Steady-State Conditions ... 29-31
Relevant Time Constant ... 29-32
Conductance ... 29-32
Current Flow (DC Conduction) ... 29-33
AC Conduction Field Simulation ... 29-34
AC Conduction Theory ... 29-34
AC Conduction Assumptions ... 29-34
Admittance ... 29-35
Current Flow ... 29-35

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Transient Simulation 29-35
Transient Simulation Assumptions 29-35
Time-Dependent Magnetic Field Simulation 29-36
Stranded Conductors 29-36
Solid Conductors 29-37
Solid Conductors with Current Sources 29-38
Solid Conductors with Voltage Sources 29-38
Translational Motion 29-39
Rotational Motion 29-39
Improving Reliability of Results for Air-Gap Modeling of Electric Motors (2D) 29-40
Core Loss for Transient Solvers 29-41
Phasor Notation 29-43
Real and Imaginary Components 29-44
Boundary Conditions 29-45
Vector Potential 29-47
Vector Potential Boundaries in Axisymmetric Models 29-47
Symmetry 29-48
Odd Symmetry 29-48
Even Symmetry 29-49
Balloon 29-50
Impedance 29-51
Resistance 29-53
Independent/Dependent 29-54
2D Sources/Excitations 29-55
2D Voltage Sources 29-56
DC Voltage Source 29-57
AC Voltage Source 29-57
2D Current/Current Density 29-57

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DC Current Source ... 29-57
AC Current Source ... 29-58
Active vs Passive Conductors ... 29-58
Passive Conductors ... 29-59
Current Sources for Touching Conductors ... 29-59
Current Sources for Perfect Conductors ... 29-60
Solid, Stranded, and Parallel Current Sources ... 29-60
2D Charge/Charge Density ... 29-61
2D Floating ... 29-62
External Circuit Connections ... 29-62
References for Electric or Magnetic Potential ... 29-63
End Connections in 2D Transient and AC Magnetic Solvers ... 29-63
Functional Boundaries and Sources ... 29-64
30 - Maxwell Examples ... 30-1
Maxwell 2D and 3D Examples ... 30-1
RMxprt Examples ... 30-3
31 - Maxwell Terminology ... 31-1
Glossary: A ... 31-1
Glossary: B ... 31-1
Glossary: C ... 31-1
Glossary: D ... 31-3
Glossary: E ... 31-3
Glossary: F ... 31-4
Glossary: G ... 31-5
Glossary: H ... 31-5
Glossary: I ... 31-5
Glossary: L ... 31-7
Glossary: M ... 31-8

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Glossary: N 31-10
Glossary: O 31-11
Glossary: P 31-11
Glossary: R 31-12
Glossary: S 31-13
Glossary: T 31-15
Glossary: V 31-15
Index **Index-1**

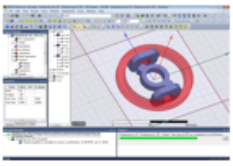
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

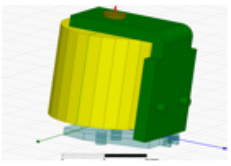
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

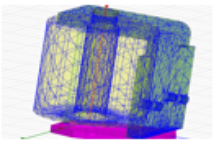
1 - Welcome to Ansys Maxwell Help

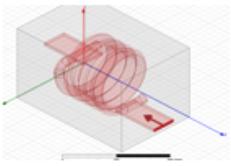
Use the following links for quick information on the following topics:

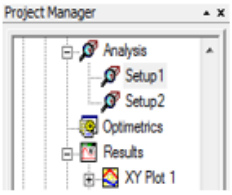
Getting Started Maxwell


Maxwell User Interface

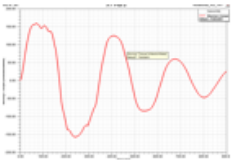

Modeling

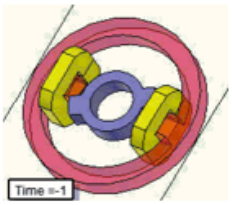

Meshing


Boundaries and Excitations

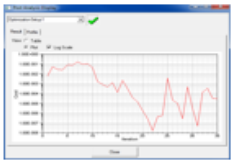


Analysis


Results



Example Projects


Optimetrics

Use the following links for Maxwell Process Flow topics:

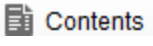
User Interface Quick Links

Use the following links for quick information on the following topics.

The Maxwell Desktop	Toolbar Options
General Purpose Keyboard Shortcuts for the Maxwell Application	Running Ansys Electronics Desktop from a Command Line

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The help system provides different ways to find information and navigate quickly:

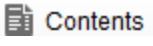
- Press **F1** on any open dialog box to open the relevant help topic.
- *A hierarchical table of contents* –  – Browse through the table of contents, expand entries, and close entries. Click an entry to see it in the content area.
- *A full text search* – To locate occurrences of a word or phrase that may be contained in the help, use the search function.

Modeling Quick Links

Use the following links for quick information on the following topics.

Setting Units of Measurement
Assigning Transparency to an Object
Subtracting Objects
Measuring Objects
Drawing Bondwires
Keyboard Shortcuts
Modifying the Model View
Working with Variables
Setting Modeler Drawing Options
Selecting Items in the Modeler Window
Drawing Objects
Choosing the Cursor Movement Mode
Importing Files
Modifying Objects
Creating a User-Defined Primitive
Assigning Materials

The help system provides different ways to find information and navigate quickly:

- Press **F1** on any open dialog box to open the relevant help topic.
- *A hierarchical table of contents* –  – Browse through the table of contents, expand entries, and close entries. Click an entry to see it in the content area.
- *A full text search* – To locate occurrences of a word or phrase that may be contained in the help, use the search function.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Boundaries/Excitations Quick Links

Use the following links for quick information on the boundary and excitation topics.

Defining Boundary Conditions for 3D Designs	Defining Boundary Conditions in 2D
Defining Excitations for 3D Designs	Defining Excitations for 2D Designs
Functional Boundaries and Excitations for 3D Designs	Functional Boundaries and Excitations for 2D Designs

The help system provides different ways to find information and navigate quickly:

- Press **F1** on any open dialog box to open the relevant help topic.
- *A hierarchical table of contents* – Contents – Browse through the table of contents, expand entries, and close entries. Click an entry to see it in the content area.
- *A full text search* – To locate occurrences of a word or phrase that may be contained in the help, use the search function.

Meshing Quick Links

Use the following links for quick information on the following topics.

Defining Mesh Operations	Plotting Finite Element Mesh
Detecting and Addressing Model Problems to Improve Meshing	Handling Complicated Models
Specifying Initial Mesh Settings	

The help system provides different ways to find information and navigate quickly:

- Press **F1** on any open dialog box to open the relevant help topic.
- *A hierarchical table of contents* – Contents – Browse through the table of contents, expand entries, and close entries. Click an entry to see it in the content area.
- *A full text search* – To locate occurrences of a word or phrase that may be contained in the help, use the search function.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Analysis Quick Links

Use the following links for quick information on the following topics.

Specifying the HPC and Analysis Options
Configuring Distributed Analysis
Selecting Optimal Configurations for Distributed Analysis
Specifying Output Variable Convergence
Running Simulations
Ansys Workbench Integration Overview
Remote Analysis
Specifying Solution Settings
High Performance Computing Integration

The help system provides different ways to find information and navigate quickly:

- Press **F1** on any open dialog box to open the relevant help topic.
- *A hierarchical table of contents* – – Browse through the table of contents, expand entries, and close entries. Click an entry to see it in the content area.
- *A full text search* – To locate occurrences of a word or phrase that may be contained in the help, use the search function.

Optimetrics Quick Links

Use the following links for quick information on the following topics.

Optimetrics
Parametric Overview
Setting up a Parametric Analysis
Setting up a Sensitivity Analysis
Setting up a Statistical Analysis
Setting up an Optimization Analysis
Tuning a Variable
Setting a Range Function
Setting up Calculations for Optimetrics

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adding a Cost Function
Running an Optimetrics Analysis

The help system provides different ways to find information and navigate quickly:

- Press **F1** on any open dialog box to open the relevant help topic.
- *A hierarchical table of contents* – Contents – Browse through the table of contents, expand entries, and close entries. Click an entry to see it in the content area.
- *A full text search* – To locate occurrences of a word or phrase that may be contained in the help, use the search function.

Results Quick Links

Use the following links for quick information on result topics.

View solution data	Creating Reports
Plot field overlay	Working with Traces
Plot the finite element mesh	Adding Data Markers to Traces
Create animations	Creating a Quick Report
Specifying Output Variables	Dynamically Updating Reports During Solution
Using the Fields Calculator	

The help system provides different ways to find information and navigate quickly:

- Press **F1** on any open dialog box to open the relevant help topic.
- *A hierarchical table of contents* – Contents – Browse through the table of contents, expand entries, and close entries. Click an entry to see it in the content area.
- *A full text search* – To locate occurrences of a word or phrase that may be contained in the help, use the search function.

Finding Information in the Online Help

The help system provides different ways to find information and navigate quickly:

- Press **F1** on any open dialog box to open the relevant help topic.
- *A hierarchical table of contents* – Contents – Browse through the table of contents, expand entries, and close entries. Click an entry to see it in the content area.

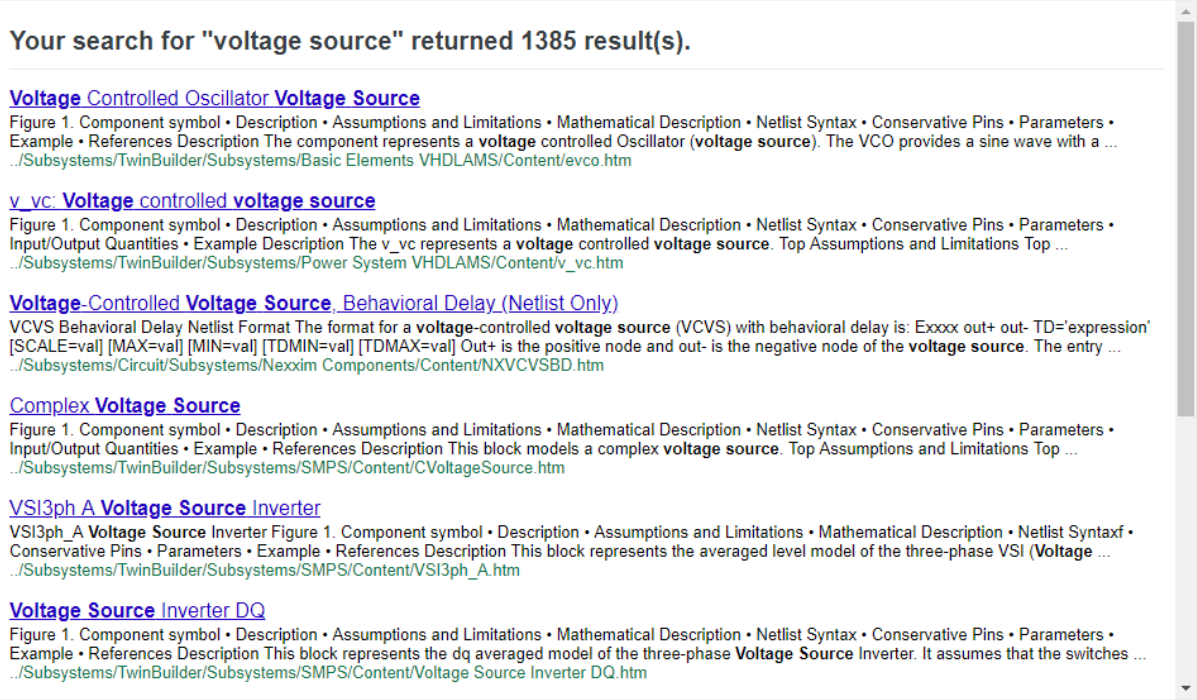
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- *A full text search* – To locate occurrences of a word or phrase that may be contained in the help, use the search function.

> **Note:**
>
> Ansys Electronics Desktop Student includes access to PDF documentation only.

Using the Search Function in the Help

When you enter words or strings to search for in the help, the search engine lists all topics in which any of the words occur. For example, if you enter “voltage source” without the quotation marks, the results show all topics that contain “voltage” or “source.”

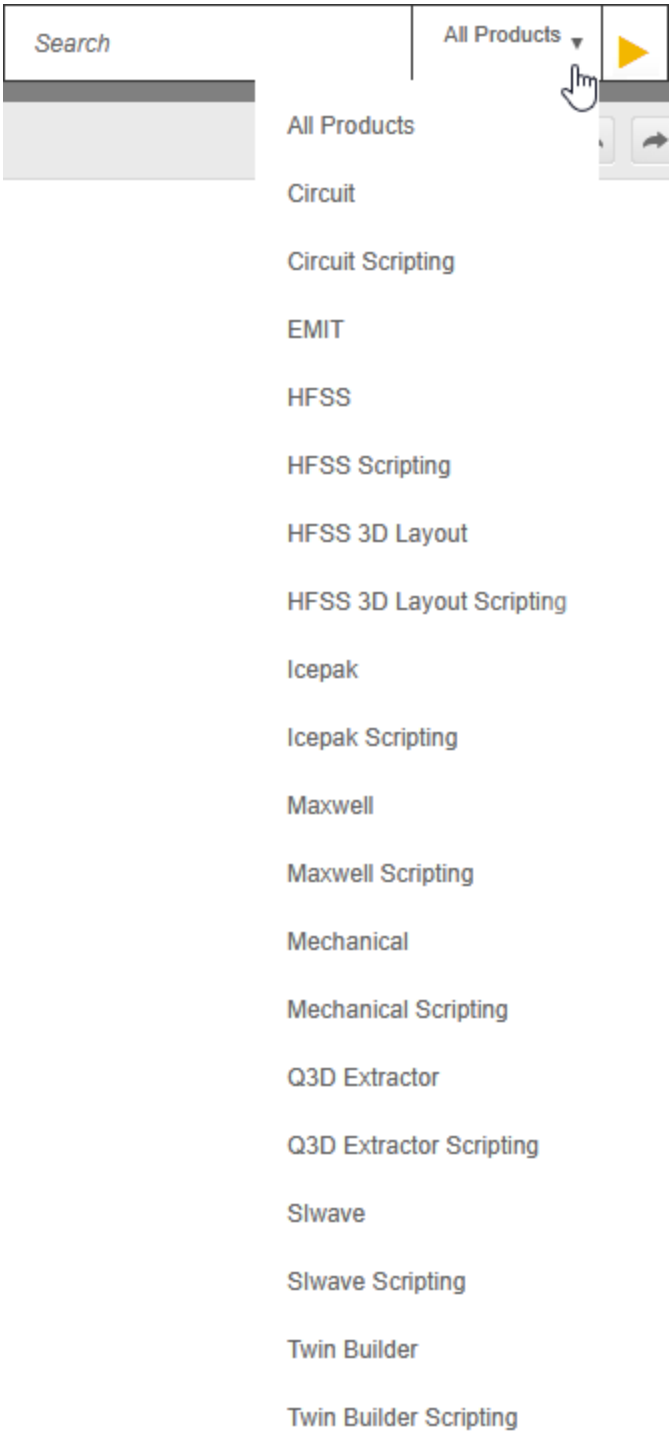


This method probably provides more hits than you want. The search function in the help provides several methods for making searches more specific.

Performing a Basic Search

1. Type the words or string in the search box.
    - If you are searching within the full Electronics help system, you see a search box that includes a drop-down filter for specifying a product, a product's scripting guide, or searching across all products. When you change the filter, the results dynamically reflect the selected filter.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



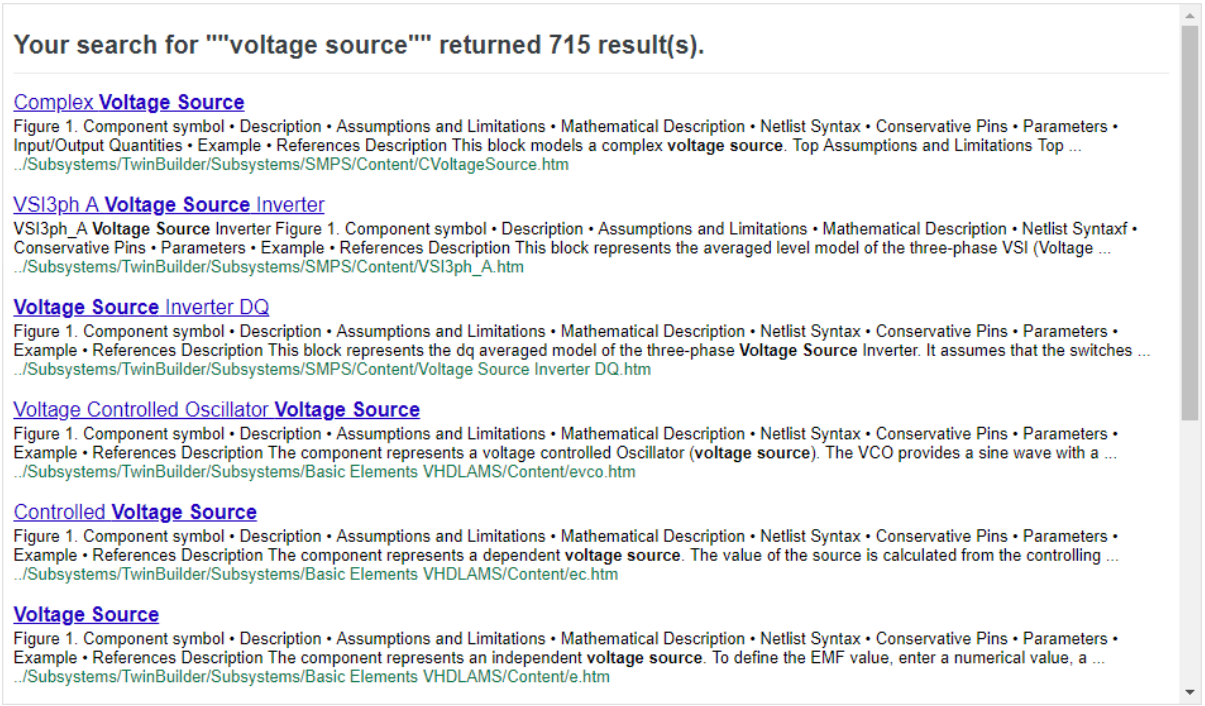
- If you accessed the help for a specific product by pressing **F1** in Electronics Desktop, the search box will have preselected that product. You can change the selection to a different product, a product's scripting guide, or all products.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click on the topic you want in the results list.
    - Some topics in different products share the same title. When searching "All Products," check the URL below each link. The URL indicates the product under which the topic falls.
    - After clicking a topic, if you want to view a different topic, click your browser's back button to return to the results list.
    - To turn off highlighting on the page you are viewing, click the Remove Highlights icon ( ).

Searching with Quotation Marks

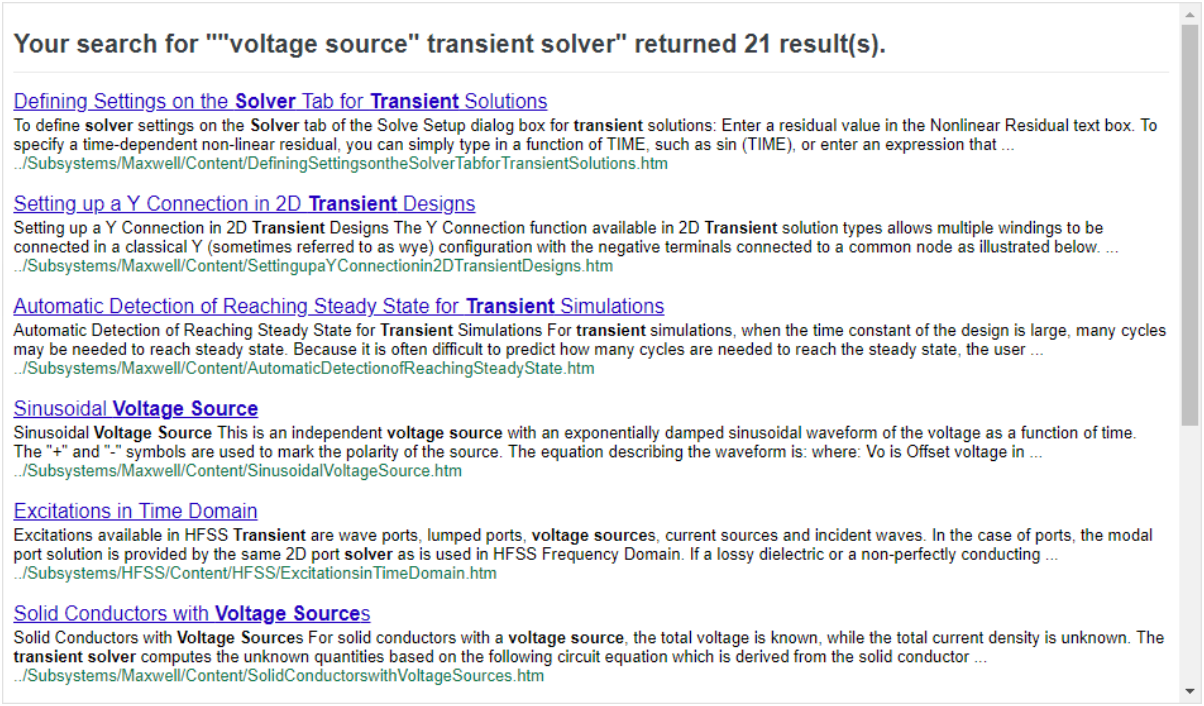
If you enter “voltage source” with quotation marks, the results show all topics that include the phrase.



As you can see, this returns far fewer results than the basic search.

To further limit the results, you can enter additional words, such as: “voltage source” transient solver

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

- Searches are not case sensitive, so you can type your search in uppercase or lowercase characters.
- You may search for any combination of letters (a-z) and numbers (0-9).
- Punctuation marks (period, colon, semicolon, comma, hyphen) are ignored during a search.
- When searching for a file name with an extension, group the entire string in quotation marks (for example, "filename.ext").

Using Boolean Operators

Note:

Functionality featured in the example(s) in this section applies to multiple design types.

You can also use boolean operators to affect the number of topics listed.

Operator (s)	Usage	Example(s)
AND	Lists all topics that contain all of the terms.	Net AND Selection

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Operator (s)	Usage	Example(s)
+ &		Net + Selection Net & Selection
OR |	Lists all topics that contain any of the terms.	Net OR Selection Net | Selection
NEAR	Lists all topics that contain the terms near the other terms.	Net NEAR Selection
NOT ! ^	Lists all topics that contain the first term but not the second.	Net NOT Selection Net ! Selection Net ^ Selection

Use parentheses to group terms and operators. For example:

- solver AND (Circuit) NOT HFSS NEAR dynamic
- solver AND (HFSS OR Circuit) NOT (Q3D OR 2d) NEAR dynamic
- “dynamic link” ! (HFSS | circuit)

> **Important:**
>
> Because the characters +, &, |, !, and ^ are used as operators, you cannot search for them in the help. Doing so will result in an error.

Help Conventions

Please take a moment to review how instructions and other useful information are presented in this documentation.

- Procedures are presented as numbered lists. A single bullet indicates that the procedure has only one step.
- Bold type is used for the following:
  - Keyboard entries that should be typed in their entirety exactly as shown. For example, “**copy file1**” means you must type the word **copy**, then type a space, and then type **file1**.
  - On-screen prompts and messages, names of options and text boxes, and menu commands. Menu commands are often separated by greater than signs (>). For example, “click **HFSS** > **Excitations** > **Assign** > **Wave Port**.”
  - Labeled keys on the computer keyboard. For example, “Press **Enter**” means to press the key labeled **Enter**.

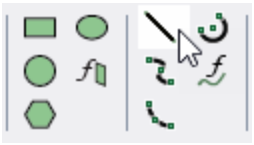
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Italic type is used for the following:
  - Emphasis
  - The titles of publications
  - Keyboard entries when a name or a variable must be typed in place of the words in italics. For example, “**copy** *filename*” means you must type the word **copy**, then type a space, and then type the name of the file.
- The plus sign (+) is used between keyboard keys to indicate that you should press the keys at the same time. For example, “Press Shift+F1” means to press the **Shift** key and, while holding it down, press the **F1** key also. You should always depress the modifier key or keys first (for example, Shift, Ctrl, Alt, or Ctrl+Shift), continue to hold it/them down, and then press the last key in the instruction.

Accessing Commands: *Ribbons, menu bars,* and *shortcut menus* are three methods that can be used to see what commands are available in the application.

- The *Ribbon* occupies the rectangular area at the top of the application window and contains multiple tabs. Each tab has relevant commands that are organized, grouped, and labeled. An example of a typical user interaction is as follows:

  "Click **Draw** > **Line**"

  This instruction means that you should click the **Line** command on the **Draw** ribbon tab. An image of the command icon, or a partial view of the ribbon, is often included with the instruction.

- The *menu bar* (located above the ribbon) is a group of the main commands of an application arranged by category such File, Edit, View, Project, etc. An example of a typical user interaction is as follows:

  "On the **File** menu, click the **Open Examples** command" means you can click the **File** menu and then click **Open Examples** to launch the dialog box.

- Another alternative is to use the *shortcut menu* that appears when you click the right-mouse button. An example of a typical user interaction is as follows:

  “Right-click and select **Assign Excitation> Wave Port**” means when you click the right-mouse button with an object face selected, you can execute the excitation commands from the shortcut menu (and the corresponding sub-menus).

Getting Help: Ansys Technical Support

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For information about Ansys Technical Support, go to the Ansys corporate Support website, http://www.ansys.com/Support. You can also contact your Ansys account manager in order to obtain this information.

All Ansys software files are ASCII text and can be sent conveniently by e-mail. When reporting difficulties, it is extremely helpful to include very specific information about what steps were taken or what stages the simulation reached, including software files as applicable. This allows more rapid and effective debugging.

Help Menu

To access help from the Help menu, click **Help** and select from the menu:

- **[product name] Help** – opens the contents of the help. This help includes the help for the product and its *Getting Started Guides*.
- **[product name] Scripting Help** – opens the contents of the *Scripting Guide*.
- **[product name] Getting Started Guides** – opens a topic that contains links to Getting Started Guides in the help system.

Context-Sensitive Help

To access help from the user interface, press **F1**. The help specific to the active product (design type) opens.

You can press **F1** while the cursor is pointing at a menu command or while a particular dialog box or dialog box tab is open. In this case, the help page associated with the command or open dialog box is displayed automatically.

Getting Help from Ansys Technical Support

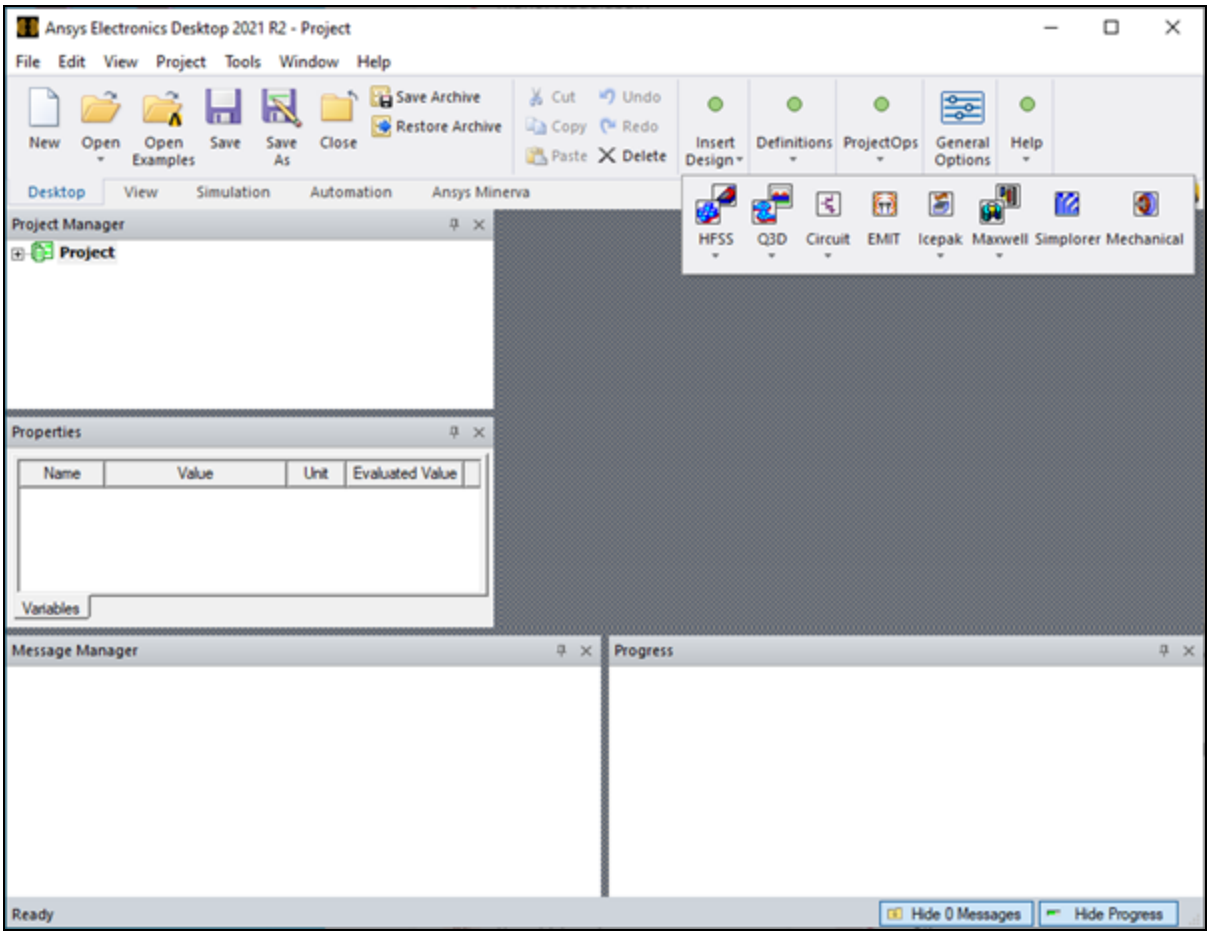
To contact Ansys technical support staff in your geographical area, please log on to the Ansys corporate website, ansys.com/support. You can also contact your Ansys EM account manager in order to obtain this information.

E-mail can work well for technical support. All Ansys EM software files are ASCII text and can be sent conveniently by email. When reporting difficulties, it is extremely helpful to include very specific information about what steps were taken or what stages the simulation reached. This allows more rapid and effective debugging.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2 - Getting Started with Ansys Electronics Desktop

The Ansys Electronics Desktop, shown in the following figure, provides a comprehensive environment for designing and simulating various electronic components and devices. The Electronics Desktop consists of a unified user interface where electromagnetic designs, thermal designs, and circuits can be created. Typically, you can create or import a design, set up the simulation, validate your design, run the analysis, and post process the results.



The desktop includes the following design types:

- **HFSS** – a general purpose 3D interface for the design, analysis, and simulation of electromagnetic components.
- **HFSS 3D Layout** – a full-wave, layout-based electromagnetic simulator with a specialized interface for geometries created in layout.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Q3D Extractor** – a quasi-static 3D solver for extracting lumped RLGC parameters and Spice models.
- **2D Extractor** – a 2D solver for extracting per-unit-length RLGC parameters of transmission lines.
- **Circuit** – a schematic-based interface to the Nexxim circuit simulator.
- **Circuit Netlist** – a netlist (text-based) interface to the Nexxim circuit simulator.
- **EMIT** – system simulation for predicting and mitigating radio frequency interference (RFI) in electronic devices.
- **Maxwell 3D** – uses finite element analysis (FEA) to solve three-dimensional (3D) electrostatic, magnetostatic, AC magnetic (eddy current), and transient problems.
- **Maxwell 2D** – uses finite element analysis (FEA) to solve two-dimensional (2D) electrostatic, magnetostatic, AC magnetic (eddy current), and transient problems.
- **RMxprt** – a template-based electrical machine design tool that provides fast, analytical calculations of machine performance and 2-D and 3-D geometry creation for detailed finite element calculations in Maxwell.
- **Maxwell Circuit** – sets up external circuit designs to supply excitations to coil terminals for Maxwell 2D and 3D AC Magnetic and Transient designs.
- **Simplorer** – an integrated, multi-domain, mixed-signal simulator for complex technical systems. Simplorer is a subset of the Twin Builder standalone product. Please see the Twin Builder help for more information.
- **Icepak** – a fluid and thermodynamic simulator for electronic systems and components.
- **Mechanical** – perform modal analyses to determine natural vibration frequencies and thermal analyses to determine temperatures and heat flux.

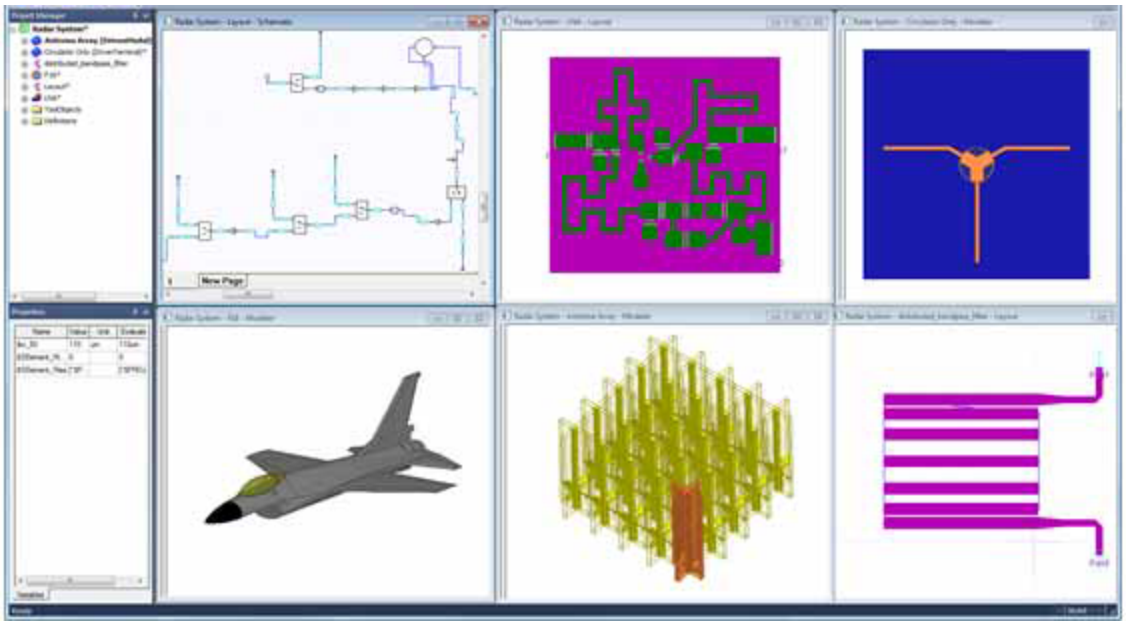
You can access all of these design types and features from the **Project** menu, and any combination of design types can be inserted into a single project file. The schematics can be used to wire up the different field solver models and create a model of a high-level system. Ansys Electronics Desktop provides an efficient way to manage complicated projects that require several different analysis tools to model all of their pieces. Designs can also be parameterized. With the help of the Optimetrics feature, the best design variations can be made available to other modules when the designs are linked into a higher-level simulation. This lets you study the effect of varying a design parameter on the behavior of the entire system.

You can access these design types and features from the Windows launcher. You can use the ACT Toolkit for HFSS-EMA3D Datalink to launch and use this tool.

The following illustration shows how the Ansys Electronics Desktop can be used to model different components for radar system analysis. An antenna array created in HFSS is linked to an IE design of an F16 aircraft. The low noise amplifier and bandpass filter are two important components in the receiver part of the radar module circuit design. The low noise amplifier and the filter can be modeled in HFSS 3D Layout and linked together in a circuit simulation, along with other components of the radar module connected to the antenna array. The outputs of the radar module can be used to drive the antennas using the push excitation feature, whereby the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

voltages on the antenna array ports can be automatically set to correspond to those of the driving circuit. The push excitation feature enables the user to view electromagnetic fields when the array is driven by the radar module circuit.



Ansys Electronics Desktop Student

> **Important:**
>
> Do not install the same release version of Electronics Desktop and Electronics Desktop Student on the same machine.

Ansys Electronics Desktop Student is a free Windows version of the Electronics Desktop that allows you to model, mesh (if applicable), solve, and post-process in HFSS, Q3D Extractor, 2D Extractor, Circuit, Maxwell 3D and Maxwell 2D, RMxprt, and Icepak. The following list of limitations applies to all supported design types in Ansys Electronics Desktop Student.

> **Note:**
>
> Unsupported design types (e.g., Mechanical) do not open in Electronics Desktop Student.

- Geometry export not supported
- Import of DXF and STEP files only

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Local solve only (remote configuration not supported)
- High Performance Computing limited to 4 cores
- optiSLang and LSDSO not supported
- Integration with Ansys Workbench not supported
- Beta features not supported
- Linux version not supported

Refer to the following sections to review limitations specific to the supported design types in Ansys Electronics Desktop Student.

HFSS Student Limitations

- SBR+ not supported
- Mesh assemblies not supported
- Mesh element count limit (analysis and post-processing)
    - 3D volume: 64,000 elements
    - 3D surface: 8,000 elements
    - 2D: 2,000 triangles
- Circuit model generation from S-parameters not supported

Q3D Extractor and 2D Extractor Student Limitations

- Mesh element count limit (analysis and post-processing)
    - 3D volume: 64,000 elements
    - 3D surface: 8,000 elements
    - 2D: 2,000 triangles
- Circuit model generation from S-parameters not supported

Circuit Limitations

- Netlist design type not supported
- Component limit: 50 components
- Command line analysis initialization not supported
- Device data generation not supported
- Access to layout functionality not supported
- Circuit model generation from S-parameters not supported

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Maxwell and RMxprt Student Limitations

- Mesh element count limit (analysis and post-processing)
  - 3D volume: 64,000 elements
  - 3D surface: 8,000 elements
  - 2D: 2,000 triangles
- 3D and 2D Transient simulations

Icepak Student Limitations

- Mesh element count limit (analysis and post-processing): 512,000 elements
- Classic Icepak files (.tzr) with ECAD not supported

System Requirements

Ansys Electronics Desktop supports certain versions of Windows and Linux. For supported platforms and system requirements, go to the Platform Support website and select the following document:

Ansys 2025 R2 **- Platform Support by Application / Product (PDF)**

This document covers all Ansys products. Refer to the **Electronics Applications** section.

Limitations of Linux Installations:

While the majority of the Ansys Electromagnetics Suite applications and features are supported on both Windows and Linux platforms, some are not supported on Linux. For up-to-date platform support by product:

1. Go to the Ansys Platform Support website.
2. From the list of documents, click the link, Ansys 2025 R2 - Platform Support by Application / Product (PDF).
3. This document includes platform support information for all Ansys products. Refer to the section labeled Electronics Applications.

Launching Ansys Electronics Desktop

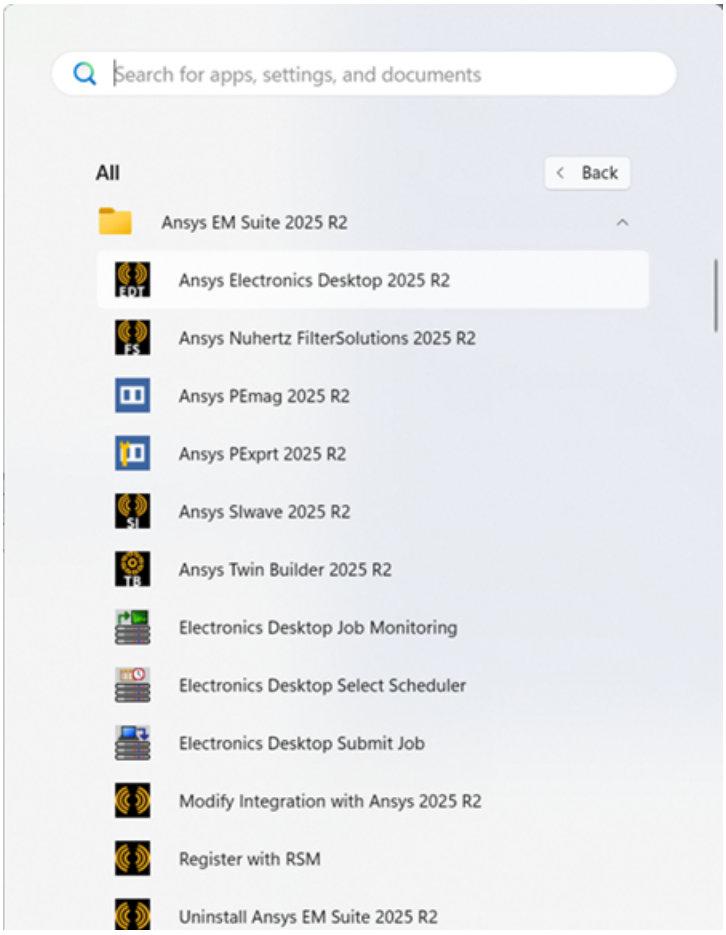
Follow the instructions below to launch Ansys Electronics Desktop in either Windows or Linux.

Windows:

Once you have installed Ansys Electronics Desktop, start the program using one of the following methods:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- From the Start menu, select **Ansys EM Suite 2025 R2** > **Ansys Electronics Desktop 2025 R2**.



If the program fails to start, ensure that:

- You have installed the licensing option provided to you. For detailed information on installing the software and licenses, see: Ansys EM Suite Installation. This topic provides links to the Ansys installation guides (Windows and Linux) and product-specific platform support information.
- Ansys QA Services are not enabled. To disable, change the value of the ENABLE_ANSYS_QA_SERVICES environment variable to 0.

Linux:

On Linux systems, an interactive Ansys Electronics Desktop session can be launched either from the command line, or from a GNOME-compliant desktop environment. The graphical session must be running directly on the machine's console or via a remote connection like VNC.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Launching AEDT from the command line:

When AEDT is installed, a sub-directory is created under the installation directory: /v252/AnsysEM/. This directory contains the software executable files and wrapper scripts.

To launch AEDT from the command line, invoke it via its wrapper script. If, for example, AEDT has been installed in the /opt/ansys_inc/ directory, the wrapper would be:

- /opt/ansys_inc/v252/AnsysEM/ansysedt

When this command is run for the first time on a machine, messages will be printed to the terminal as the software automatically configures itself. AEDT will then launch.

> **Note:**
>
> The installation directory contains a file named ansysedt.exe. Do not attempt to run this directly. The corresponding wrapper scripts perform essential environment setup tasks that the software needs to function properly.

Starting from GNOME desktop:

- Click the Activities button on the GNOME panel to show installed applications:

- Click the icon for Ansys EM:

Ansys Product Improvement Program

This product is covered by the Ansys Product Improvement Program, which enables ANSYS, Inc., to collect and analyze anonymous usage data reported by our software without affecting your work or product performance. Analyzing product usage data helps us to understand customer usage trends and patterns, interests, and quality or performance issues. The data enable us to develop or enhance product features that better address your needs.

How to Participate

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The program is voluntary. To participate, select **Yes** when the Product Improvement Program dialog box appears. Only then will collection of data for this product begin.

How the Program Works

After you agree to participate, the product collects anonymous usage data during each session. When you end the session, the collected data is sent to a secure server accessible only to authorized Ansys employees. After Ansys receives the data, various statistical measures such as distributions, counts, means, medians, modes, etc., are used to understand and analyze the data.

Data We Collect

For all products that offer the Ansys Product Improvement Program, we only collect anonymous data such as session statistics, hardware information, types of loading, solution types, solution statistics, and similar data. The specific data collected varies from product to product.

For Ansys Electronics, we collect the following information:

- Application
    - Build information
    - System information
        - Country
        - Country code
        - CPU architecture
        - CPU brand
        - CPU identifier
        - Graphics card
        - Operating system
        - Operating system version
        - Processor count
        - Time zone
        - Total RAM value
- Session
    - Workbench session
    - Total CPU time
    - Execution mode
    - Start method
    - Number of processes
    - Number of compute nodes (HPC)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

    - Session begin
    - Session end
- Mesh
    - Number of nodes
    - Number of elements
    - Number of zones
    - Number of faces

Data We Do Not Collect

The Product Improvement Program does not collect any information that can identify you personally, your company, or your intellectual property. This includes but is not limited to names, addresses, file names, part names, geometry- or design-specific inputs, material property values, etc. We make no record of where we collect data from.

Opting Out of the Program

You may stop your participation in the program any time you wish. To do so, select menu item **Help > Ansys Product Improvement Program** or ribbon item **Help > Ansys > Ansys Product Improvement Program**. This activates the **Ansys Product Improvement Program** dialog box. On this dialog select the **No I am not willing to participate** radio button to opt out of the program. Select the **Yes, I am willing to participate in the Ansys Product Improvement Program** radio button to opt in to the program. Click the **OK** button to accept the radio button selection. Data is no longer collected or sent if you opt out.

> **Note:**
>
> You can disable the Ansys Product Improvement Program for all users so that each user is not prompted to enable the Program when they first start Electronics Desktop. After installing the software, run the following command as a user with permissions to modify the installed file set:
>
> ```
> UpdateRegistry.exe -set -ProductName ElectronicsDesktop2025.2 -
> RegistryKey
> Desktop/Settings/ProjectOptions/ProductImprovementOptStatus -
> RegistryLevel install -RegistryValue 1
> ```

The ANSYS, Inc., Privacy Policy

All Ansys products are covered by the ANSYS, Inc., Privacy Policy, which you can read here.

Frequently Asked Questions

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. *Am I required to participate in this program?*

   No, your participation is voluntary. We encourage you to participate, however, as it helps us create products that will better meet your future needs.

2. *Am I automatically enrolled in this program?*

   No. You are not enrolled unless you explicitly agree to participate.

3. *Does participating in this program put my intellectual property at risk of being collected or discovered by ANSYS?*

   No. We do not collect any project-specific, company-specific, or model-specific information.

4. *Can I stop participating even after I agree to participate?*

   a. Select the menu item **Help** > **Ansys Product Improvement Program** or ribbon item **Help** > **Ansys** > **Ansys Product Improvement Program**. This will activate the **Ansys Product Improvement Program** dialog box.

   b. Select the **No I am not willing to participate radio** button to opt out of the program.

   c. Click the **OK** button to accept the radio button selection.

   d. Data will no longer be collected or sent.

5. *Will participation in the program slow the performance of the product?*

   No, the data collection does not affect the product performance in any significant way. The amount of data collected is very small.

6. *How frequently is data collected and sent to Ansys servers?*

   The data is collected during each use session of the product. The collected data is sent to a secure server once per session, when you exit the product.

7. *Is this program available in all Ansys products?*

   Not at this time, although we are adding it to more of our products at each release. The program is available in a product only if this Ansys Product Improvement Program description appears in the product documentation, as it does here for this product.

8. *If I enroll in the program for this product, am I automatically enrolled in the program for the other Ansys products I use on the same machine?*

   Yes. Your enrollment choice applies to all Ansys products you use on the same machine. Similarly, if you end your enrollment in the program for one product, you end your enrollment for all Ansys products on that machine.

9. *How is enrollment in the Product Improvement Program determined if I use Ansys products in a cluster?*

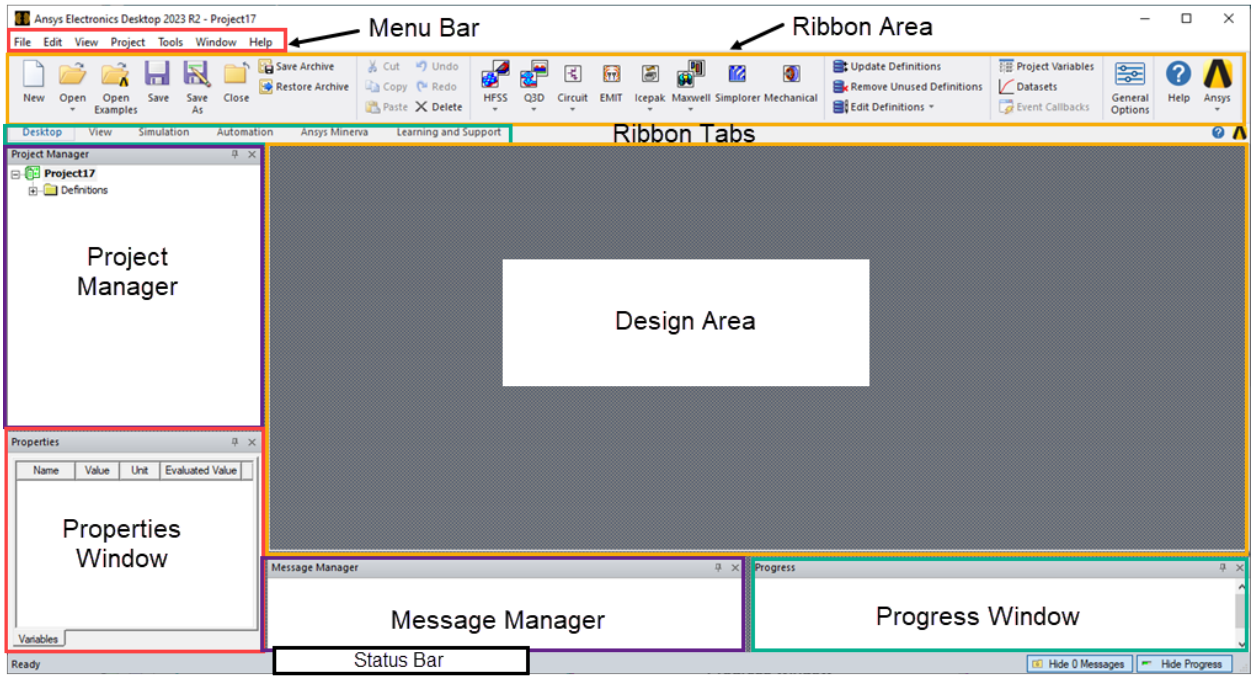
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In a cluster configuration, the Product Improvement Program enrollment is determined by the host machine setting.

Ansys Electronics Desktop Overview

The Ansys Electronics Desktop consists of several windows, a menu bar, multitab ribbon, and a status bar. You can customize the appearance of the desktop by moving, hiding, or showing windows. Some menus, and other features, change depending on the type of project that is loaded and the editor that is active in the Design Area.

> **Note:**
>
> To ensure that all Electronics Desktop dialog boxes are visible on your monitor, it is recommended to not scale the display resolution above 100%.

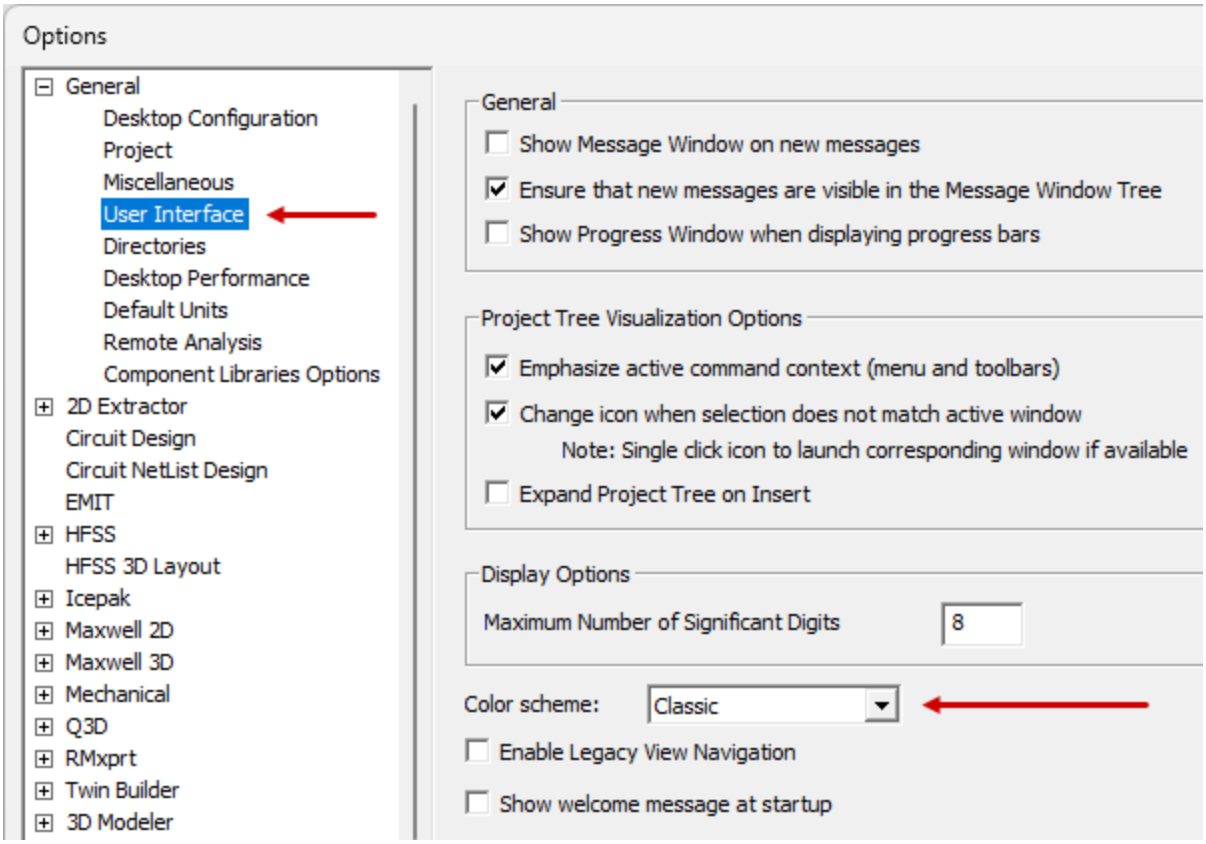


Choosing a Color Scheme [Beta]

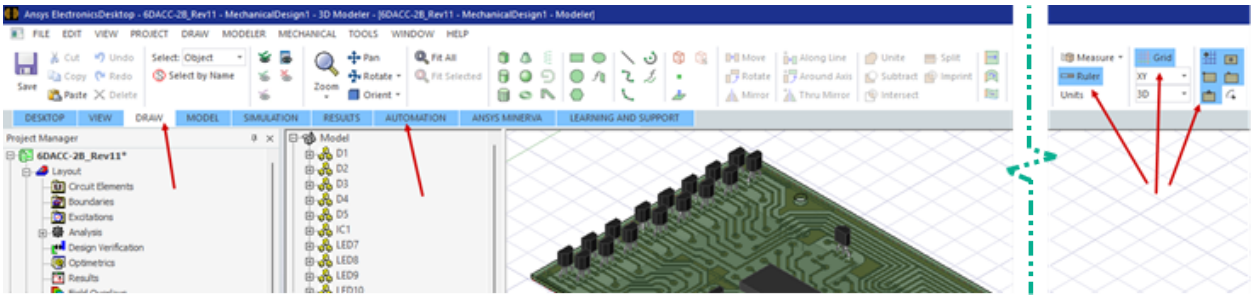
In addition to the classic Ansys Electronics Desktop color scheme, two additional color schemes are now available as beta features. Choose the desired color scheme as follows:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Access the *Options* dialog box using one of the following two methods:
   - On the **Desktop** ribbon tab, click **General Options**.
   - From the menu bar, click **Tools** > **Options** > **General Options**.
2. In the tree at the left side of the dialog box, expand **Desktop Configuration** and select **User Interface**:

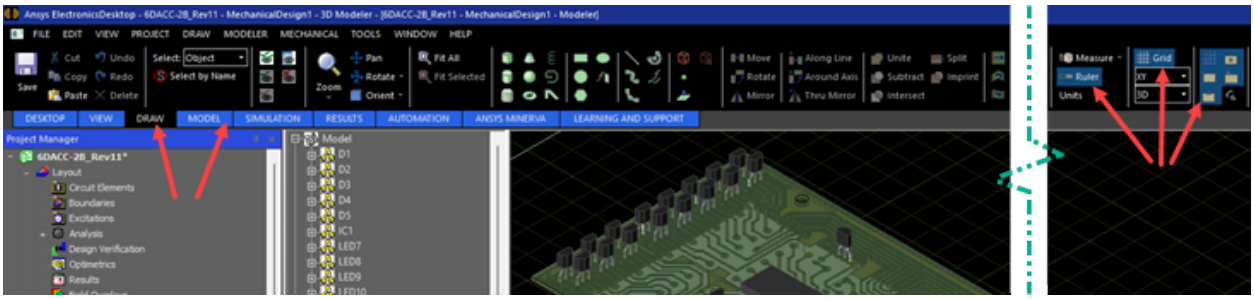


3. From the **Color scheme** drop-down menu, choose one of the following three options:
   - **Light (beta):** A light gray and white scheme with light blue highlighting for selected ribbon icons and for inactive ribbon tabs. The active ribbon has a white background (including the tab):

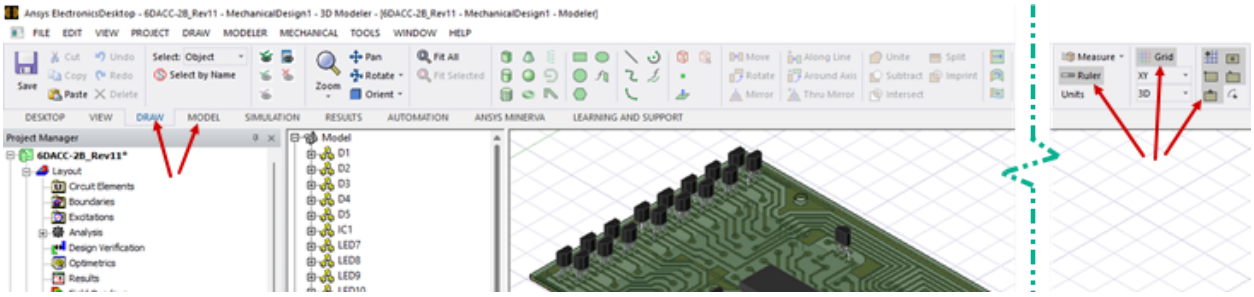


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Dark (beta):** A dark gray and black scheme with medium blue highlighting for selected ribbon icons and for inactive ribbon tabs. The active ribbon has a black background (including the tab):



- **Classic:** A light gray and white scheme with medium gray highlighting for selected ribbon icons. The ribbon background is light gray. The name of the active ribbon tab has a blue font. A dark gray font is used for the inactive ribbon tab names:



4. Click **OK** to close the *Options* dialog box.

Mouse Button Usage

In general, the left mouse button is used to select items (menu commands, objects, properties, *etc*.).

Double-clicking the left mouse button on an item opens a properties window and allows you to visualize settings and edit properties.

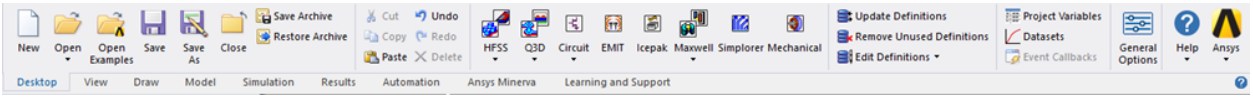
Right-clicking is context-dependent and, in general, allows you to perform context-specific operations (such as assigning boundary conditions or excitations or performing field plots on previously selected geometry entities, exiting the zoom mode or draw mode, *etc*.).

Working with Ribbons

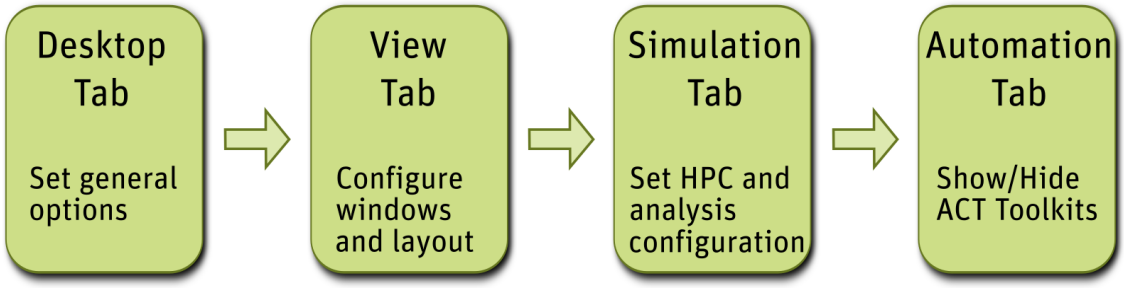
The ribbon is the rectangular area across the top of the application. It comprises various tabs, each one representing a subset of commands available from the menus. The initial set of tabs (Desktop, View, Simulation, Automation) offers commands for adding and opening projects, selecting solvers, configuring the Desktop display (window choice, size, and position),

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

configuring the simulation environment, setting scripted Event Callbacks or General Options, and showing Automation features for recording and using scripts.



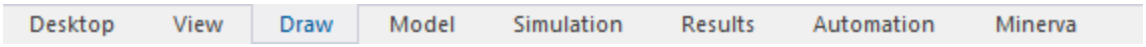
The initial workflow for local configuration and personal customization is:



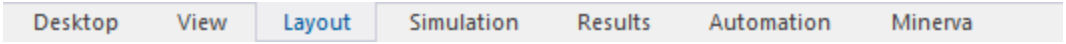
Available ribbon tabs depend on the design type. After you open or add a project and insert a design, you see additional ribbon tabs appear. The visible tabs and features are those that are appropriate for the design type and solver.

The **Desktop, View, Simulation, Automation, Minerva** and **Learning and Support** tabs appear for all design types. Additional tabs appear as follows:

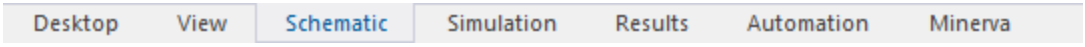
- For HFSS, Icepak, Maxwell, Mechanical, and Q3D designs, the **Draw**, **Model**, and **Results** tabs also appear:



- For an HFSS 3D Layout design, the **Layout** and **Results** tabs also appear:



- For Circuit and Simplorer designs, the **Schematic** and **Results** tabs also appear:



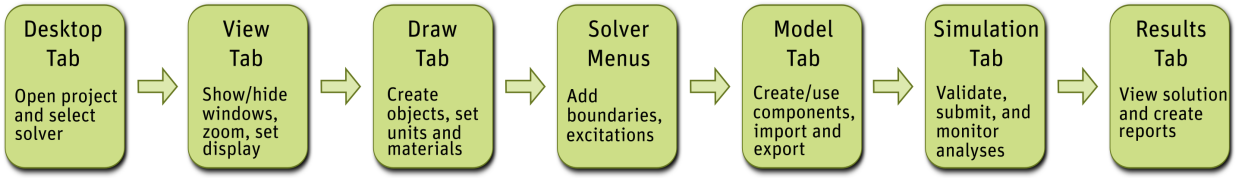
In addition to **Desktop**, **View**, **Minerva** and **Learning and Support** the ribbon tabs that are applicable to Maxwell designs are described as follows:

- **Draw Tab** – contains tools that allow you to draw objects, set units, and assign materials.
- **Model Tab** – contains tools to group and ungroup objects, create 3D components, and import or export geometry.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Simulation Tab** – contains tools that allow you to run simulations, perform validation checks, and schedule and monitor tasks.
- **Results Tab** – contains tools for creating reports and viewing solution data.
- **Automation Tab** – contains tools that allow you to run and record scripts, show or hide ACT extensions (Windows only), and install PyAEDT (Beta).

Each tab contains features specific to the design type, and the general workflow is from left to right.

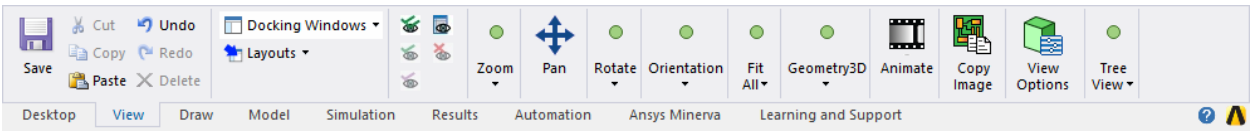


Note that workflow can vary depending on tasks at hand. For example, if you open a completed model in HFSS or use 3D components or the ACT Antenna Design toolkit (Windows only), you may not need the features of the **Draw** tab. If Validation on the **Simulation** tab identifies problems in a model, you may need to use the **Modeler** menu commands for Analysis and Heal, or use the Discovery link feature. For some tasks, such as assigning excitations or boundaries in HFSS, you must use the command menus, rather than the tabs.

Sizing the Ansys Electronics Desktop window affects the icons displayed on each tab, with priority given to the most used features.

For example, If you have inserted an HFSS project, the **View** tab displays commands appropriate for the active editor:

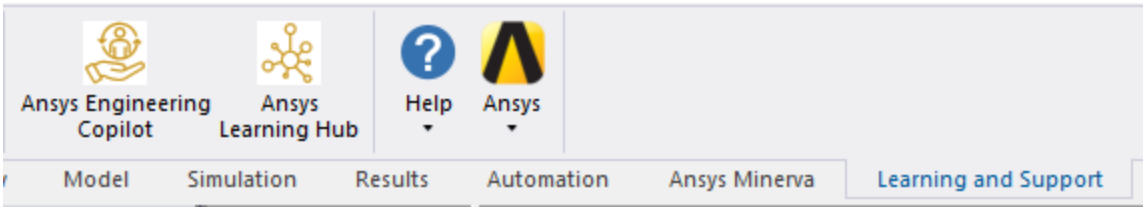
If you reduce the size of the Desktop window, the ribbon tabs become compressed. Fewer icons are shown, and available features are moved into drop-down menus rather than shown separately. The following example shows the **View** tab in a compressed state:



Other ribbon tabs are similarly compressed when the window size does not support the fully expanded arrangement of features.

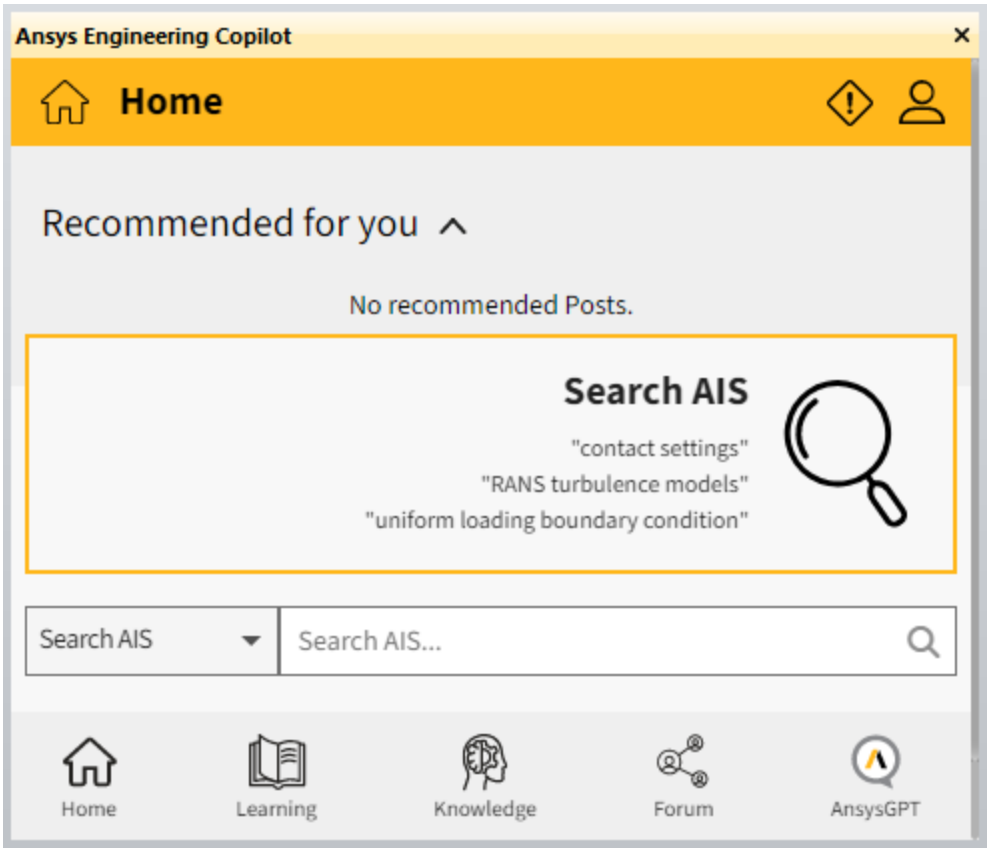
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Learning and Support** tab provides easy access to Technical advice, instruction, and examples on Ansys websites.



Ansys Engineering Copilot – Single point access to Ansys Innovation Space resources, such as AnsysGPT, Support Cases, Courses, Knowledge, Forum, and more.

You first login to your account. Then you are provided with a new window.



- **Ansys Innovation Courses** – Provide a wide range of Ansys Electronics Engineering courses using on-demand, self-paced video training and quizzes.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

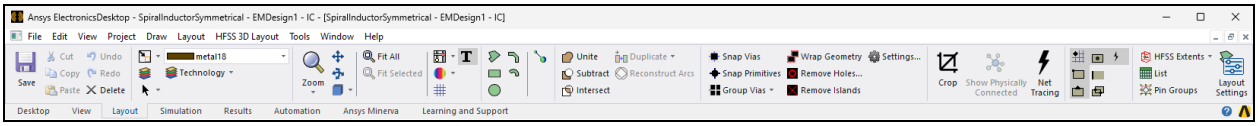
- **Ansys Knowledge** – Provides expert curated knowledge materials from FAQs to tutorials on simulation topics.
- **Ansys Learning Forum** – Provides Ansys blog containing discussion and presentations from Ansys experts, partners and customers.
- **AnsysGPT** – AnsysGPT is a Machine Learning (ML) Large Language Model (LLM) which has been trained in Ansys company information, including online help contents.

Ansys Learning Hub – opens a web page with subscription based access to, virtual and self-paced learning across the Ansys Software portfolio.

About Electronics Desktop - opens a dialog with version and release information, Ansys Electronics Desktop installed components, and licensing information.

IC Mode Layout Ribbon

In addition to exclusive tabs, HFSS 3D Layout contains two potential environments (i.e., General mode and IC mode). General mode typically opens by default. Different commands and options are available in each mode (e.g., after opening a GDSII file or selecting the **IC** layout from the **Design Settings** window (i.e., **HFSS 3D Layout** > **Design Settings** > **Design Mode** tab), the **Layout** tab will reconfigure to match the following screenshot).



The general workflow remains the same in both environments. Refer to Switching to IC Mode.

Ansys Electronics Desktop Windows

Ansys Electronics Desktop contains the following windows:

- Project Manager
- Message Manager
- Progress
- Properties

You can toggle the display of each window, auto-hide windows, or move and resize windows. Additionally, you can toggle the status bar or add ACT Extensions (Windows only).

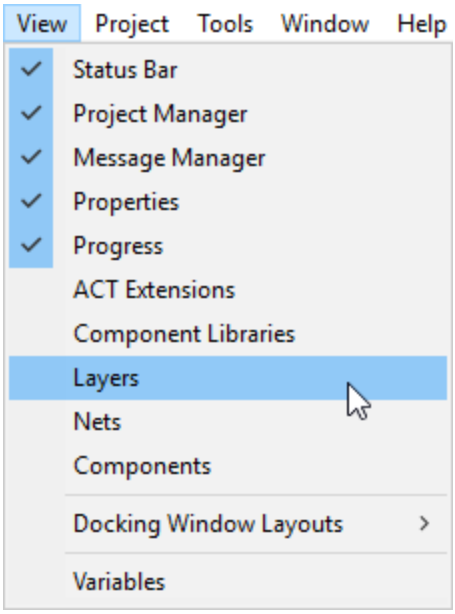
Showing and Hiding Windows

To toggle the windows that comprise Ansys Electronics Desktop, use the **View** menu or the **View** tab on the ribbon.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

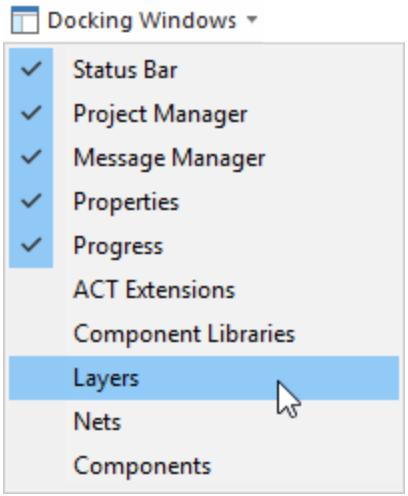
From the **View** menu:

- Click a window to show or hide it.



From the **View** tab:

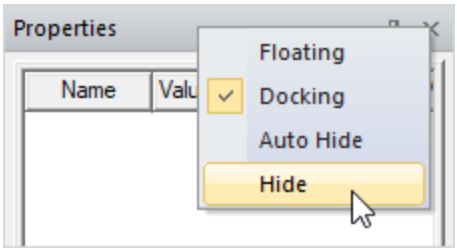
- Use the **Docking Windows** drop-down menu to show and hide windows.



You can also hide a window directly from the window:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click the X in the window title bar, or
- Right-click in the window title bar and select **Hide**.



The visibility setting of a window is retained from one desktop session to the next.

Auto Hiding Windows

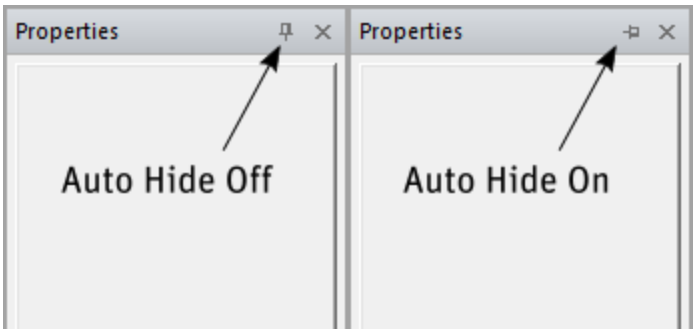
Windows can be moved out of the way so that only their title bars show. When you hover over this title bar, the window appears. This mode is called Auto Hide.

To automatically hide a window until the cursor is hovered over it:

- Click the pin icon in the window title bar, or
- Right-click in the window title bar and select **Auto Hide**.

> **Note:** If a window shares vertical or horizontal space with another window, the first click will expand it to take up the entire area while hiding all other windows. Click the pin again to hide the window.

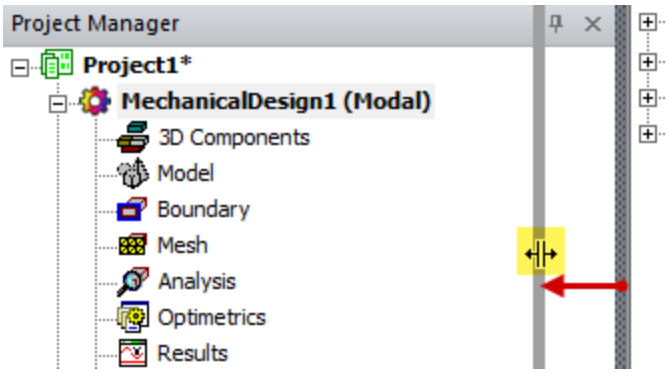
When a window is in Auto Hide mode, the pin icon flips on its side. Click the icon again to disable Auto Hide.



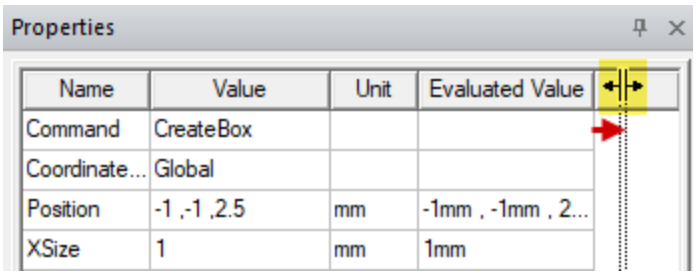
Moving and Resizing Windows

You can resize a window by clicking and dragging a vertical or horizontal window border. The cursor appearance changes to indicate when you're over a movable window border (as highlighted in yellow below):

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can also click and drag the borders of columns within certain windows containing tabular data, such as the **Properties** window:



Major Ansys Electronics Desktop windows can be docked in various locations or can float over other windows. To move a window, click and drag the window's title bar. While dragging a window's location, several docking position icons appear on the screen.

By releasing the mouse button over the following icons, you can dock a window in the following four locations along the perimeter of the user interface:

Left edge of user interface:

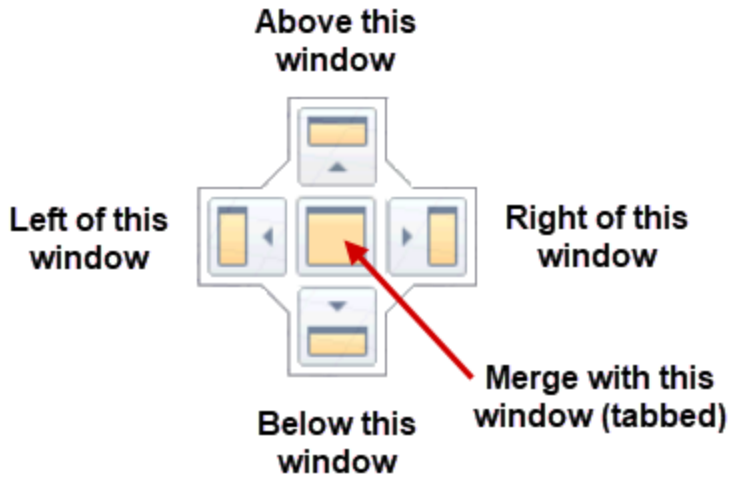
Right edge of user interface:

Top of user interface:

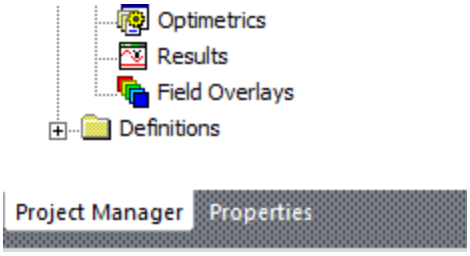
Bottom of user interface:

Additionally, you can use the following group of five docking locations to position the window relative to another window:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When you release the mouse button over the middle icon, the two windows are merged, with tabs at the bottom to select which one to view:



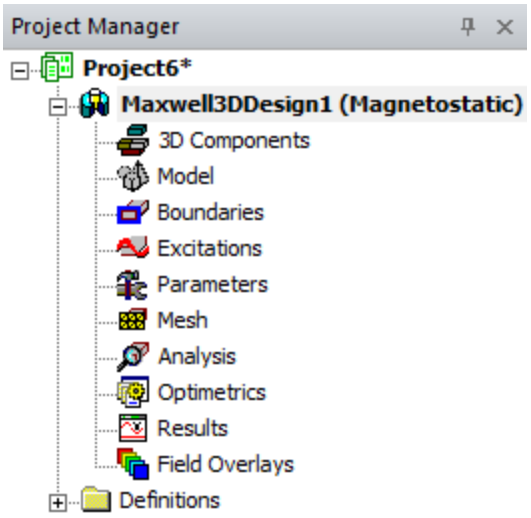
If the middle icon does not appear, it's because tabbed windows are not supported by one of the windows. For example, you cannot merge any window with the Modeler window.

If you release the mouse button while not over any of the above nine icons, the window stays in its current dragged location as a floating window.

Project Manager Window

The **Project Manager** window displays the open project's structure, which is referred to as the project tree.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Project Manager** window displays details about all open Ansys Electronics Desktop projects. The tree display is specific to the design type. For example:

- For any HFSS, Maxwell, or Q3D design, each project ultimately includes a geometric model, its boundary conditions and material assignments, and field solution and post-processing information. Any 3D Components appear under the 3D Components icon.

To show or hide the **Project Manager** window, do one of the following:

- Click **View** > **Project Manager**.

  A check box appears next to this command if the **Project Manager** window is visible.

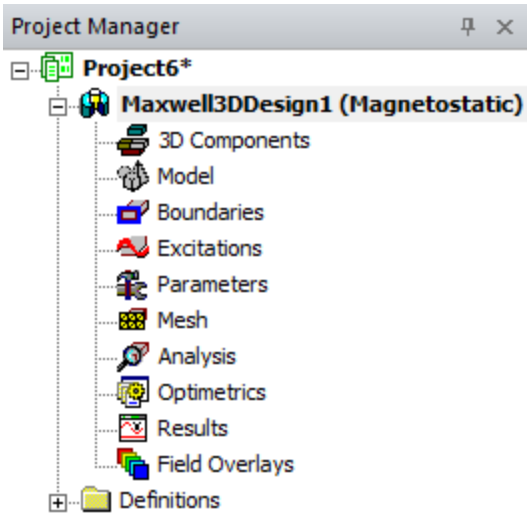
- Right-click in the toolbars area on the desktop, and then click **Project Manager** on the shortcut menu.

  A check box appears next to this command if the **Project Manager** window is visible.

Working with the Project Tree

The project tree is located in the **Project Manager** window and contains details about all open Ansys Electronics Desktop projects, as shown below:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The top node listed in the project tree is the project name. It is named Project*n* by default, where *n* represents the order in which the project was added to the current session of Ansys Electronics Desktop. Expand the project icon to view all the project's design information, material definitions, and 3D Components (if any).

By default, the Project tree icon for the active window is highlighted. See General Options: Miscellaneous for options.

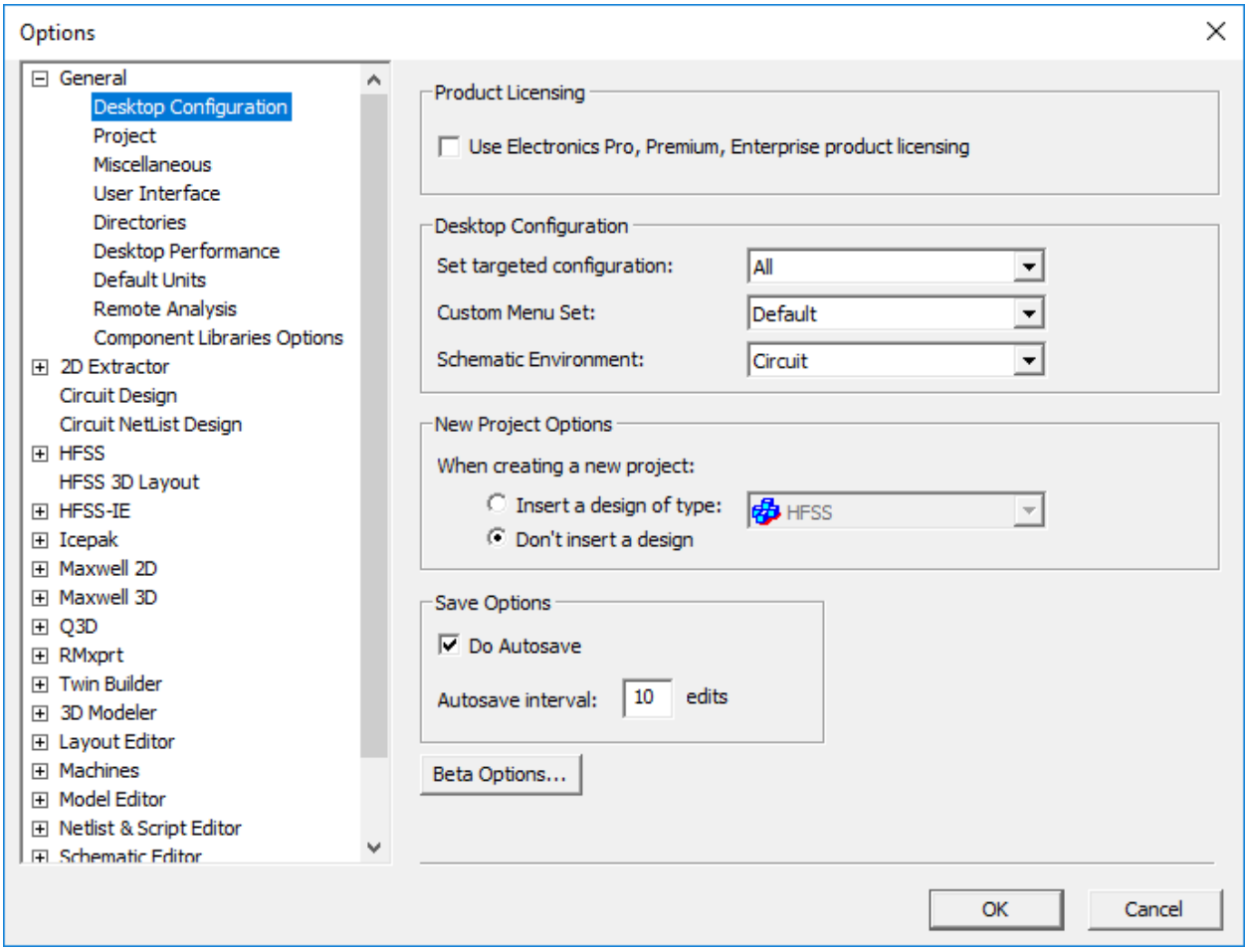
Setting the Project Tree to Expand Automatically

You can set the project tree to automatically expand when an item is added to a project.

1. Click **Tools** > **Options** > **General Options**.

   The **Options** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

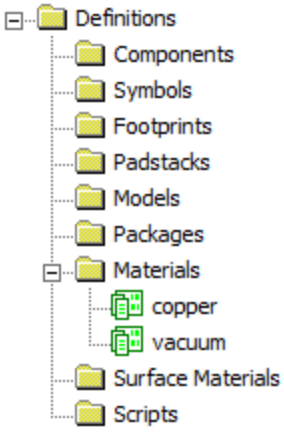


2. Click **User Interface**.
3. In the **Project Tree Visualization Options** area, check the **Expand Project Tree on Insert** check box.
4. Click **OK**.

Viewing Material Definitions

The definitions node is listed at the bottom of the project tree and displays all of the material definitions that are assigned to the objects in the active model.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Viewing Maxwell Design Details

Once you insert a Maxwell design into a project, it is listed as the second node in the project tree. It is named MaxwellModel*n* by default, where *n* is the order in which the design was added to the project. Expand the design icon in the project tree to view specific data about the model.

The MaxwellModel*n* node contains the following project details:

3D Components	Lists any 3D components added to the design.
Model	Allows you to display the design list, define motion setup, and set the symmetry multiplier. For Transient solutions only, when the Model icon is selected, the Properties window displays a Motion Variables tab showing Speed and Direction variables values (not user-editable) if a motion setup has been defined.
Boundaries	Allows you to define 2D and 3D boundary conditions for Maxwell 2D and 3D designs.
Excitations	Allows you to define 2D and 3D excitations for Maxwell 2D and 3D designs.
Parameters	Allows you to assign executive parameters to solve for (such as force and torque).
Mesh	Displays the mesh operations specified for objects or object faces. Mesh operations are optional mesh refinement settings that are specified before a mesh is generated.
Analysis	Displays the solution setups for a Maxwell design. A

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	solution setup specifies how Maxwell computes the solution.
Optimetrics	Displays any Optimetrics setups added to a Maxwell design.
Results	Displays any postprocessing reports that have been generated.
Field Overlays	Displays field overlay plots, which are representations of basic or derived field quantities on surfaces or objects. Plot folders are listed under **Field Overlays.** These folders store the project's plots and can be customized. See Setting Field Plot Defaults for information on how to customize the plot folders.
Documentation	Displays files you have added as documentation.
Definitions	Displays definitions for Maxwell, including Materials.

Note:

To edit a project's design details:

- In the project tree, double-click the design setup icon that you want to edit.

A dialog box appears with that setup's parameters, which you can then edit.

In the project tree, an "unknown" section lists objects that are not solid, sheet, or wire.

Design List Window

You can view the design list for a model in the following ways:

- By clicking **Maxwell 3D**, **Maxwell 2D**, or **RMxprt** and then selecting **List**.
- By double-clicking **Model** in the **Project Tree**.
- By right-clicking items in the project tree and selecting **List** from the shortcut menu.

Information about the model, boundaries and excitations, mesh operations, and analysis setup can be viewed in the **Design List** window that appears, with the following tabs available:

- **Model**
- **Machine** (for RMxprt projects)
- **Motion**
- **Boundaries**
- **Excitations**
- **Parameters**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

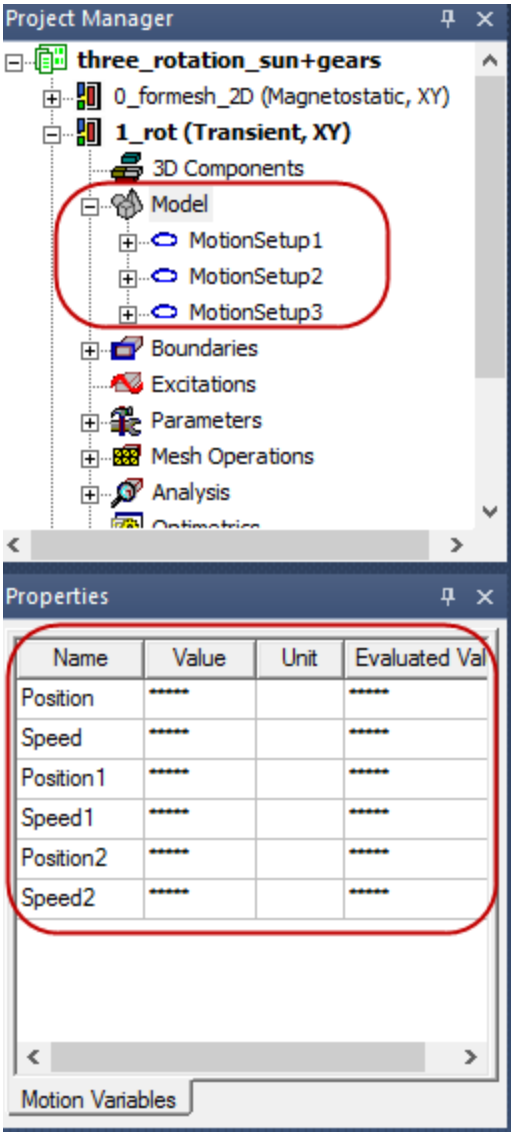
- **Mesh Operations**
- **Analysis Setup**

In the **Design List** window, all objects are shown, regardless of the type of dimensionality (solid, sheet, wire, or unknown); whereas, in the **Project Tree**, an "unknown" section lists objects that are not solid, sheet, or wire.

Motion Variables Tab

For 3D Transient solutions, when the Model icon in the project tree is selected, the **Properties** window displays a **Motion Variables** tab showing **Speed** and **Position** variables values if a motion setup has been defined. Similarly, for 2D transient solutions, which support multiple motion setups, the tab shows speed and position variables corresponding to each motion setup: **Position** and **Speed** for **MotionSetup1**; **Position1**, **Speed1** for **MotionSetup2**; **Position2**, **Speed2** for **MotionSetup3**, and so forth. Below is an example with three motion setups.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Position and speed variables are treated as intrinsic variables and are not user-editable. Their values are dynamically calculated during solve time and are expressed in standard SI units. While these variables are related to motion setups, they are not necessarily used in motion setups. For example, they may also be used with the transient design's excitations. Like other variables, they can be used in expressions and reports.

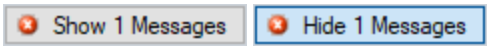
Message Manager Window

The **Message Manager** window displays informational messages about various processes in Ansys Electronics Desktop, including messages related to simulations and any errors they may have produced.

Display or hide the **Message Manager** in one of two ways:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click **View** > **Message Manager**
  A check box appears next to this command if the **Message Manager** is visible.
- On the status bar, click **Show Messages** or **Hide Messages**.



Messages in the **Message Manager** window are organized first by project, then by circuit. Because a design can contain multiple circuits and subcircuits, sometimes with multiple analyses for each, this organization helps you to quickly determine where errors have occurred. The following icons appear next to a message to indicate information, warnings, errors, or actions:

Indicates an informative message.

Indicates a warning message that may require your attention.

Indicates an error message that may require your attention.

Indicates the existence of an action that is associated with the message. Click on the message to invoke the action (the cursor will change to a hand icon when it is placed over the action message).

Right-click in the **Message Manager** window to open a pop-up menu with the following options:

- **Clear messages** for the current model.
- **Copy messages** to the clipboard. This can be helpful for sending messages to application engineers.
- **Details** – opens an information dialog that contains the project and design data for the specified message.
- **Go to Reference** – allows you to right-click on an intersection error message after running a validation check. This selects intersecting objects in the current design being validated.

Setting the Message Manager to Open Automatically

You can set the **Message Manager** to open automatically to show new messages and errors and warnings.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Showing New Messages

You can set the **Message Manager** to automatically open when a new message appears.

1. Click **Tools** > **Options** > **General Options**.

   The **Options** window appears.

2. Expand **General** and click **User Interface**.
3. In the **General** area, check the **Show Message Window on new messages** check box.
4. Click **OK**.

Automatically Expanding the Message Manager Tree

You can set the **Message Manager** Tree to automatically expand when a new message is added.

1. Click **Tools** > **Options** > **General Options**.

   The **Options** window appears.

2. Expand **General** and click **User Interface**.
3. In the **General** area, check the **Ensure that new messages are visible in the Message Window Tree** check box.
4. Click **OK**.

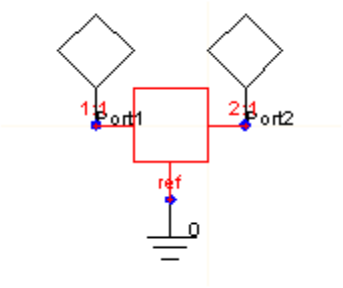
Action Messages

Messages displayed in the **Message Manager** can be associated with actions that can be invoked in order to address a condition you are alerted to by the message. If a message has an associated action, a magnifying-glass icon will be present to the left of the message icon. When the cursor moves over a message with an action, the cursor changes to a hand.

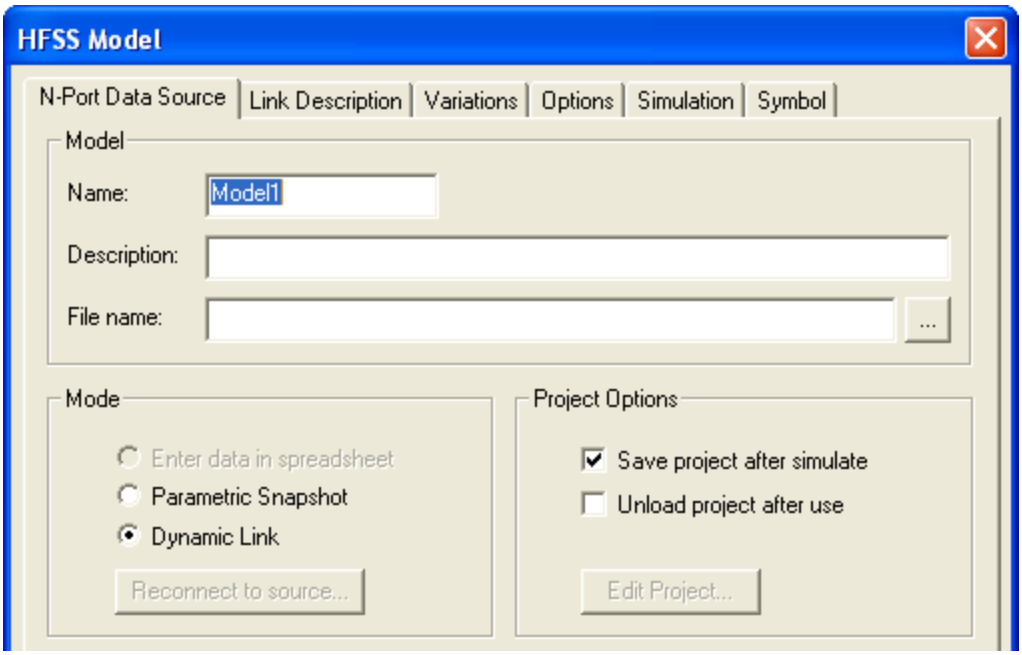
You can invoke a message action by clicking the magnifying glass icon, by double-clicking the message, or by right-clicking the message and selecting **Go To Reference** from the pop-up menu.

One example of a message-associated action is the selection of an associated object:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Another example is the opening of a related window:



Clearing Messages

The **Message Manager** clears at the start of each analysis. To manually clear messages for a project, right-click the message tree and select **Clear Messages for *<ProjectName>***.

Clearing Messages for the Model

You can clear all the messages for a particular model. To clear messages:

1. Right-click the *MaxwellModel#* in the **Message Manager**.

   A pop-up appears.

2. Click **Clear messages for MaxwellModel#**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copying Messages

You can copy all the messages for a particular project. To copy messages:

1. Right-click in the **Message Manager**.

   A pop-up appears.

2. Click **Copy messages to clipboard**.

Hiding the Message Manager Window until Messages Appear

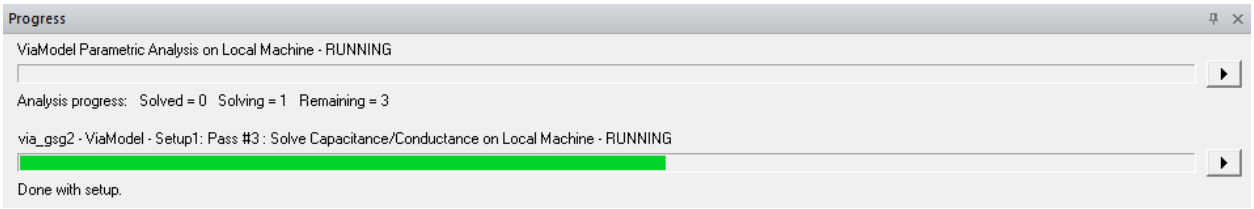
If you prefer to hide the Message Manager until a message is added:

1. Select **View** > **Progress Window** to hide the window. For more information see the View drop-down menu.
2. Click **Tools** > **Options** > **General Options**.
3. Check the **Show message window on new messages** check box.

The **Message Manager** window will re-open when Ansys Electronics Desktop reports errors, warnings, or successful completion of any simulation.

Progress Window

The **Progress** window monitors a simulation while it is running. Each simulation has its own progress bar. Right-clicking the bar allows you to abort the simulation or view simulation details.



To display or hide the **Progress** window, do one of the following :

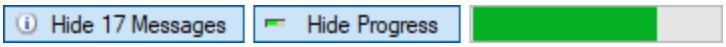
- Click **Show Progress** or **Hide Progress** on the status bar:

- Click **View** > **Progress Window**.
- Right-click the history tree, and then click **Progress** on the shortcut menu.

A check box appears next to this command if the **Progress** window is visible.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

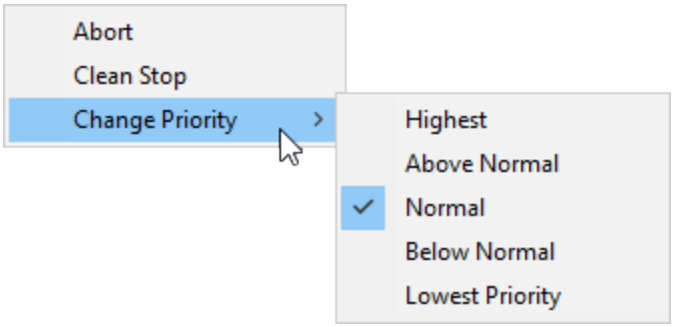
When more than one progress bar is active, the top progress bar is represented on the status bar with a progress indicator.



The progress window is also dockable, so you can reposition it.

Stopping or Aborting Simulation Progress

- To abort progress, right-click in the **Progress** window and select **Abort**.
- To stop the simulation cleanly between time steps, right-click in the **Progress** window and select **Clean Stop**.
- To change priority, right-click. From the **Change Priority** submenu, select from Highest, Above Normal, Normal, Below Normal, and Lowest.



Stopping and Restarting Time Decomposition Method Analyses

- For Time Decomposition Method (TDM) analyses, invoking **Clean Stop** stops the analysis after completion of the current time subdivision, instead of after completion of the current time step. As a result, continuation of the solution process will start from the next time subdivision after clicking **Analysis**. In such a case, the stop time cannot be modified when TDM is used.
- When **Clean Stop** is invoked for projects in which a TDM transient source design is linked to a transient target design, after completion of the source design the solution process can continue from the target design. In such a case, the target design must be non-TDM.

Viewing Distributed Analysis Subtasks

While a distributed analysis is running, you can access parent and child progress bars. By default, only the main progress bar is displayed while the child progress bars (or subtasks) remain hidden. You can toggle between showing and hiding the child progress bars:

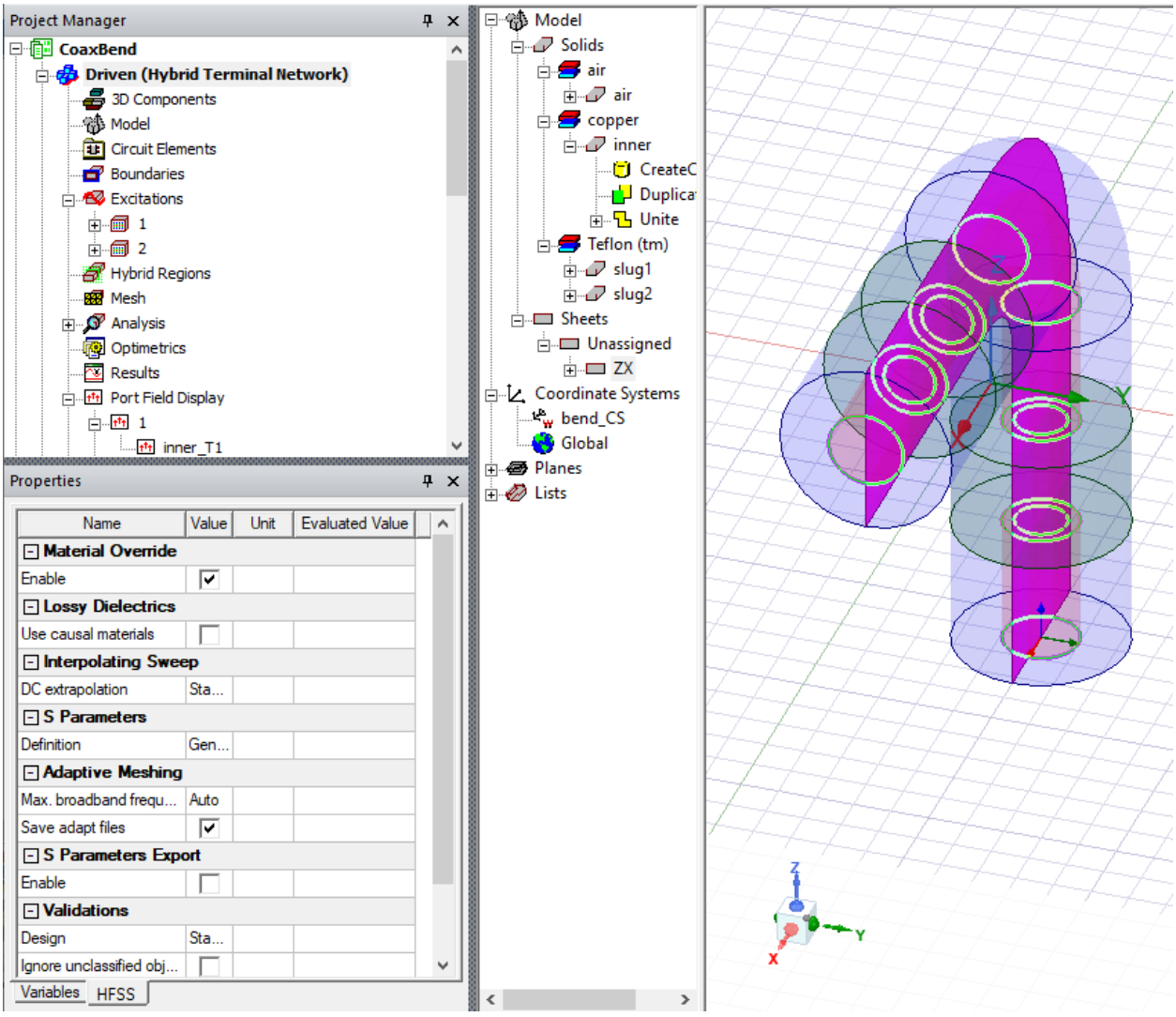
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To show the child progress bars: Right-click the progress window, and select **Show Subtask Progress Bars**.
- To hide the child progress bars: Right-click the progress window, and select **Hide Subtask Progress Bars**.

Properties Window

The **Properties** window displays and allows you to edit the properties of an item selected in the project tree, the history tree, or the **3D Modeler** window. The properties shown and their editability vary depending on the type of item selected.

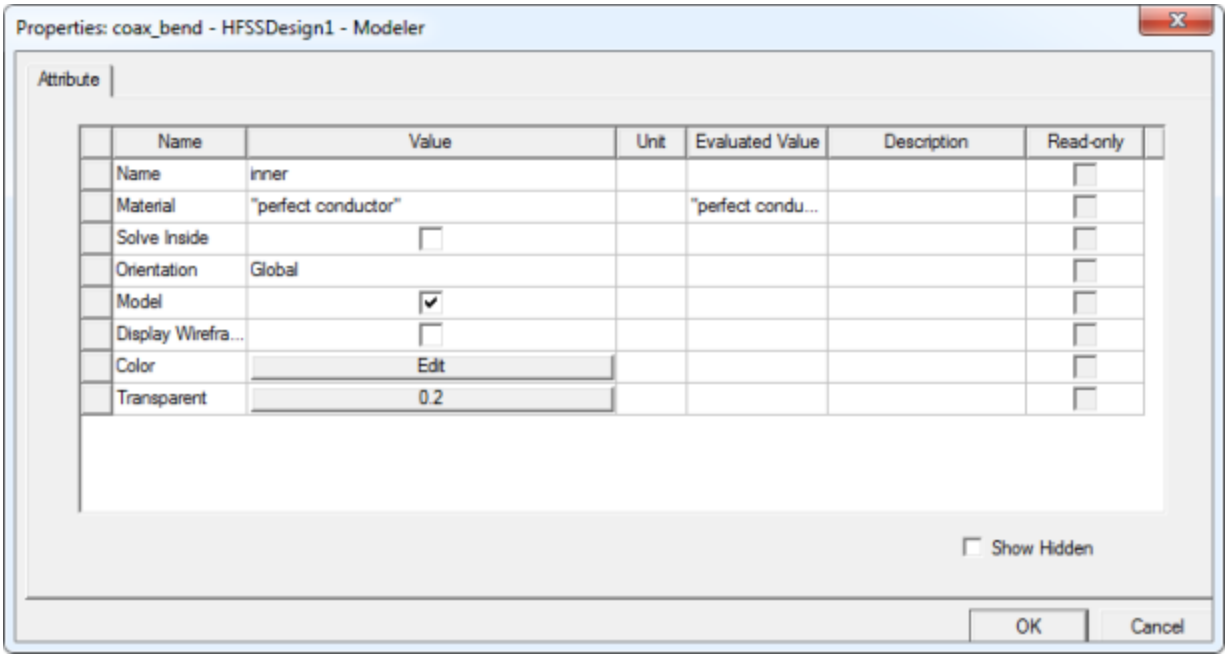
You can choose to show or hide a docked **Properties** window as part of the desktop.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can move and resize the docked **Properties** window within the desktop to suit your work style. When the **Properties** window is docked, it displays the properties of any item you select in the Project tree, the History Tree, or the 3D Modeler window. Select **View** > **Properties** to remove the docked properties window.

Regardless of whether or not you display a docked **Properties** window on the desktop, you can open an undocked **Properties** window for any item in the project tree or history tree by double-clicking the item.



Opening the Properties Window

1. Select the object whose properties you want to view.
2. Click **Edit** > **Properties**.

   The **Properties** window for that object appears.
3. When you are finished making changes, click **OK**.

Rather than opening a separate window, you can have the **Properties** window displayed within the desktop.

Setting the Properties Window to Open Automatically

To set the **Properties** window to open after an object is drawn, do the following:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Tools** > **Options** > **General Options**.

   The **Options** window appears.

2. Expand **3D Modeler** and select **Drawing**.
3. Check the **Edit property of new primitives** check box.

Hereafter, when you draw an object in point mode the **Properties** window opens. However, if you draw an object in Dialog mode, this setting is ignored.

Showing and Hiding the Properties Window

To show or hide the **Properties** window on the desktop, do one of the following:

- Click **View** > **Properties**.

A check box appears next to this command if the **Properties** window is visible.

- Right-click in the toolbars area at the top of the desktop, and then click **Properties** on the shortcut menu.

A check box appears next to this command if the **Properties** window is visible.

Modifying Object Attributes Using the Properties Window

1. Select the object for which you want to edit attributes by clicking it in the view window or clicking its name in the history tree.
2. Under the **Attribute** tab in the **Properties** window, edit the object attribute.

Depending on the attribute type, you can edit it by doing one of the following:

- Select the check box to apply the attribute; clear the check box to disable the attribute.
- Click in the field and edit the numeric values or text, and then press **Enter**. You can modify names, but names must include only letters, numbers and underscores. Illegal names are not accepted and generate a message in the Message Manager window.
- Click the button and edit the current settings in the window that appears.
- Click the attribute and select a new setting from the menu that appears.

Modifying Object Command Properties Using the Properties Window

The **Command** tab in the **Properties** window displays information about an action selected in the history tree that was performed either to create an object (such as the **Draw** > **Box** command) or to modify an object (such as the **Edit** > **Duplicate** > **Mirror** command).

Not all command properties can be modified. In general, command properties you can typically modify are numeric values, such as position values, size values, and various other coordinate values. You can also modify many of the unit settings for a command property. You can modify

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

names, but names must include only letters, numbers and underscores. Illegal names are not accepted and generate a message in the Message Manager window.

1. In the history tree, select the command for which you want to edit properties.

   > **Tip:**
   >
   > Press and hold **Ctrl** to select multiple commands. If you select multiple commands, only the common (shared) properties will display under the **Command** tab.

2. Under the **Command** tab in the **Properties** window, edit the command's properties.

Depending on the property type, you can edit it by doing one of the following:

- Select the check box to apply the property; clear the check box to disable the property.
- Click in the field and edit the numeric values or text, and then press **Enter**.
- Click the button and edit the current settings in the window that appears.
- Click the attribute and select a new setting from the menu that appears.

Property Window for Schematic and Layout Editors

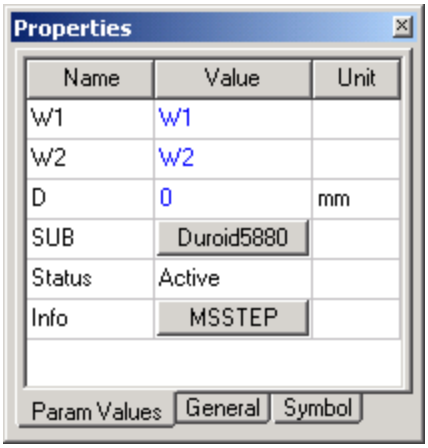
The **Property** window shows the properties or parameters of objects selected in the schematic and layout editors, and objects in the **Project Manager** window. It allows you to modify property values. The **Property** window has three tabs:

- The Param Values Tab
- The General Tab
- The Symbol Tab

Param Values Tab

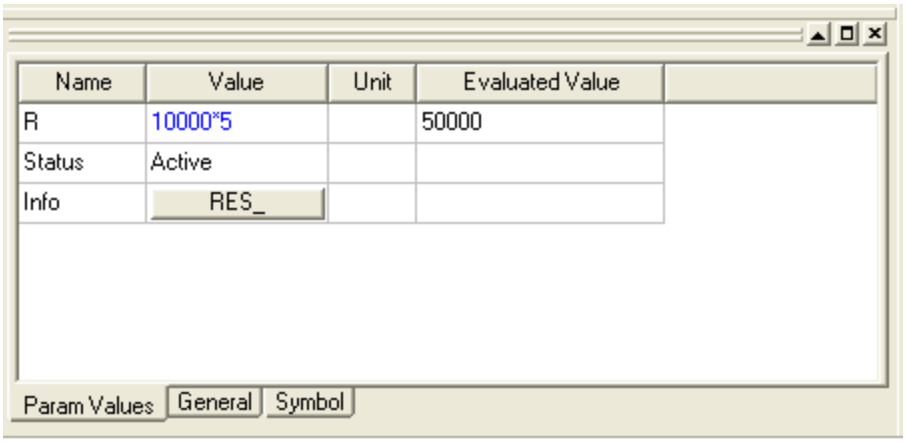
The **Param Values** tab in the **Property** window of the Ansys Electronics Desktop lists the simulation parameters of the component or components selected.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To set or change the value of a component parameter, click on the **Value** field and enter the new value. If the value requires a multiplier unit (such as **K** for 1000) click on the **Unit** field to select the multiplier unit. The Evaluated value shows the resulting number.

You can specify a parameter value using an expression that evaluates to a constant. The expression is retained in the **Value** field, while the **Evaluated Value** field shows the constant resulting from evaluation the expression. This allows you to identify and modify the expression in the future.



Launching Help from the Property Window

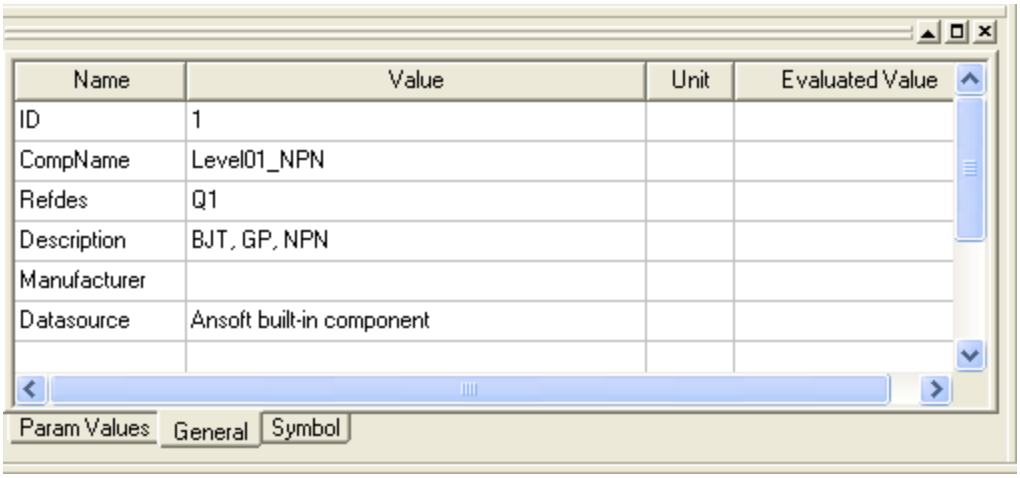
You can launch the online help for a component from the **Property** window. To do this:

1. Click the **Param Values** tab.
2. Scroll down to the **Info** parameter entry, and then click the button in its **Value** cell.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Tab

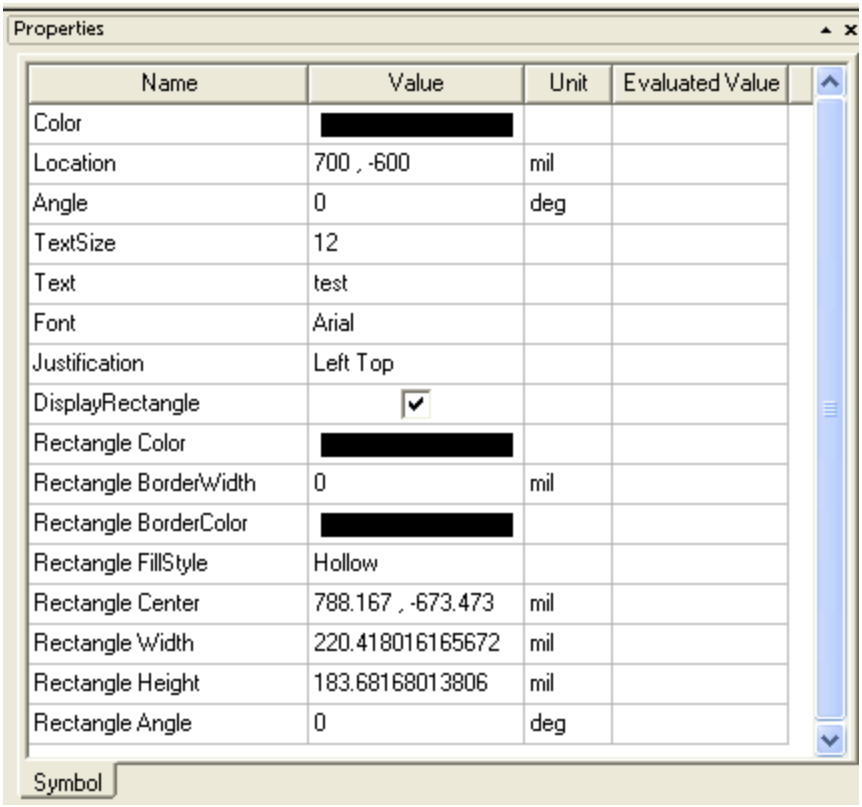
The **General** tab in the **Property** window of the Ansys Electronics Desktop lists the selection's name, symbol name, reference designator, and other data. These are generally not editable.



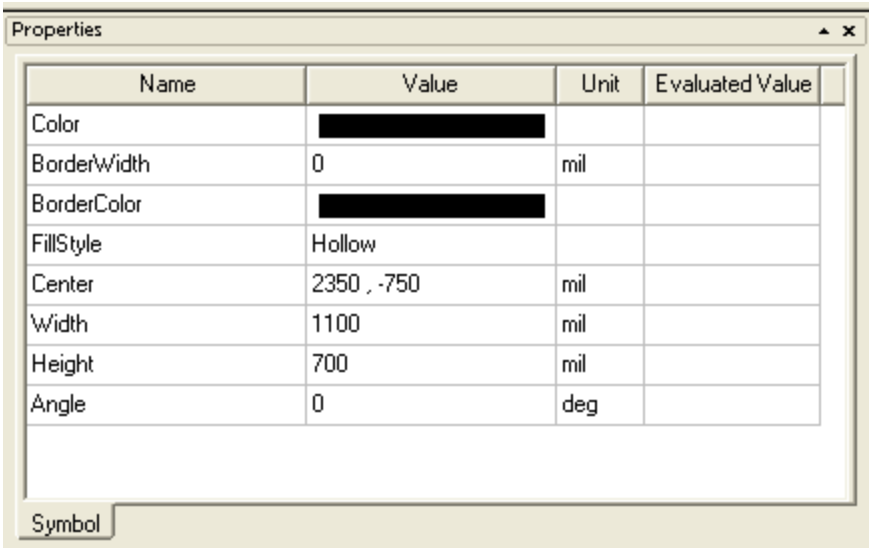
Symbol Tab

The **Symbol** tab in the **Property** window of the Ansys Electronics Desktop provides information on a number of modifiable attributes of components and symbols displayed in the schematic.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

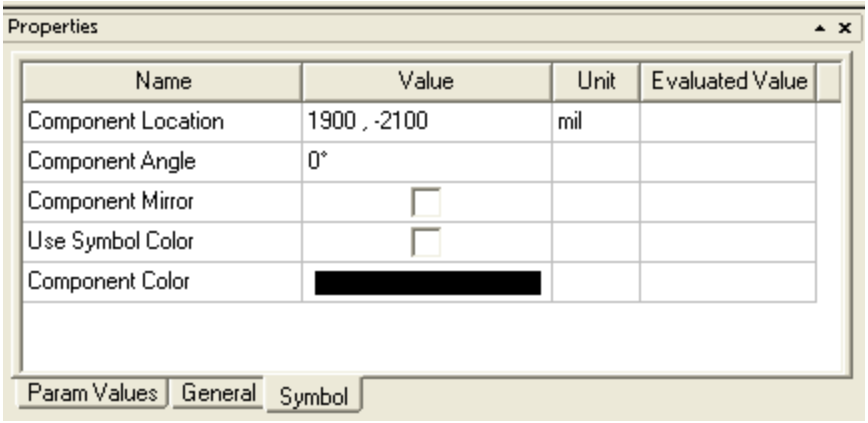


The contents of the **Symbol** tab varies depending upon the number and type of components or symbols selected in the schematic.



But whatever components or symbols are selected, each field displayed in the **Symbol** tab is modifiable.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Symbol tab Value fields can be modified using the following guidelines:

- Click in the **Value** field for **Component Location** to enter a new set of X,Y coordinates for the symbol. Press **Return** to move the symbol to the new location.
- Click in the **Value** field for **Component Angle**, and select an angle from the drop-down menu (choices are 0, 90, 180, and 270 degrees). The symbol rotates as you select an angle.
- Click in the check box in the **Value** field for **Component Mirror** to flip the component left-to-right. The mirror operation is performed as soon as you check the box. Uncheck the box to return the symbol to its original orientation.
- Click in the **Value** field to enter a new value for whatever symbol attribute you wish to change. Press **Return** to move the symbol to the new location.
- Click the colored bar displayed in the **Value** field to open a palette from which to select a new color for the symbol. The new color is applied when the symbol is unselected.

Closing the Property Window

To close the **Property** window, click **View** > **Properties** to deselect it. You can repeat this step to reopen the window.

For more information, see the View Pulldown Menu.

Properties Dialog Box

Like the **Property** window, the **Properties** dialog box shows the properties or parameters of selected objects, and, where appropriate, allows editing the values of these properties. It extends **Property** window functions with additional editing commands, and with settings for tuning, optimization, sensitivity, and statistical analysis, that are not available through the **Property** window. It is also used to define project variables. The **Properties** dialog box has the following tabs:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Parameter Values
- General
- Symbol
- Property Displays

Opening the Properties Dialog Box

You can open the **Properties** dialog box in two ways:

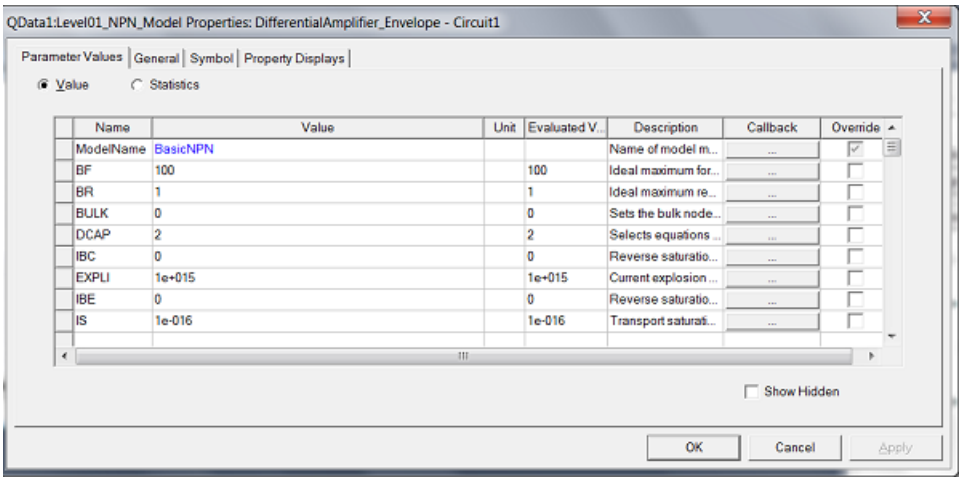
- To open the **Properties** dialog for a component, double-click the component.
- Right-click the component and select **Properties**.

You can manipulate the **Properties** dialog box in the following ways:

- Close the **Properties** dialog box by clicking the **Esc** key.
- Resize the **Properties** dialog box by dragging its edges.
- Change the relative widths of adjacent columns by dragging the header separators between them.

Parameter Values Tab

The **Parameter Values** tab in the Properties Dialog Box lists the simulation parameters of the component or components selected.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To set or change the value of a component parameter, click on the **Value** field and enter the new value. You can enter the units in the value field, or you can click on the **Unit** field to enter the units.

You can specify a parameter value using an expression that evaluates to a constant (as shown in the picture above). The expression is retained in the **Value** field, while the **Evaluated Value** field shows the constant resulting from evaluation the expression. This allows you to identify and modify the expression in the future. The Property Displays Tab allows you to display the expression, the evaluated value, or both.

When the **Properties** dialog box is opened for a design (**Design Properties**), the **Parameter Values** tab is initially empty. You can add properties to the design by clicking the **Add** button. Note that the **Add** button does not appear on the **Properties** dialog box for a component. See: Project Variables.

> **Note:**
>
> Clicking the **Show Hidden** check box allows you to view hidden properties of a component, and may result in the display of complex-value computations upon which some calculated values are based. Hidden properties contain system-defined values and rules for interpreting predefined component parameters. Modifying hidden properties requires specialized knowledge of the component, and is not needed for normal operation.

General and Symbol Tabs

The **General** tab in the Properties Dialog Box lists the selection's name, symbol name, reference designator, and so on. These are generally not editable. The information is identical to that on the General tab of the Property Window.

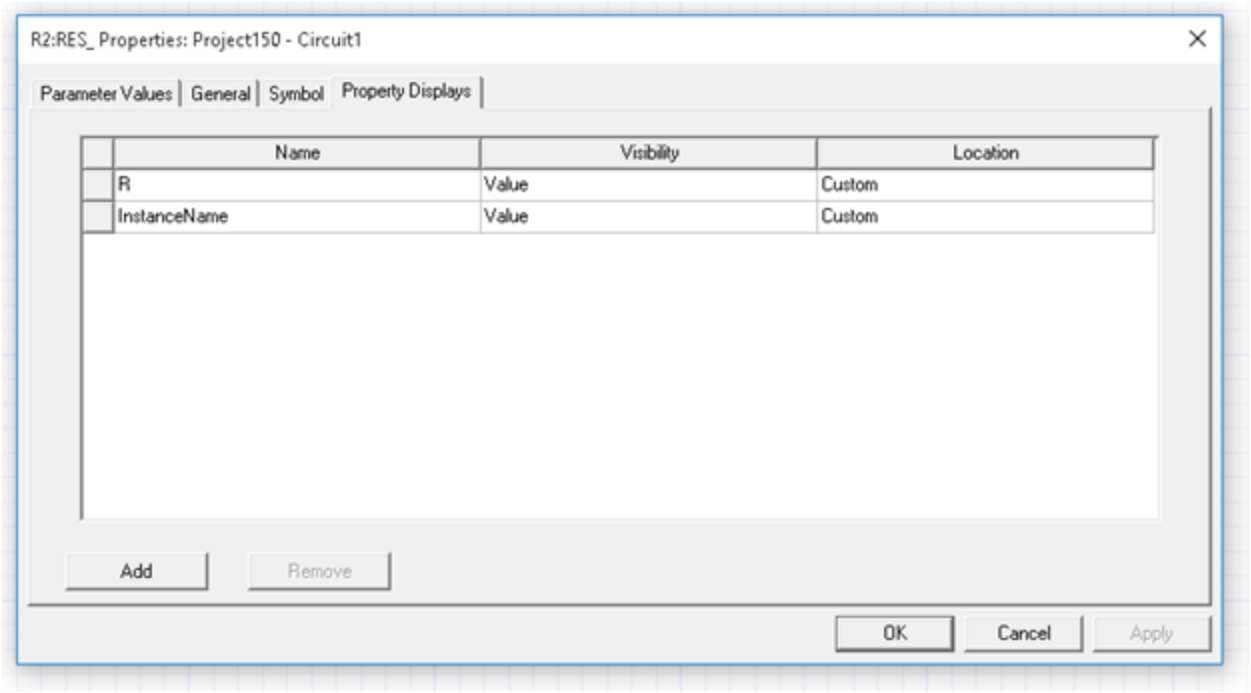
The **Symbol** tab provides information on the location of the component symbol in the schematic. The information is identical to that on the Symbol tab of the Properties Window.

When the **Properties** dialog box is opened for a design (**Design Properties**), both the General and Symbol tabs are initially empty.

Property Displays Tab

The **Property Displays** tab in the Properties Dialog Box controls how the properties of the component appear on the schematic.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Click in the **Visibility** field for a parameter to select how the information for that parameter is displayed:

- **None** – results in no label being shown.
- **Name** – displays the parameter name (e.g., R).
- **Value** – displays the component value, which can be a single value (e.g., 10000) or an expression (e.g., 10000*5).
- **Both** – displays the parameter name and its value (e.g., R = 10000, R=10000*5).
- **Evaluated Value** – displays the evaluated value of an expression that has been used for the value of a parameter (e.g., 50000 for the expression 10000*5).
- **Evaluated Both** – displays the parameter name and the Evaluated Value (e.g., R = 50000 for the expression 10000*5).

Click in the **Location** field for a parameter to specify the location for the display. The locations are: **Left**, **Top**, **Right**, **Bottom**, **Center**. When you have set the location with the cursor in the schematic, the **Location** field has the entry **Custom**.

- To add a parameter to the display, click the **Add** button. Click on the **Name** field and select the parameter to add from the drop-down menu.
- To delete a parameter from the display, select it in the **Property Display** list and click **Remove**.

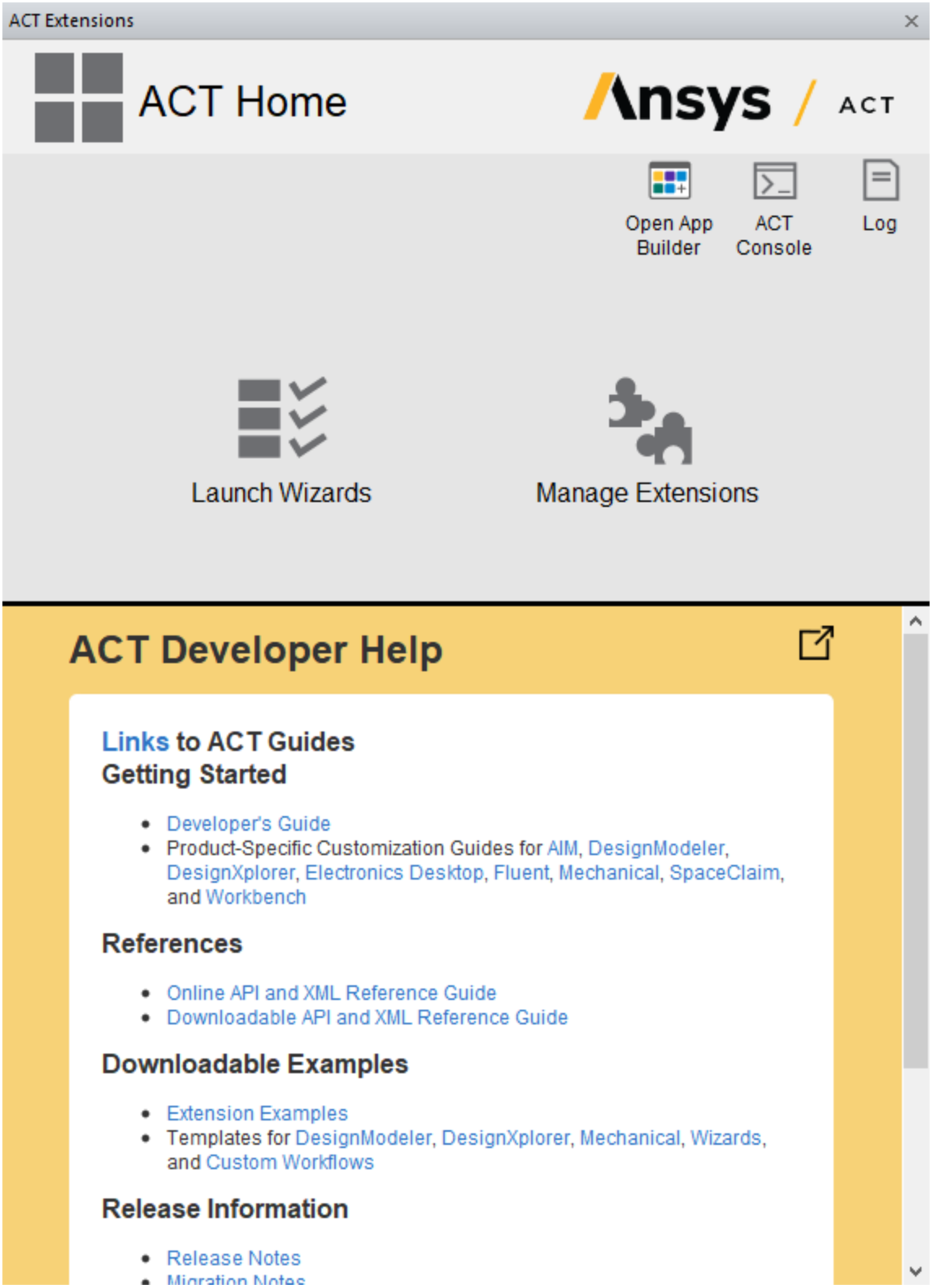
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ACT Extensions Window

> **Note:**
>
> The **ACT Extensions** window and the design wizards it contains (*5G Wizard, HFSS Antenna Design Toolkit, HFSS-EMA3D Link,* and *Maxwell Eccentricities*) are only available for the Windows version of the Ansys Electronics Desktop software. These items are not available when the software is installed on a Linux platform.

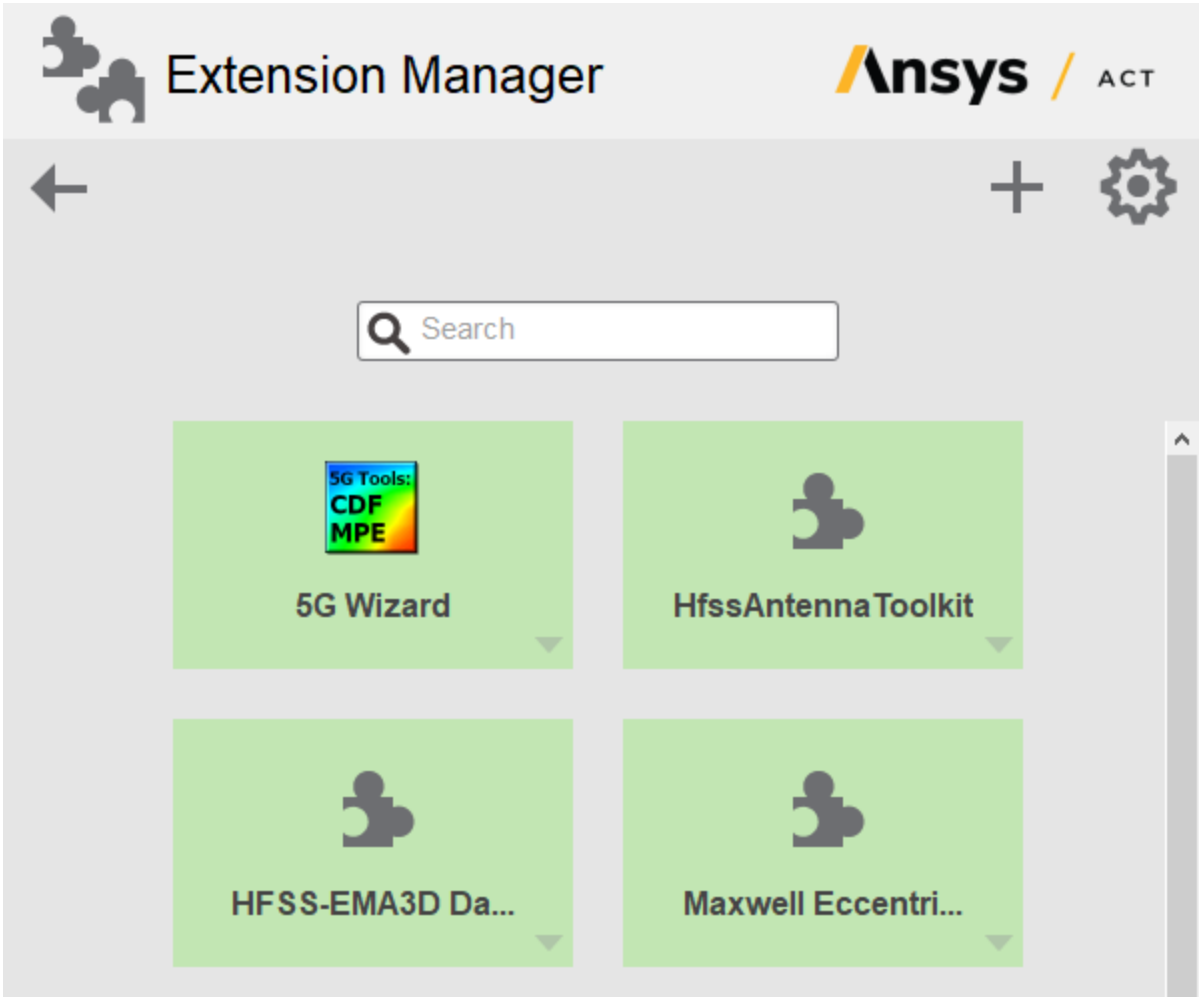
The **View** > **ACT Extensions** command opens the ACT Extensions window for toolkit extensions implemented via Ansys ACT. Depending on how you size and arrange the desktop, the appearance may vary. You may choose to detach and size the window to suit your needs.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



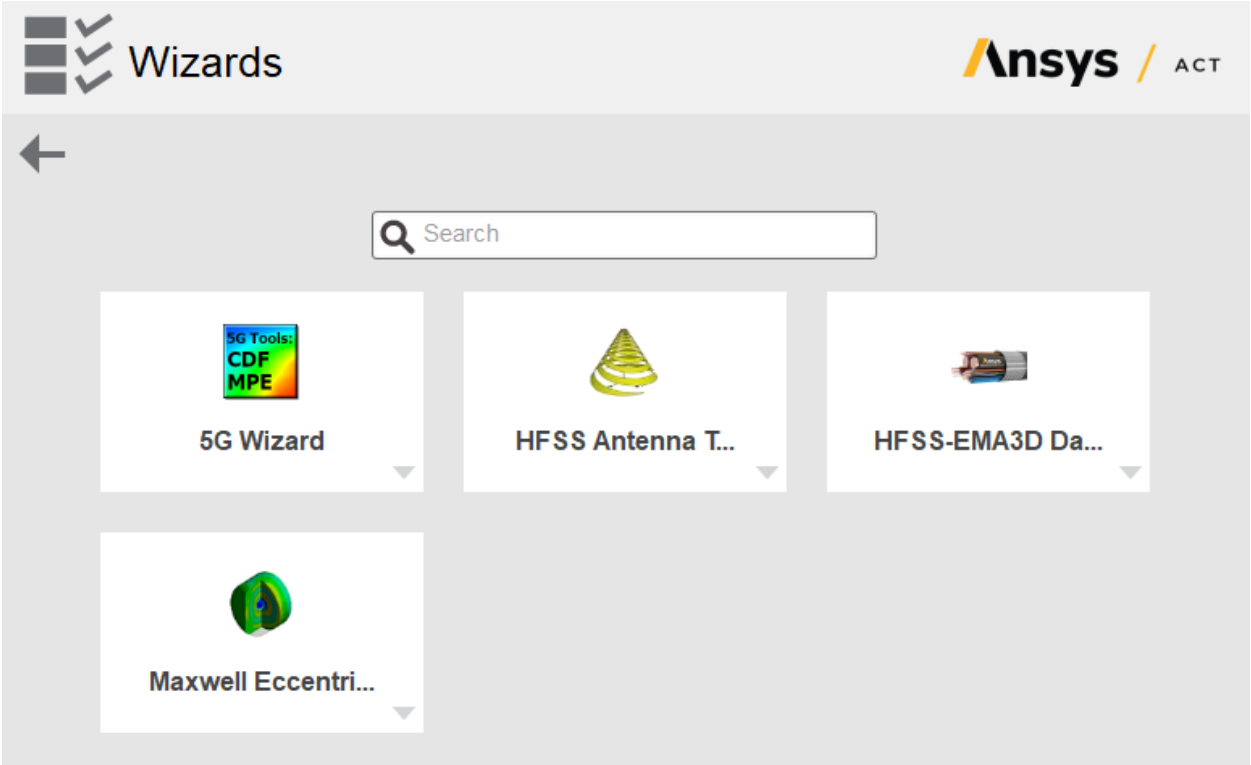
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Click the **Manage Wizards** icon to load the **Extension Manager** to select from available extensions. Active extensions are highlighted. Selectable extensions show while. Scroll bars and window sizing lets you view addition extensions. You can search, or access the Ansys App Store.



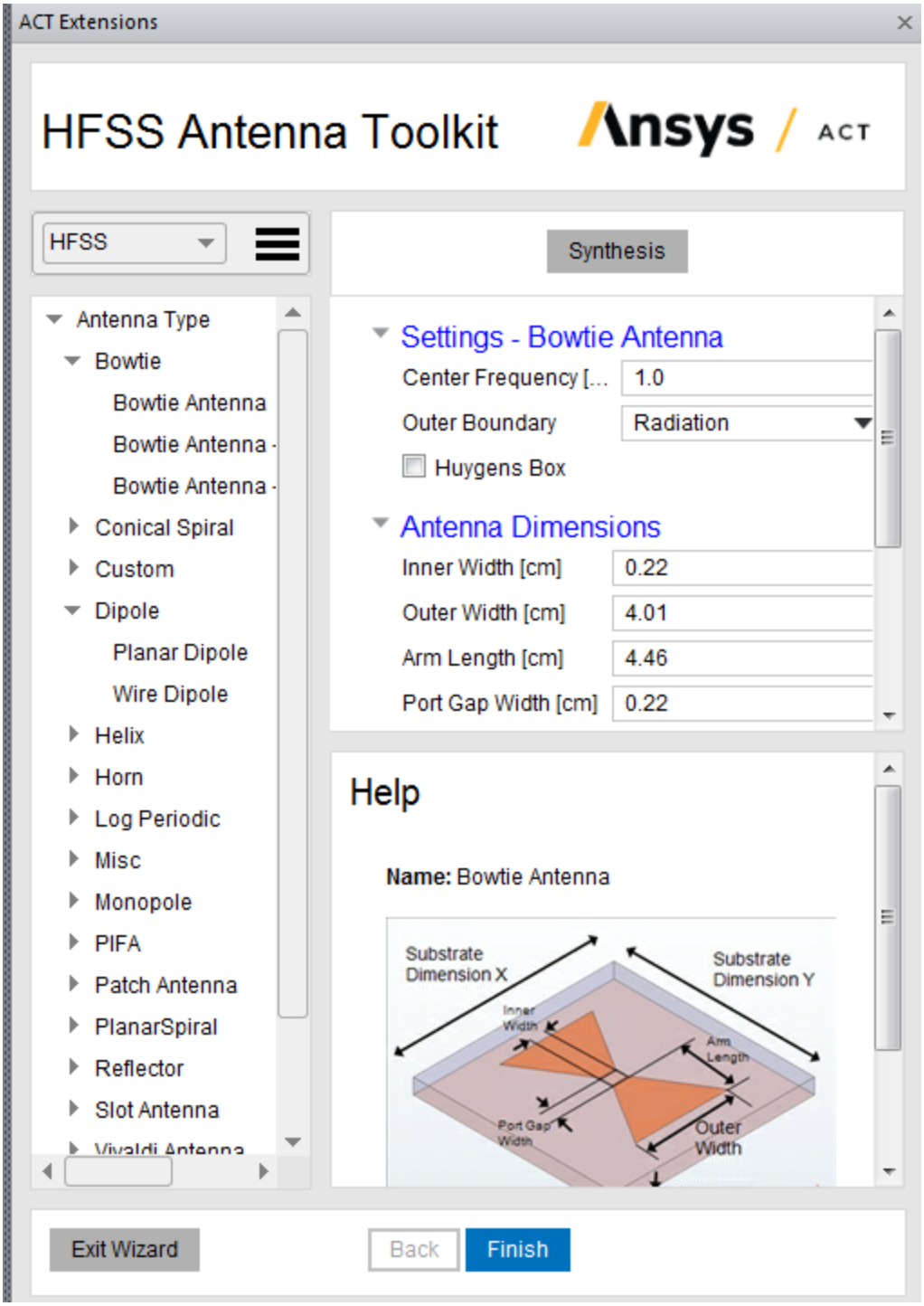
Click the **Launch Wizards** icon to load existing toolkits for activation. If there are many available extensions, you can use the **Search** bar. Search is not case sensitive.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Click the desired ACT extension to load it. The selected Wizard operates in the window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electronics Desktop Menus

The menu bar enables you to perform all Ansys Electronics Desktop tasks, such as managing project files, customizing the desktop, drawing objects, and setting and modifying project parameters.

Ansys Electronics Desktop contains the following menus, which appear at the top of the desktop:

File	Contains commands to manage Ansys Electronics Desktop project files and printing options.
Edit	Contains commands to modify the objects in the active model and undo and redo actions.
View	Contains commands to display or hide desktop components and model objects, modify 3D Modeler window visual settings, and modify the model view.
Project	Contains commands to add a specific design type to the active project; view and define datasets, project variables, and event callbacks.
Tools	Contains commands to modify the active project's material library, arrange the material libraries, run and record scripts, update project definitions from libraries, customize the desktop's toolbars, and modify many of the software's default settings.
Window	Contains commands to rearrange the 3D Modeler windows and toolbar icons.
Help	Contains commands to access the help system and view the current Ansys Electronics Desktop version information.

Once you have inserted a design type, the menu bar also includes menus specific to that type. These may include:

Draw	Contains commands to draw one-, two-, or three-dimensional objects, and sweep one- and two-dimensional objects.
Modeler	Contains commands to import, export, and copy 2D Modeler files and 3D Modeler files; assign materials to objects; manage the 3D Modeler window's grid settings; define a list of objects or faces of objects; control surface settings; perform boolean operations on objects; and set the units for the active design.
Ansys Electronics Desktop	Contains commands to set up and manage all the parameters for the active project. Most of these project properties also appear in the project tree.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Bar

The **Menu Bar** appears at the top of the screen immediately beneath the Ansys Electronics Desktop title bar. It contains menus used for controlling the application and the various editors and viewers used to construct the models, set up the analyses, and postprocess the simulation results. The menu bar and the command ribbon contain many of the same commands and options. However, the menu bar provides the more comprehensive set of commands and options. The ribbon is more visual and generally requires fewer mouse clicks to navigate, but you can navigate the menu bar without using a mouse.

Click on a **Menu Bar** entry to open its menu. Alternatively, the **Alt** key makes it easy to access the menus without using a mouse, as follows:

- You can select a menu in the **Menu Bar** by pressing the **Alt** key and then pressing the underlined letter of the menu you wish to activate (the underlines appear when the **Alt** key is pressed). Background highlighting indicates which menu is selected. Finally, press **Enter** to open the menu.
- After pressing the **Alt** key, you can change the selected menu by pressing the left (⇐) and right (⇒) arrow keys. Again, press **Enter** to open the desired menu.

Once a menu is open, you can press additional letters to drill down within the menu. Alternatively, use the down (⇓) and up (⇑) arrows to change the selection. Use the right (⇒) arrow key to open a subordinate menu for the selected command. As always, press **Enter** to execute the selected command.

Order of Menus

When a project is loaded, the Ansys Electronics Desktop drop-down menus (from left to right) are:

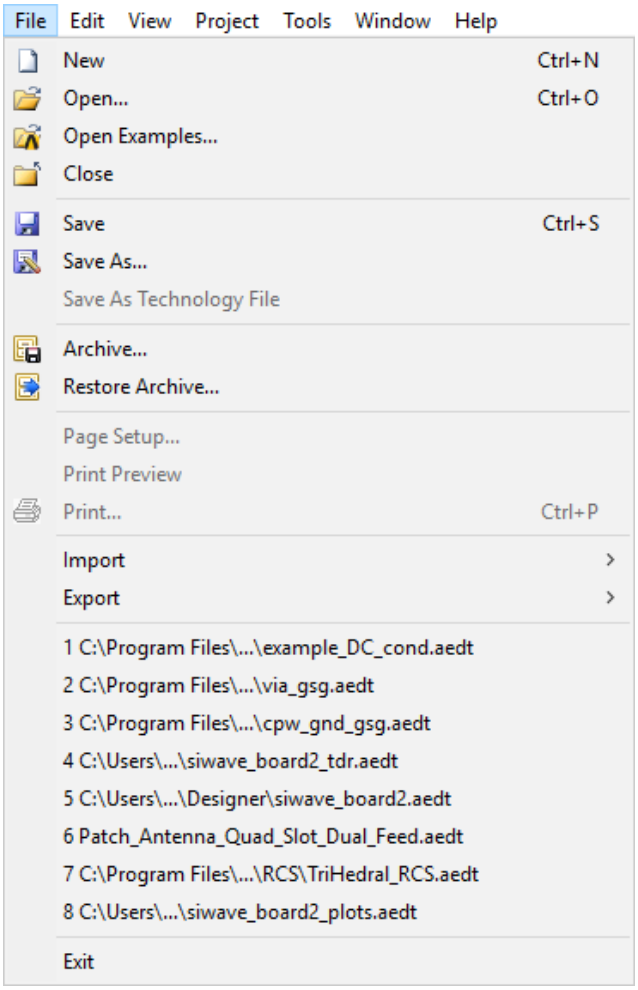
File Edit View Project Draw [Modeler | Layout | Schematic](1) <*Design-Type*>(2) Tools Window Help

> **Note:**
> 1. These three menu bar entries depend on the active design type:
>   - **Modeler** appears for all designs that use the *Modeler* window (including all 3D types, 2D Extractor, and Maxwell 2D).
>   - **Layout** appears only for HFSS 3D Layout designs.
>   - **Schematic** appears for Circuit and Maxwell Circuit designs.
> 2. The name of the active *Design-Type* appears in this position, and the menu contains the design-type-specific commands.

These menus are described in the following sections.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File Menu



File menu items are common operations on files and projects. You can search the Help to find information on any of the commands that appear on Ansys Electronics Desktop menus.

- Click **New** (or type **n**) to set up a new project.
- Click **Open** (or type **o**) to open an existing project.
- Click **Open Examples** to open an example project.
- Click **Close** (or type **c**) to close the selected project. If the project has changed since the last save, you will be prompted to save the project before closing.
- Click **Save** (or type **s**) to save the selected project.
- Click **Save As** (or type **a**) to save the selected project under a different name or in a different directory.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click **Save As Technology File** (or type **t**) to save the selected top-level design as a technology file (*.asty). You must have a design selected to activate this menu item. You cannot export a technology file from a project or from a subcircuit.
- Click **Archive** to archive the selected project and any selected additional files. If the project has changed since the last save, you will be prompted to save the project before closing.
- Click **Restore Archive** (or type **r**) to restore a previously archived project.
- Click **Page Setup** (or type **u**) to set up formatting to print the active window in the Design Area.
- Click **Print Preview** (or type **v**) to display a preview of the print job.
- Click **Print** (or type **p**) to print the active window.
- Click **Import** to open a submenu and select a file for import.
- Click **Export** to open a submenu and select a file for export.
- Click the name of a project (or type its number) to open a project from the listing of recently opened projects.
- Click **Exit** (or type **x**) to exit Ansys Electronics Desktop. If any project has unsaved changes, you will be prompted to save the project before closing.

Edit Menu

The **Edit** menu changes, depending on the active window in the Design Area.

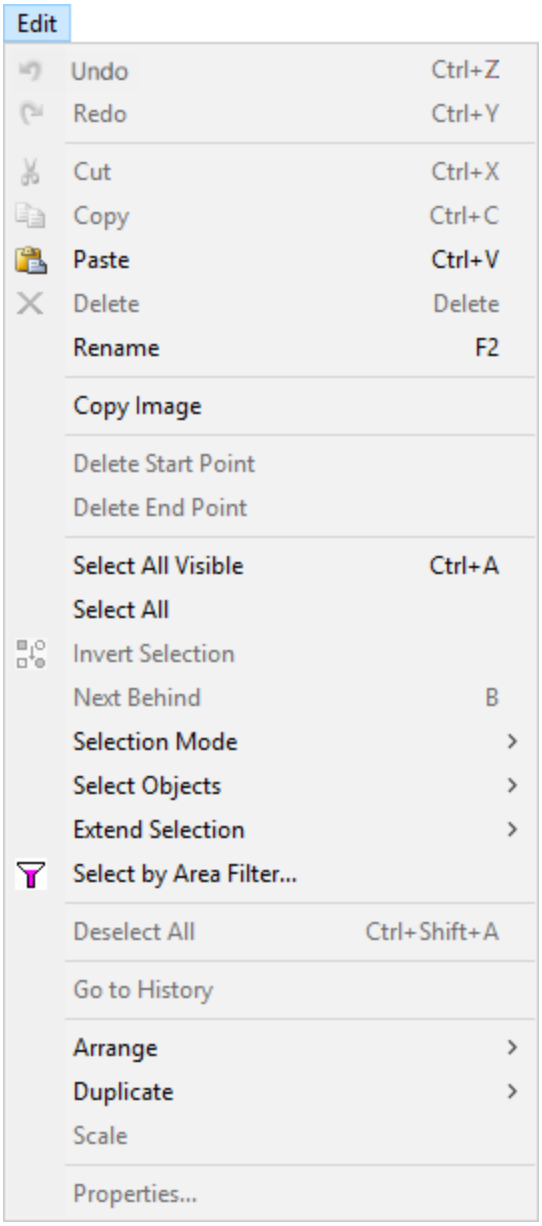
These menus include:

- Schematic Editor
- 3D Modeler
- Netlist Editor
- Report Window

Edit Menu for 3D Modeler

When a Modeler window is active in the Design Area, the **Edit** menu is similar to the following image:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

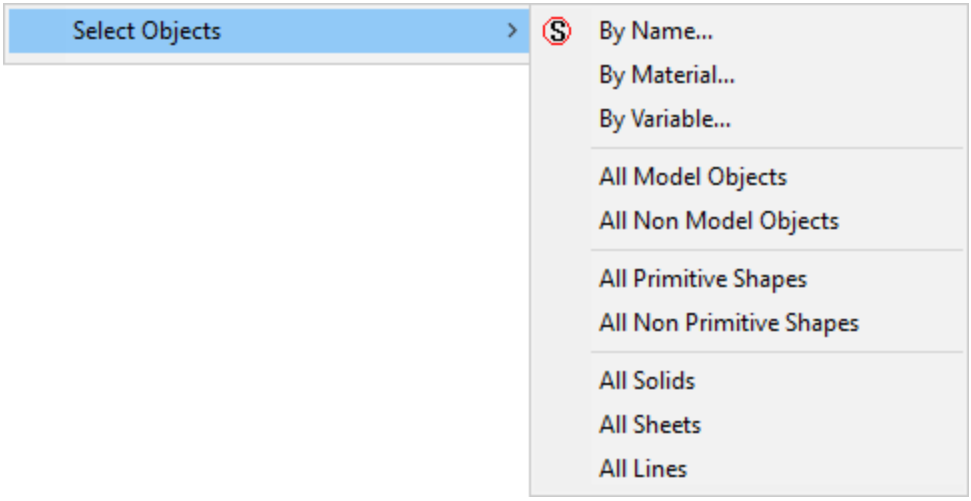


Comands will be available or not available for clicking depending on what is currently selected or on whether any operations have been performed, undone, or redone. The 3D Modeler's Edit menu contains the following options:

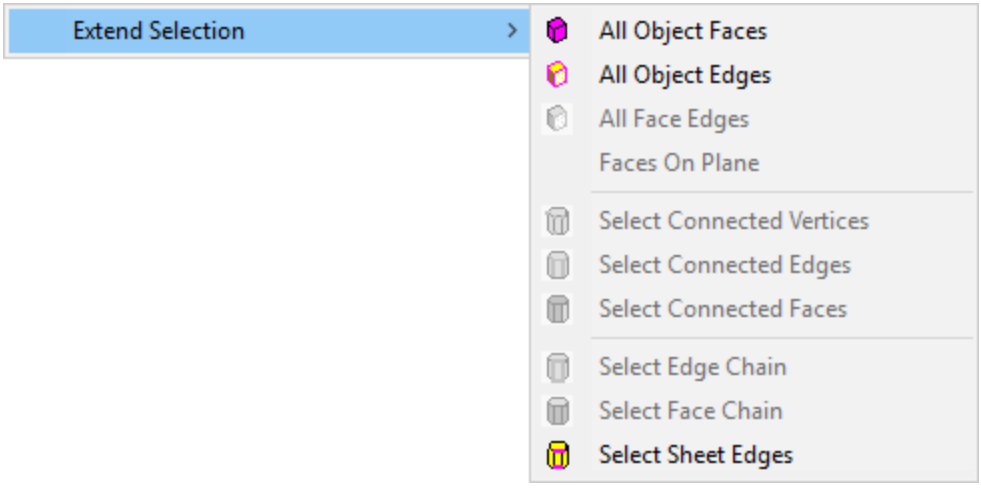
- **Undo** – allows you to undo the last action, which is specified after the word "Undo."
- **Redo** – allows you to redo an undone action, which is specified after the word "Redo."
- **Cut / Copy / Paste** – allows you to copy and paste elements.
- **Delete** – deletes a selected element.
- **Rename** – renames a selected element.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Copy Image** – copies the visible design area to the clipboard, in bitmap format.
- **Delete Start Point / Delete End Point** – allows you to delete start points and end points.
- **Select All Visible / Select All** – selects either all objects or all objects visible in the design area.
- **Invert Selection** – selects the opposite of the current selection.
- **Next Behind** – selects the object behind a selected face, edge, vertex, or object.
- **Selection Mode** – changes the selection mode.
- **Select Objects** – opens a submenu that allows you to select objects and shapes by name, material, and type.



- **Extend Selection** – opens a submenu that allows you to extend your current selection.



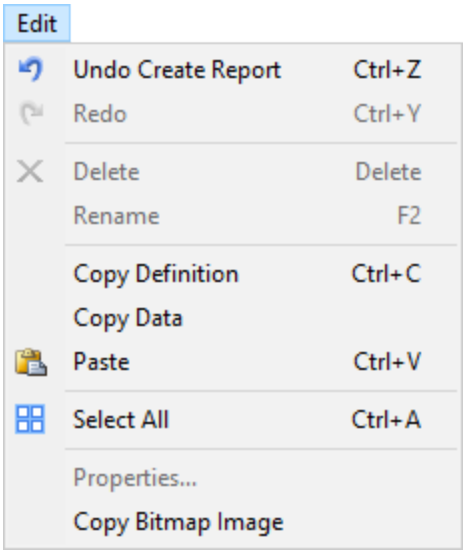
- **Select by Area Filter** – opens the **Select By Area Filters** window.
- **Deselect All** – deselects all selected objects.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Go To History** – selects the History Tree entry for the selected object.
- **Arrange** – allows you to move, rotate, or mirror the selected object.
- **Duplicate** – allows you to duplicate the selected object along a line, along an axis, or mirrored to the selection.
- **Scale** – allows you to scale the selected object.
- **Properties** – opens the relevant Properties window for the selected object.

Edit Menu for Report Window

When the Report Window is the active window, the **Edit** menu appears similar to the following:



The Report Window Edit menu contains the following options:

- **Undo** – allows you to undo the last action, which is specified after "Undo."
- **Redo** – allows you to redo an undone action, which is specified after "Redo."
- **Delete** – deletes a selected report element.
- **Rename** – renames a selected report element.
- **Copy Definition** – copies the report definition.
- **Copy Data** – copies report data.
- **Paste** – pastes the copied information at your selected location.
- **Select All** – selects everything in the report window.
- **Properties** – opens the relevant Properties window for the selected report element.
- **Copy Bitmap Image** – copies the entire report window as a bitmap image that can be pasted into another program or document.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

View Menu

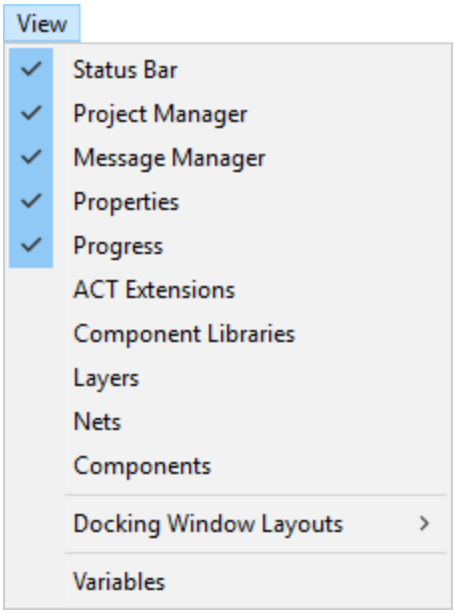
The **View** menu changes, depending on the active window in the Design Area.

These menus include:

- Basic View Menu
- Schematic Editor's View Menu

Basic View Menu

When no editor is open in the design area, or when the Netlist Editor or Report Window is active, the **View** menu appears similar to the following:

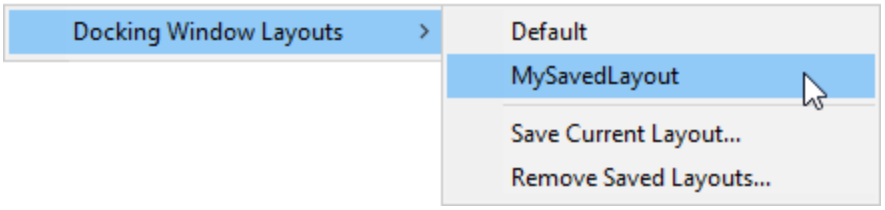


- The check boxes toggle the display of the **Status Bar**, **Message Manager Window**, **Project Manager**, **Properties Window**, and **Progress Window**. You can enable the display of **Component Libraries**, **Layers**, **Nets**, **Components** and **Variables** windows.
- The **ACT Extensions** command opens the *ACT Extensions* window, which lets you work with ACT integration tools. For more information, see the ACT Extensions Window section.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> The **ACT Extensions** window and the design wizards it contains (*5G Wizard, HFSS Antenna Design Toolkit, HFSS-EMA3D Link,* and *Maxwell Eccentricities*) are only available for the Windows version of the Ansys Electronics Desktop software. These items are not available when the software is installed on a Linux platform.

- The **Docking Window Layouts** submenu lets you select from the **Default** and any saved window layouts, **Save Current Layout**, or **Remove Saved Layouts** by selecting them from a list.

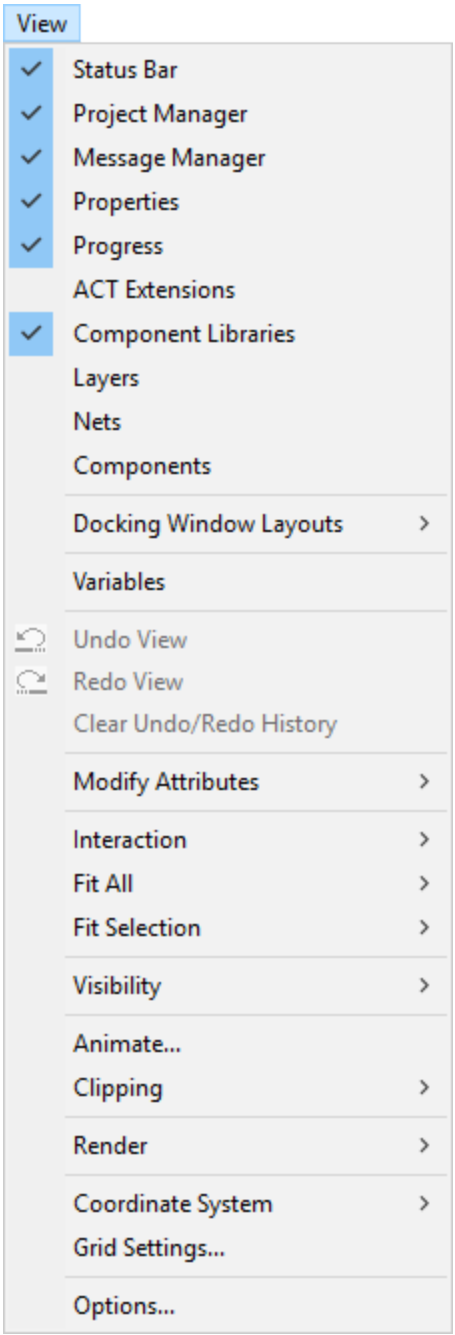


- The **Variables** option opens the **Project** and **Design** **Variables** window, allowing you to set variables.

3D Modeler View Menu

When a Modeler window is active in the design area, the **View** menu's appearance is similar to the following image:

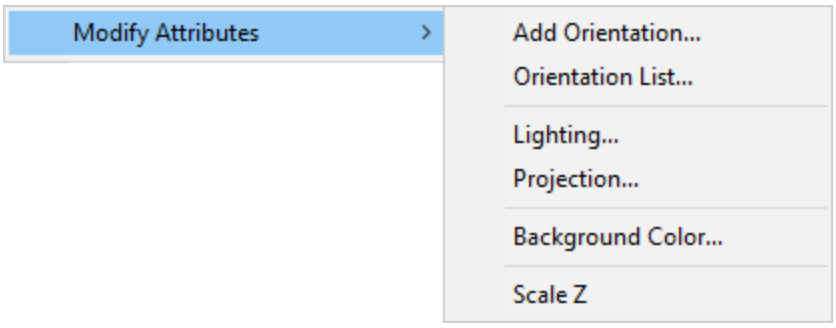
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



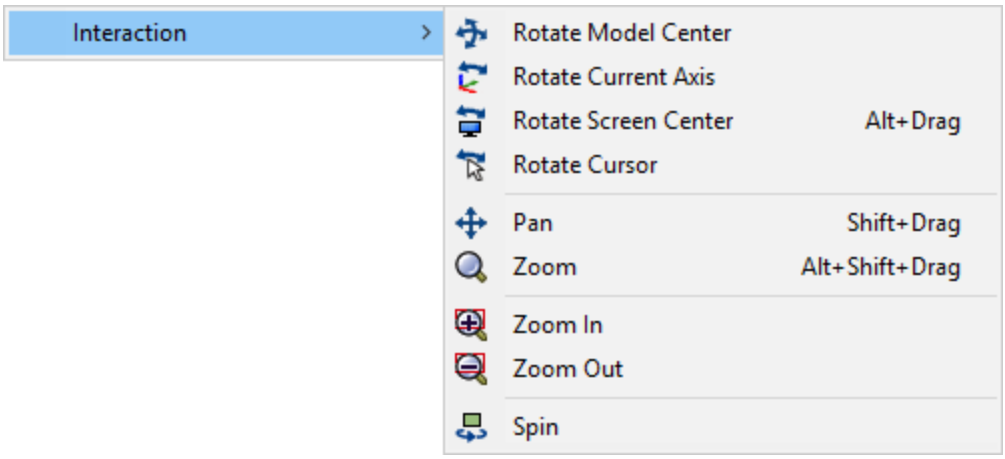
Thi View menu includes the options available on the Basic View Menu. The following additional options are available when a *Modeler* window is active. Note that a number of these options also appear in the View branch of the shortcut menu that appears when you right-click in the Modeler window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Undo View** / **Redo View** / **Clear Undo/Redo History** – allow you to change the view in the design area, based on view history.
- **Modify Attributes** – offers options to **Add Orientation**; view the **Orientation List**; change **Lighting**, **Projection**, or **Background Color**; and **Scale Z**.



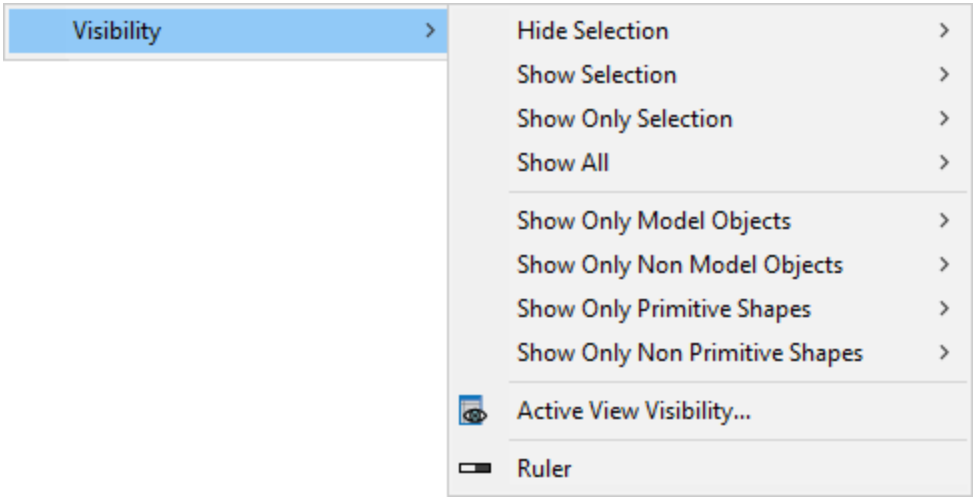
- **Interaction** – offers options to rotate, pan, zoom, and spin the drawing in the design area.



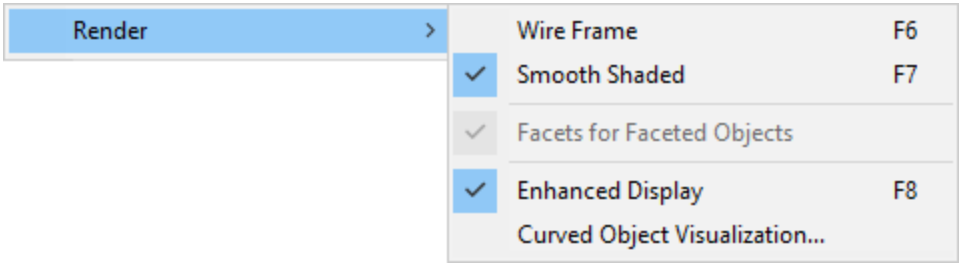
- **Fit All** – allows you to fit all objects to the view window.
- **Fit Selection** – allows you to fit selected objects to the view window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Visibility** – offers options to show and hide selections, objects, shapes, and the ruler.

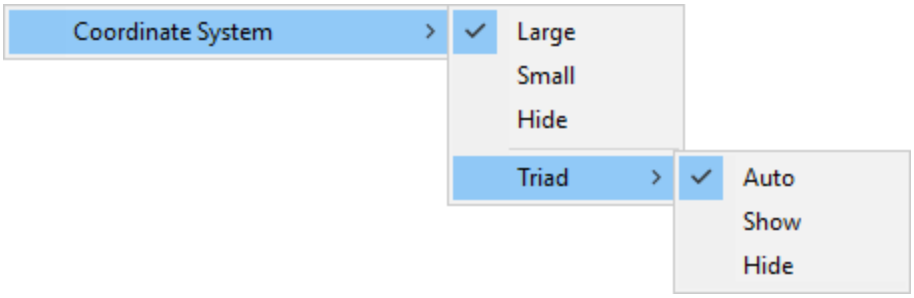


- **Animate** – opens the **Create Animation Setup** window, so that you can begin creating animations.
- **Clipping** – allows you to place a 3D coordinate system manipulator in the design area so that you can define clip planes.
- **Render** – offers options to switch between wire frame and smooth shaded render, toggle enhanced display, and change how curved objects are visualized.

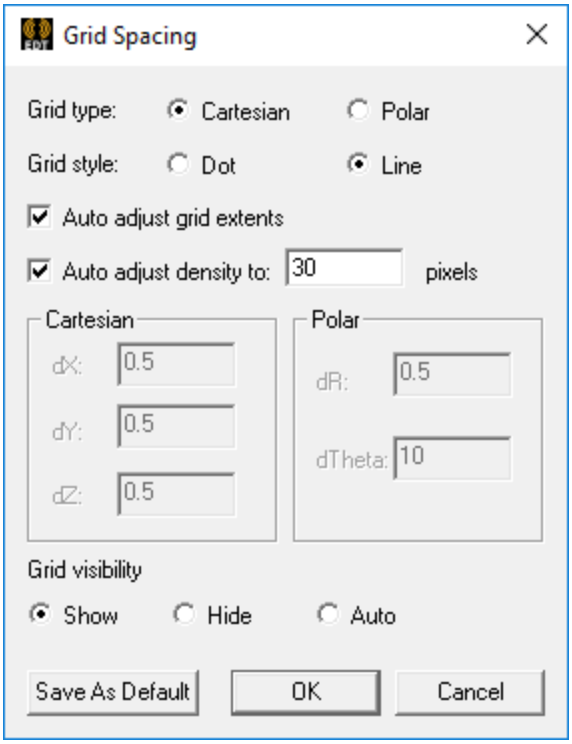


- **Coordinate System** – offers options to change how the coordinate system displays in the design area.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

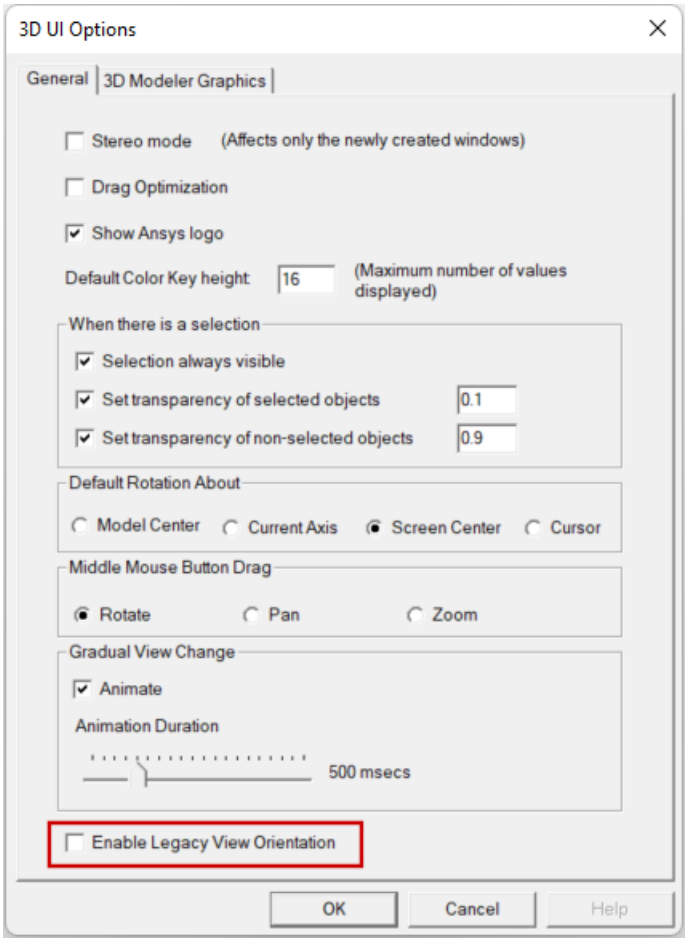


- **Grid Settings** – opens the **Grid Spacing** window, where you can adjust spacing.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

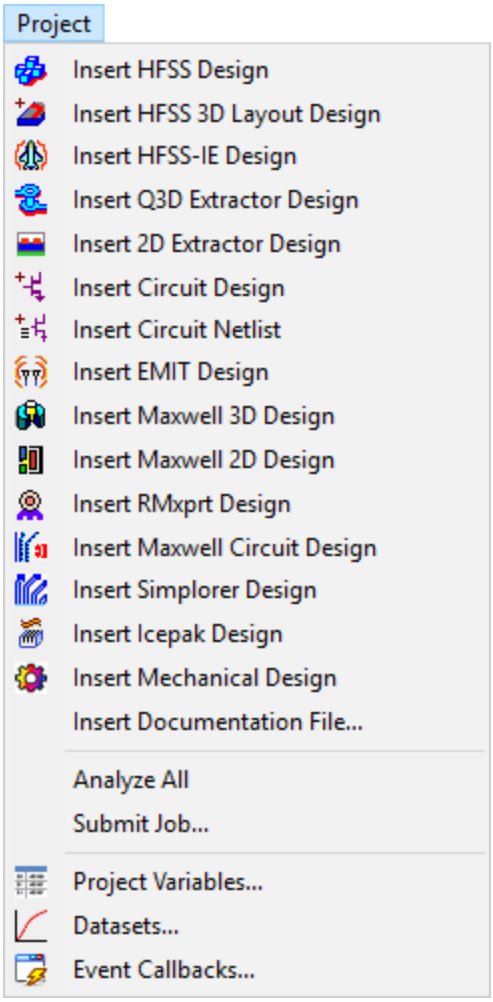
- **Options** – opens the **3D UI Options** window, where you can change additional settings.



Project Menu

To create (insert) a new design, you must first open a new project folder by selecting **New** from the **File** menu. Then select a project to insert from the Project menu. Or you can click the icon corresponding to the type of design you wish to create.

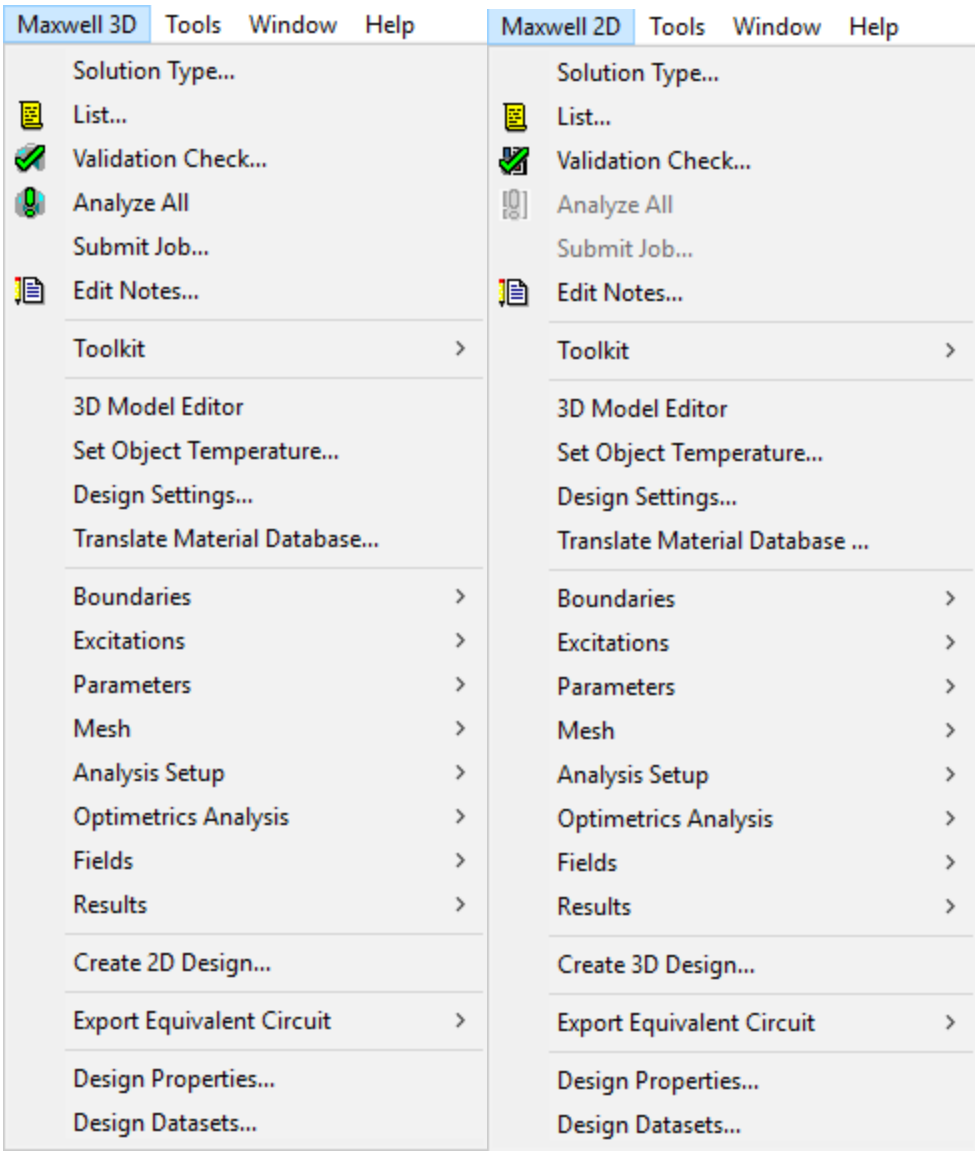
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



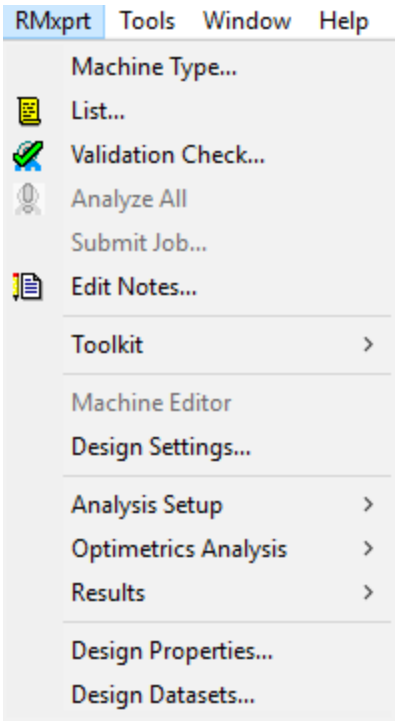
Editor and Design Specific Menus

The Top Menu bar contains editor and design menus that are specific to the editor or viewer that is active in the Design area. The specific menus correspond to the type of design that is inserted using the Project menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



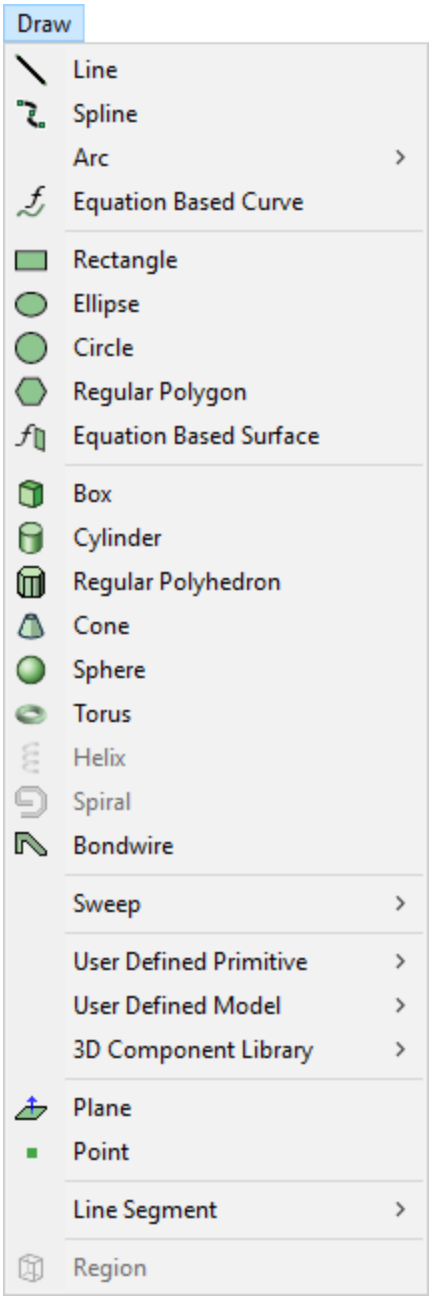
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Layout Editor Draw Menu

The **Draw** menu for the **Layout Editor** is context sensitive and appears slightly different depending on the type of design that is loaded.

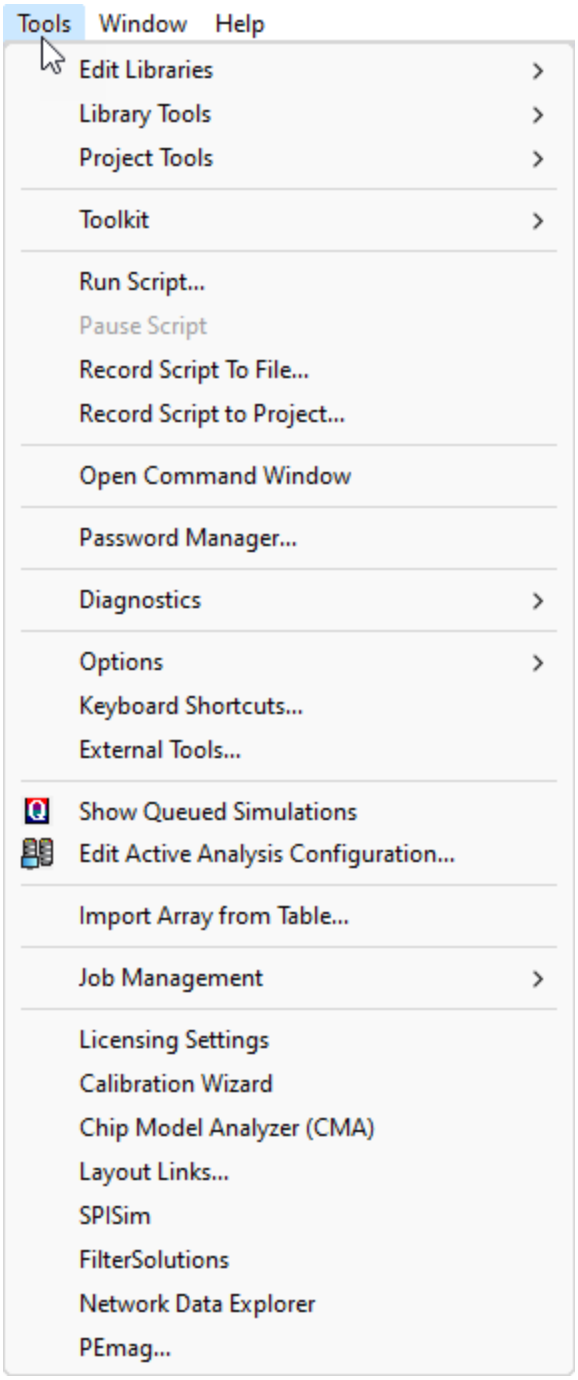
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Tools Menu

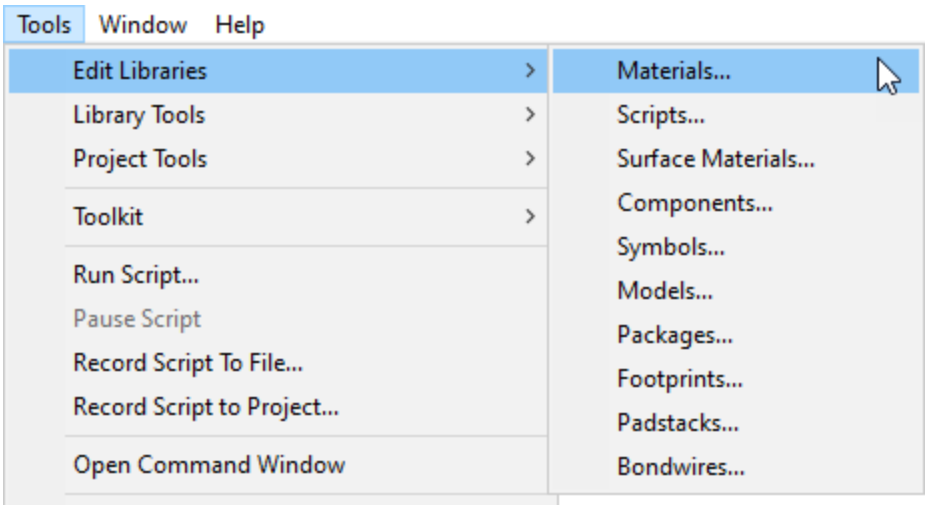
The **Tools** menu contains operations that are common to all design types:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

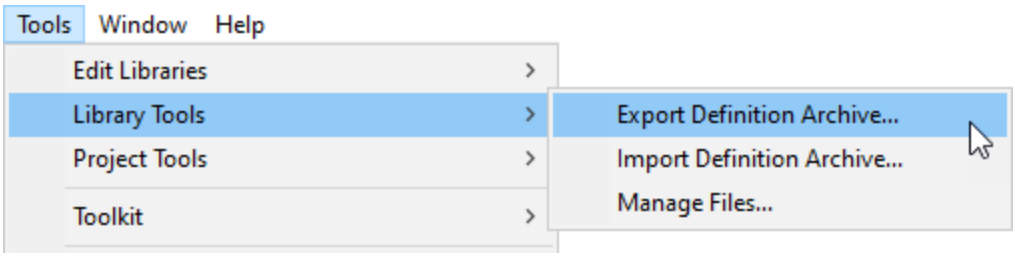


- Click **Tools** > **Edit Libraries** to view commands allowing access to the libraries available for the various Ansys Electronic Desktop solves, such as Materials, Scripts, and Components, Symbols and Models for Simplorer, Icepak, and Circuit.

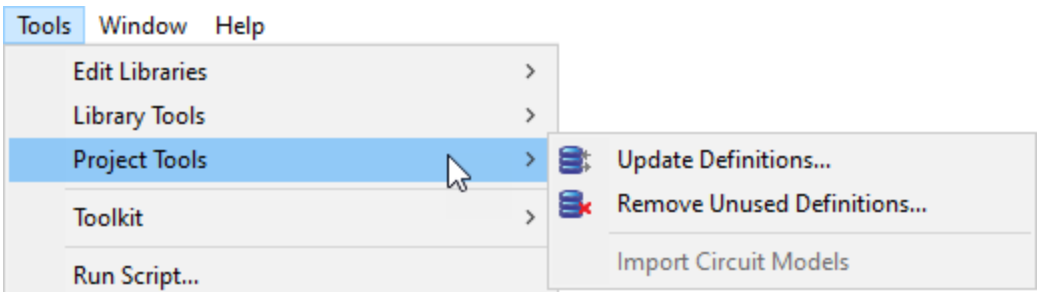
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Click **Tools** > **Library Tools** to view tools for importing or exporting definition archives for UserLib and Syslib and to manage files relevant for various solver libraries.



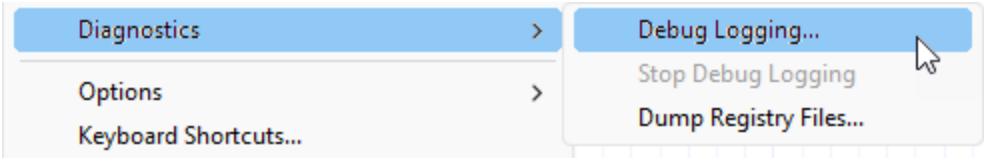
- Click **Tools** > **Project Tools** to manage definition libraries relevant to various solvers.



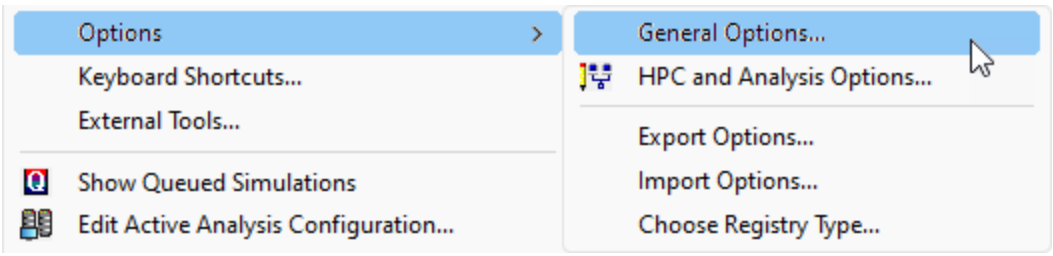
- Click **Tools** > **Toolkit** to install PyAEDT or update product menus to show any added Python-based toolkits.
- Click **Tools** > **Runscript...** to run scripts that you have recorded. A Pause command is available. You can also **Record Script to File...** or **Record Script to Project...**. See the Scripting Help for more information.
- Click **Tools** > **Open Command Window...** to open an IronPython command window. See the Scripting Help for more information.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click **Tools** > **Password Manager...** to open the *Password Manager* window to control access to resources.
- Click **Tools** > **Diagnostics** to access the following diagnostics commands:



  - Click **Debug Logging** to enable debug logging via an environment variable.
  - Click **Stop Debug Logging** to end debug logging. This option is only available while debug logging is active.
  - Click **Dump Registry Files** to make a copy of all user, administrator, and default-settings configuration files, placing them in the folder you select.
- Click **Tools** > **Options** to set General Options and HPC options.



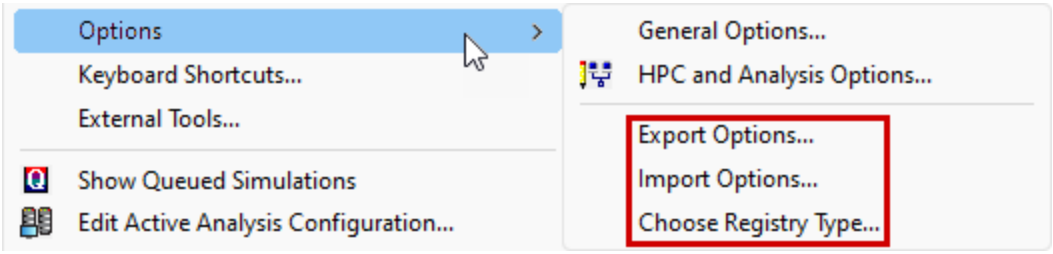
  Other available commands in this fly-out menu are **Export Options**, **Import Options**, and **Choose Registry Type**. See Understanding Registry Tools for more information on these three commands.

- Click **Tools** > **Keboard Shortcuts...** to manage your keyboard shortcut behavior.
- Click **Tools** > **External Tools...** to manage menu access to external tools.
- Click **Tools** > **Show Queued Simulations** to view a dialog listing simulations in a queue and providing tools to manage the queue.
- Click **Tools** > **Edit Active Analysis Configuration** to view a dialog that lets you view and manage the current configuration.
- Click **Tools** > **Import Array from Table** to import an array definition from a .csv file.
- Click **Tools** > **Job Management...** to select a scheduler, and to submit and monitor remote simulations.
- Click **Tools** > **License Settings** to launch the Licensing Settings tool.
- Additional Tools listed include Calibration Wizard, and Network Analyzer, and other tools used to help you with Ansys Electronics Desktop projects and solvers.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

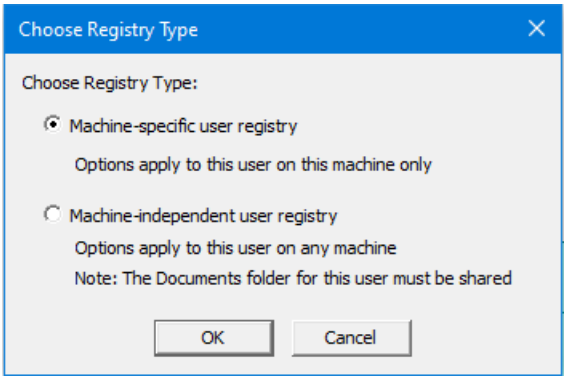
Understanding Registry Tools

In the context of this topic, when we say *"Registry"* we are referring to the files where the various Ansys Electronics Desktop application options are stored. We are not referring to the Windows registry. Specifically, this topic discusses the **Export Options**, **Import Options**, and **Choose Registry Type** commands that are listed in the fly-out menu that appears when you click **Tools** > **Options** from the menu bar.



Choose Registry Type:

Click **Choose Registry Type** to switch between two available options. The following dialog box appears:



The choices are summarized as follows:

- **Machine-specific user registry:** This is the default type. You must set up the configuration manually for each user account on each machine where the Ansys Electronics Desktop software is run. You can do so by exporting and importing options (see below), setting the options in the user interface on each machine, or copying configuration files between machines and renaming them. The hostname of the computer is part of the filename for machine-specific registries.
- **Machine-independent user registry:** Allows you to share a single user configuration file among multiple machines. This choice has the advantage of keeping the options synchronized between all the user's machines.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> The term "host" is frequently used to refer to any computer used to run a particular application. In the Ansys Electronics Desktop Help, the terms *"host-specific"* and *"machine-specific"* are used interchangeably, as are *"host-independent"* and *"machine-independent."*

When you switch registry types, the settings from the currently active registry are copied into the new registry, which then becomes the active one. The filenames for user registries are as follows:

- Machine-specific: **<*hostname*>_user.xml**
- Machine-independent: **user.xml**

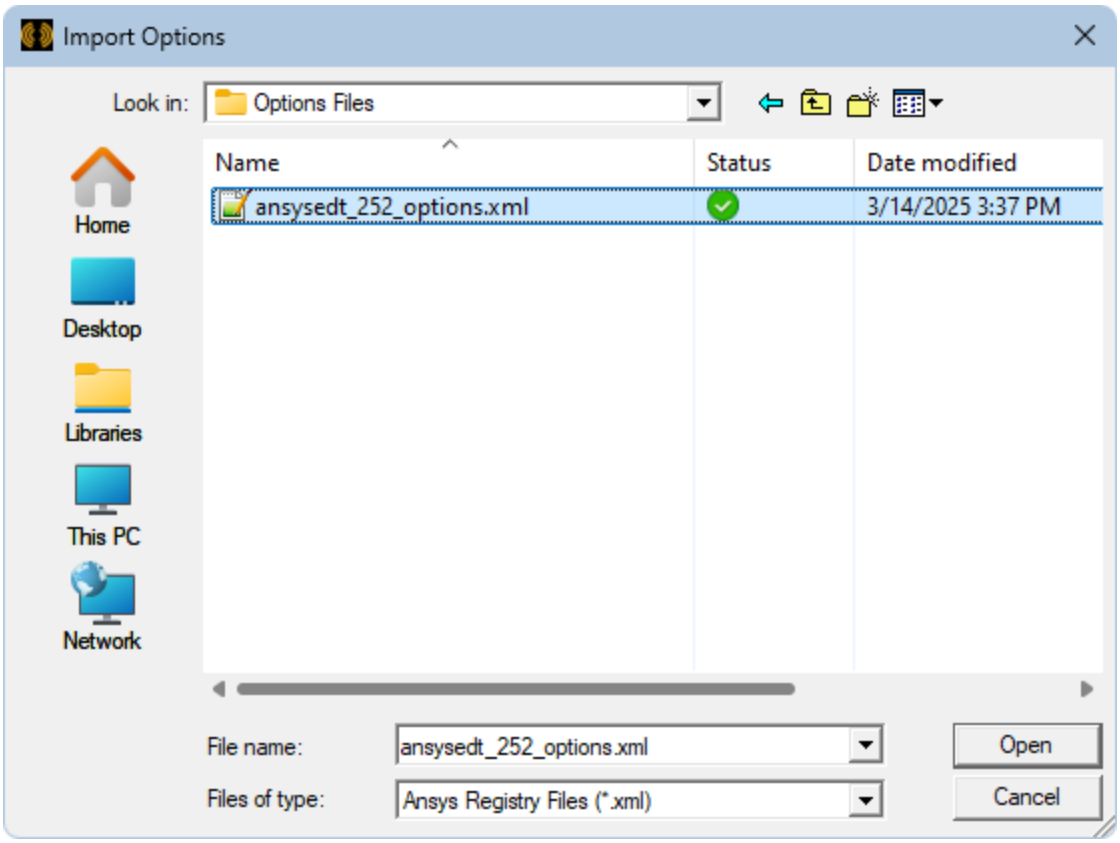
> **Note:**
>
> For a machine-independent user registry to work, the *Documents* folder must be shared by all machines on which you run the software, and you must log into the same account on each machine. The user registry file is stored in the path, ...Documents\Ansoft\ElectronicsDesktop20*yy.m*, where *yy* is the last two digits of the version year, and *m* is the minor release number. The Documents folder must be mapped to a network folder accessible to all machines or to an online folder (such as OneDrive, for example). Otherwise, you would have to copy the registry file to each machine, and they would *not* remain synchronized, defeating the purpose of the machine-independent registry.

Exporting and Importing Options:

You can export options to an XML file, which can then be imported into the registry. This capability is useful if you want to use one set of options for a given situation and a different set for another. Simply export the options for each different configuration you use. Then, import the file that is applicable to your current work. You can also use this capability to transfer options to a different user or machine.

When you click **Import Options**, the following dialog box appears, in which you can navigate to the option file folder and select the desired file to import:

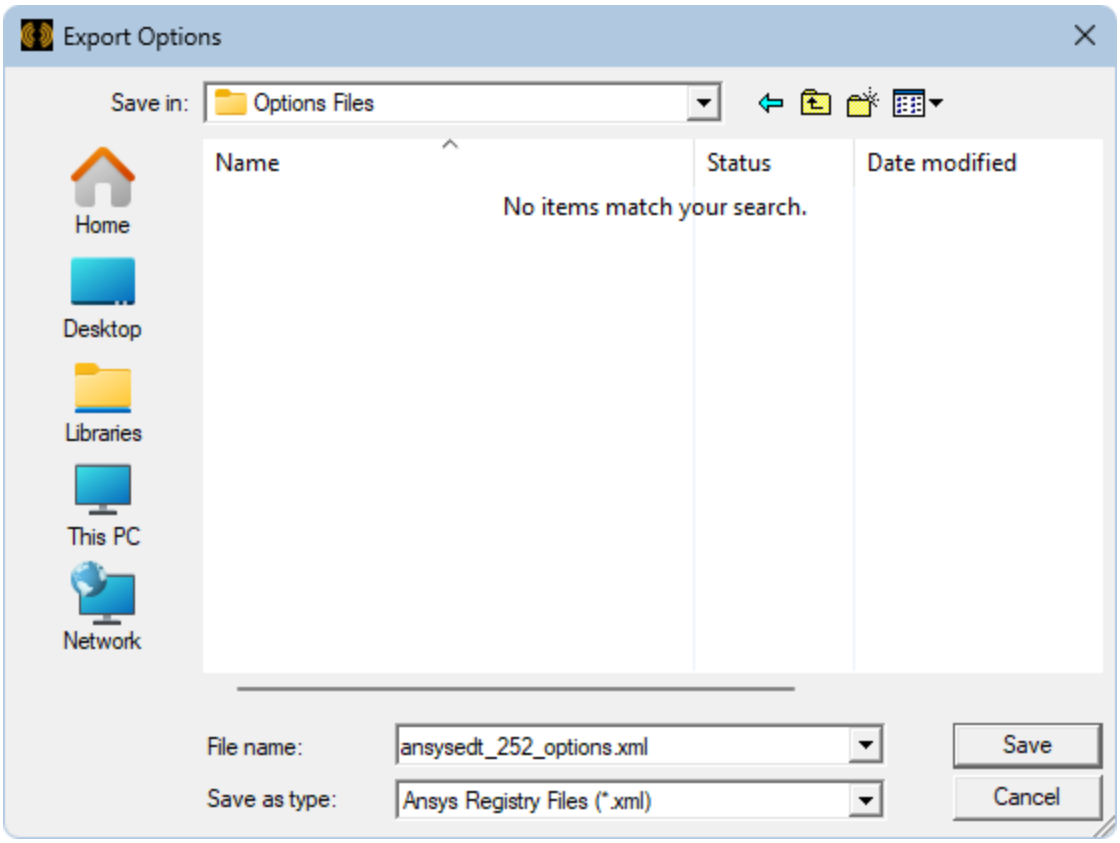
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When you import an options file, the currently active registry type (specified using the **Choose Registry Type** command) determines to which registry file the options are written. After importing an options file, you will be prompted to restart the application, which is recommended.

When you click **Export Options**, the following dialog box appears, in which you navigate to the folder where you want to place the file and specify the filename:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Options are always exported from the currently active registry. The usual precedence rules apply when getting the value to export. For example, if the current registry being exported is machine-specific, and a particular value is missing, the value in the machine-independent registry will be used (if available). See the **Note** at the bottom of Setting Options via Configuration Files for more information on the order of precedence.

> **Note:**
>
> An exported user options file contains only the settings that are exposed in the general options (that is, the *Options* dialog box) and view options (*3D UI Options* dialog box). So the exported file contains a subset of the configuration settings. HPC and other settings (such as window locations and window size data) are *not* included in the exported options.

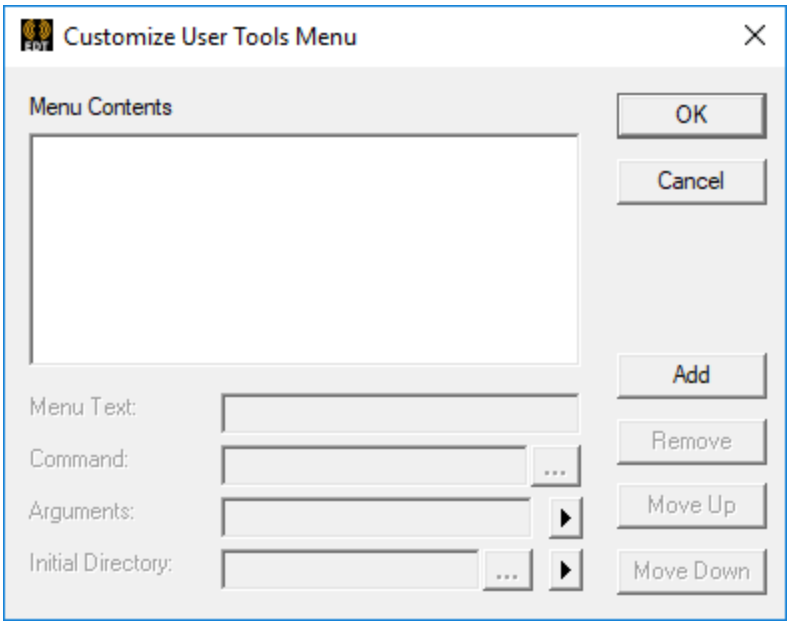
Adding External Tools to the Tools Menu

To add an executable to the Tools menu:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Tools** > **External Tools**.
2. This displays the **Customize User Tools Menu** dialog box.

   If a **User Tools** menu item has been defined, its contents are displayed. Command buttons let you Add new commands and Delete selected commands, and Move Up and Move Down commands. You can specify the command line arguments to the program and the directory from which it will run.



3. To add a custom Tools menu entry, click **Add**.

   This enables the following fields:

   Menu Text field – displays [new tool] as text you will replace with the text you want to appear in the User Tools menu.

   Command field – displays the external executable. An ellipsis button [...] lets you navigate to the file location.

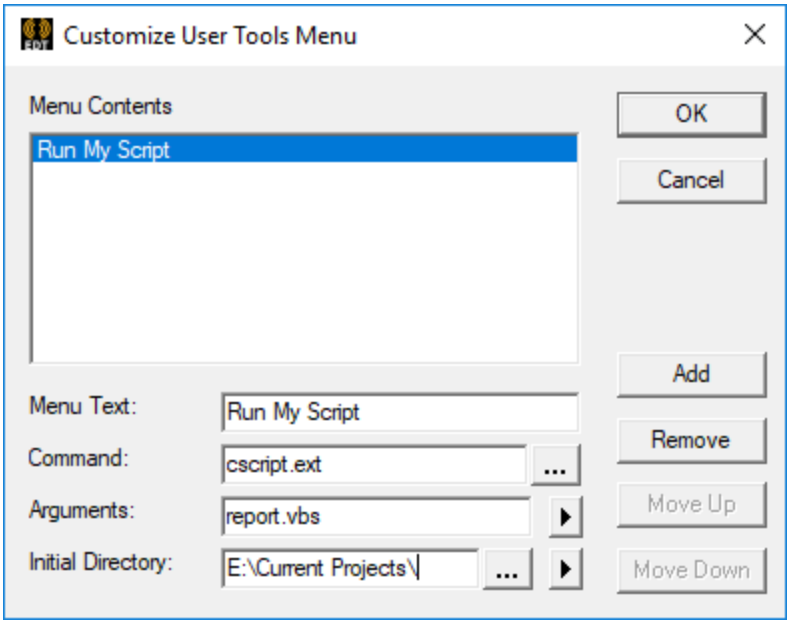
   Arguments field – accepts command arguments from the **>** button menu selections for File Path, File Directory, File Name, File Extension, Project Directory, or Temp Directory.

   Initial Directory – specifies the initial directory for the command to operate. The ellipsis button {...] displays a dialog that lets you navigate folders in your desktop, or across the network.

4. Click **OK** to add the External Tools menu to Maxwell or Cancel to close the dialog without changes.

   You can also add scripts to the **Tools** menu. Assuming you have a script to generate custom reports called report.vbs, use the cscript.exe program to execute your script.

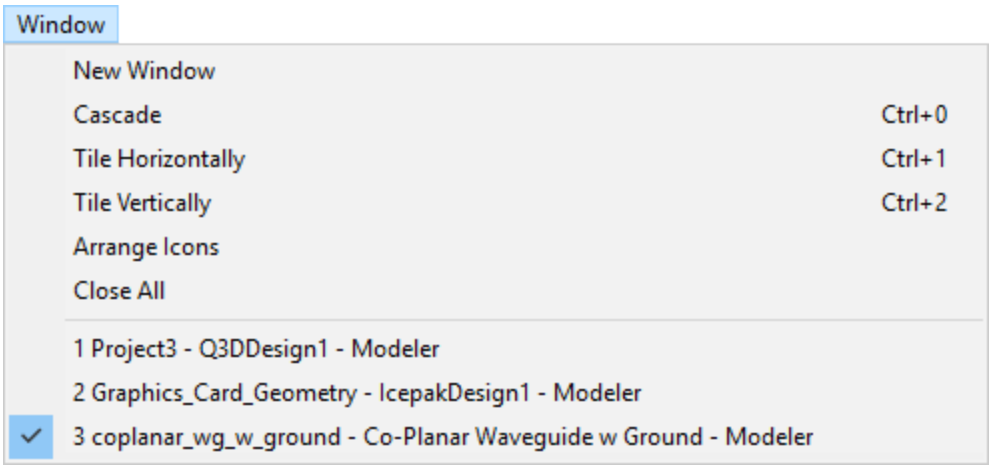
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This example shows the cscript.exe program added to the **Tools** menu as **Run My Script**. The command line argument to the csscript.exe program is report.vbs. You can also name the directory in which it will be run.

Window Menu

The **Window** menu contains common window control operations, as well as a list of projects that are currently open.



- Click **New Window** (or type **n**) to open a new window in the Design Area. The new window will show the active design.
- Click **Cascade** (or type **c**) to arrange the open design windows in overlapping sequence:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can use the **Cascade** operation to restore all windows to their default sizes after one or more of them have been maximized (enlarged to fill the entire Design Area).

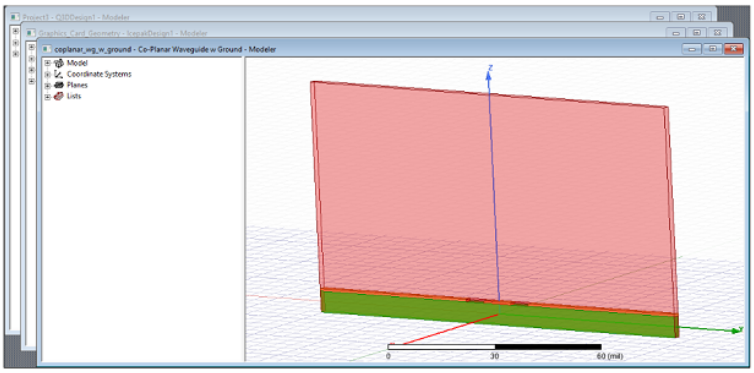
- Click **Tile Horizontally** (or type **h**) to arrange the open design windows in a top-to-bottom sequence:
- Click **Tile Vertically** (or type **v**) to arrange the open design windows in a side-to-side sequence:

> **Tip:**
>
> You can use the shortcut key sequences Ctrl+0, Ctrl+1, and Ctrl+2 to execute the Cascade, Tile Horizontal, and Tile Vertical operations, respectively. The shortcuts can be used at any time, bypassing the Window menu.

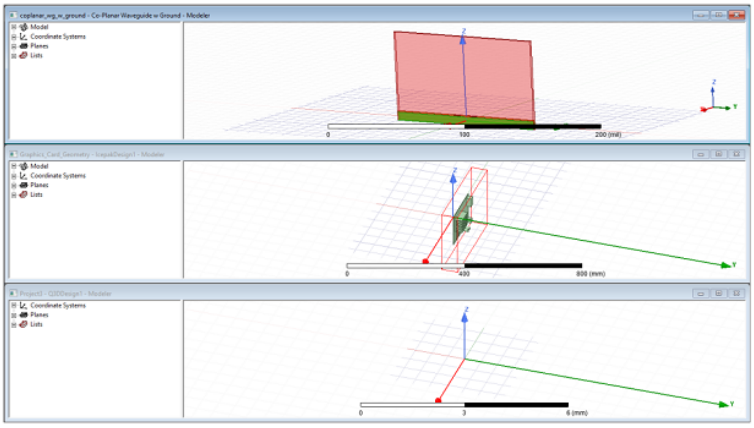
- Click **Arrange Icons** (or type **a**) to restore iconized windows to a neat row at the bottom of the Design Area, after the icons have been repositioned manually.
- Click **Close All** (or type **l**) to close all the editor windows in the Design Area.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

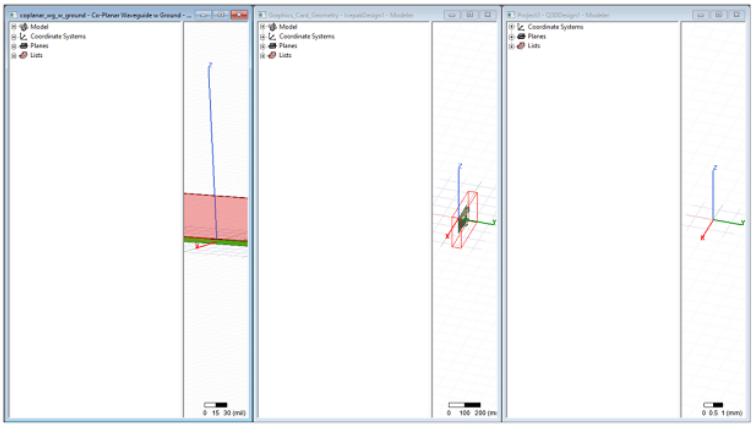
Cascade



Tile Horizontally



Tile Vertically



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Help Menu

Ansys Electronics Desktop features extensive help documentation, including text search and context-sensitive help items. To access the main help system, click **Help** on the top menu bar.

The Help Menu displays different selections depending upon the type of design inserted to the active project. The basic selections for Maxwell are:

> **Note:** Ansys Electronics Desktop Student includes access to PDF documentation only.

- **Maxwell Help** – opens the Maxwell help within the Electronics help system. You can also access PDF versions from within the help system.
- **Maxwell Scripting Help** – opens the Maxwell scripting help.
- **Maxwell Getting Started Guides** – opens a list of links to the Maxwell Getting Started Guides, which walk you through projects that demonstrate product features.
- **Maxwell PDFs** – provides access to PDFs for Maxwell, including the main help, scripting guide, and Getting Started Guides.
- **Components** – enabled when you insert a circuit design to the active project; it opens the Circuit Components help.
- **Ansys Customer Support** – opens a browser page to the Ansys Customer Portal. At the website you can learn more about Ansys products and services and log on to contact Ansys technical support staff.
- **What's New in this Release** – opens a PDF that describes *What's New in Ansys Electronics Desktop* for 2025 R2.
- **Ansys Product Improvement Program** – opens a window describing the Product Improvement Program option.
- **License Settings** – opens help for the License Settings tool.
- **Ansys Engineering Copilot** – opens a feature for learning and support within select Ansys products. This feature displays in an independent window within supported applications. It retrieves content from various sources, including the Ansys Innovation Space (AIS), Ansys documentation and learning resources, and the AnsysGPT™ virtual AI assistant.
    - **Home** – includes a search function that will search the courses, knowledge base, and learning forum for information relevant to the search term you have entered (searches the entire Ansys Innovation Space (AIS)). The Search AIS field also provides filters to narrow your search in a particular information category; however, this feature does not automatically filter based on physics type or the product you are using.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Ansys Innovation Courses** – the Search function can be used to access a wide range of Ansys Electronics Engineering courses that include on-demand, self-paced video training and quizzes.
- **Ansys Knowledge** – provides access to Ansys expert knowledge articles. The search filter is preset based on the application you are using, but you can change the filter as desired.
- **Ansys Learning Forum** – provides access to discussions and presentations from Ansys experts, partners and customers. Users can also post questions to the forum, which is moderated by Ansys employees.
- **AnsysGPT** – is a multilingual Ansys Artificial Intelligence (AI) Virtual Assistant. The AnsysGPT™ assistant takes your query and produces answers based on internal and external expert knowledge of the product. It draws upon information from Ansys Knowledge articles, Ansys Innovation Courses, Ansys Learning Hub courses, the Ansys Learning Forum, Ansys documentation, Ansys.com, and the official Ansys YouTube channels, as well as other Ansys information sources.

  To use the Copilot tool:

  - You must have a current AnsysGPT™ AI+ license. Ansys Engineering Copilot is a licensed product at no additional cost.
  - You must be registered on Ansys Customer Support Space (ACSS).

  For more information on Ansys Engineering Copilot, see the Ansys Engineering Copilot User Guide.

- **Ansys Learning Hub** – opens a web page with subscription-based access to virtual and self-paced learning across the Ansys Software portfolio.
- **About Ansys Electronics Desktop** – opens a dialog box that displays the Ansys Electromagnetics Suite release number and contains tabs that show information about the **Installed Components** and **Client License Settings**.

Context-Sensitive Help

To access context-sensitive help from the Ansys Electronics Desktop user interface, press **F1** while your cursor is on an item. The help system specific to the product opens.

Obtaining Information about the Software and Release

To obtain information about the software and release:

1. Click **Help** > **About Ansys Electronics Desktop**.
   The **About Ansys® Electromagnetics Suite [release number]** dialog box appears, listing information about the product.
2. Click the **Installed Components** tab to view a list of software installed.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click the **Client License Settings** tab to view information about the following:
   - Provider Name
   - Ansys License Version
   - FlexNet Publisher Servers
   - Admin Directory
   - Customer Number
   - FLEXlm Version
   - Redirect Info
4. To export the software information:
   a. Click **Export**.
   The **Save As** dialog box appears.
   b. Browse to the location where you want to save the information as a text file.
   c. Type a name for the file in the **File name** text box. The **Save as type** drop-down menu is already specified as **Export (*.txt)**.
   d. Click **Save**.
5. Click **OK** to close the **About Ansys® Electromagnetics Suite [release number]** dialog box.

Shortcut Menus

A variety of shortcut menus — menus that appear when you right-click a selection or in a window — are available in the **3D Modeler** window, in the **Project Manager** window, in the **History Tree**, and in the **Progress** window.

Shortcut menu in the 3D Modeler window	Use the shortcut menu in the **3D Modeler** window to select, magnify, and move objects (zoom, rotate, etc.); change the view; perform boolean operations; assign materials, boundaries, excitations, or mesh operations to objects; and work with field overlays.
Shortcut menus in the Project Manager window	Use the shortcut menus in the **Project Manager** window to manage Ansys Electronics Desktop project and design files and design properties; assign and edit boundaries, excitations, and mesh operations; add, analyze, and manage solution setups; add Optimetrics analyses; create post-processing reports; insert far- and near-field radiation setups; edit project definitions; and run Maxwell SPICE.
Shortcut menus in the History	Use the shortcut menus in the **History** tree to expand or collapse groupings. If you select particular objects in the history tree, the shortcut menu lists the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tree	commands that you can apply to the selected object (s).
Shortcut menus in the Progress window	Use the shortcut menus in the **Progress** window during a simulation to **Abort** or **Clean Stop**.

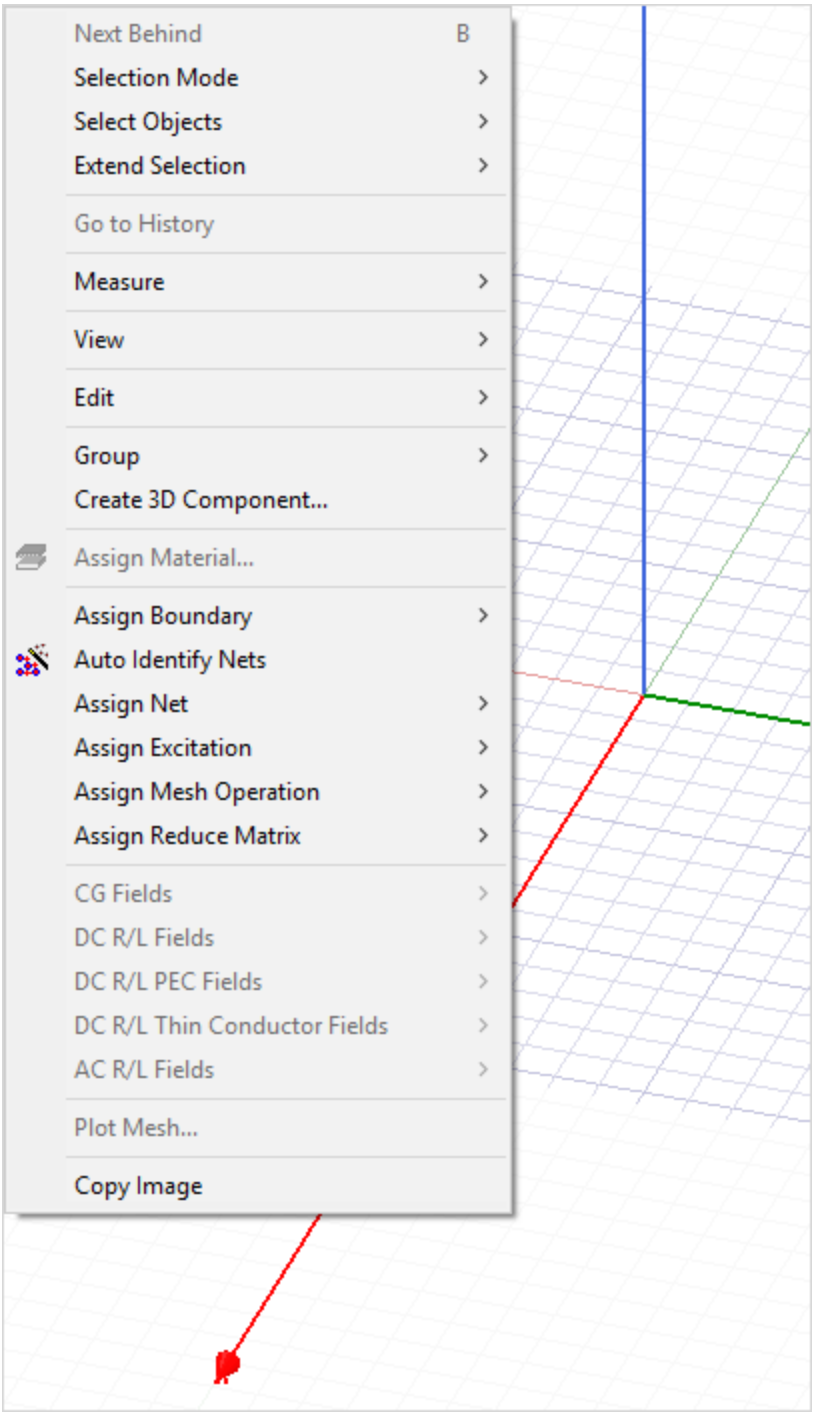
> **Note:**
>
> All commands available on shortcut menus are also available from the menu bar.

Shortcut Menu in the 3D Modeler Window

Use the shortcut menu in the **3D Modeler** window to select, magnify, and move objects (zoom, rotate, etc.); change the view; perform boolean operations; assign materials, boundaries, excitations, or mesh operations to objects; and work with field overlays.

To access the shortcut menu, right-click in the 3D Modeler window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

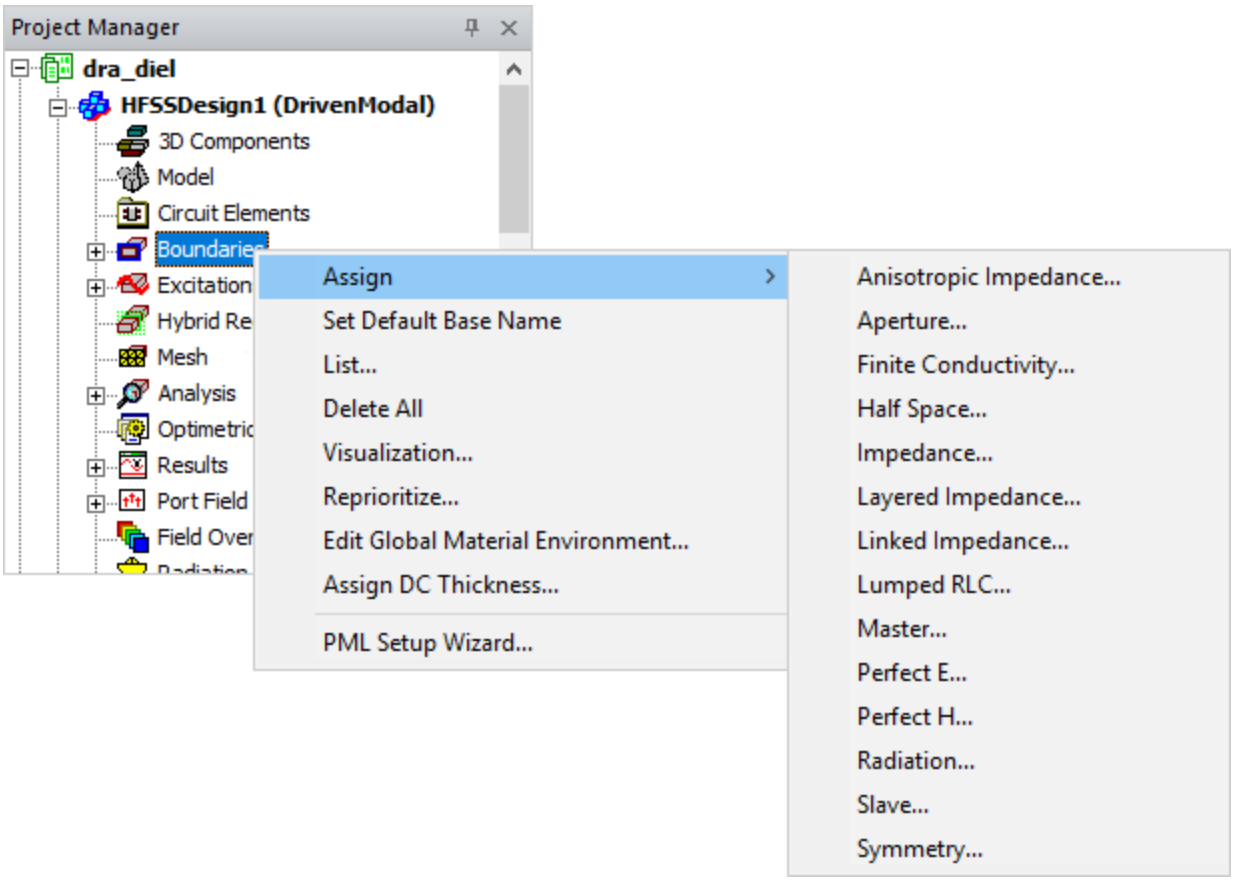


Shortcut Menus in the Project Manager Window

Each node, or item, in the project tree has a shortcut menu. For example, from the shortcut menu for the **Boundaries** icon, you can assign boundaries to selected objects; review

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

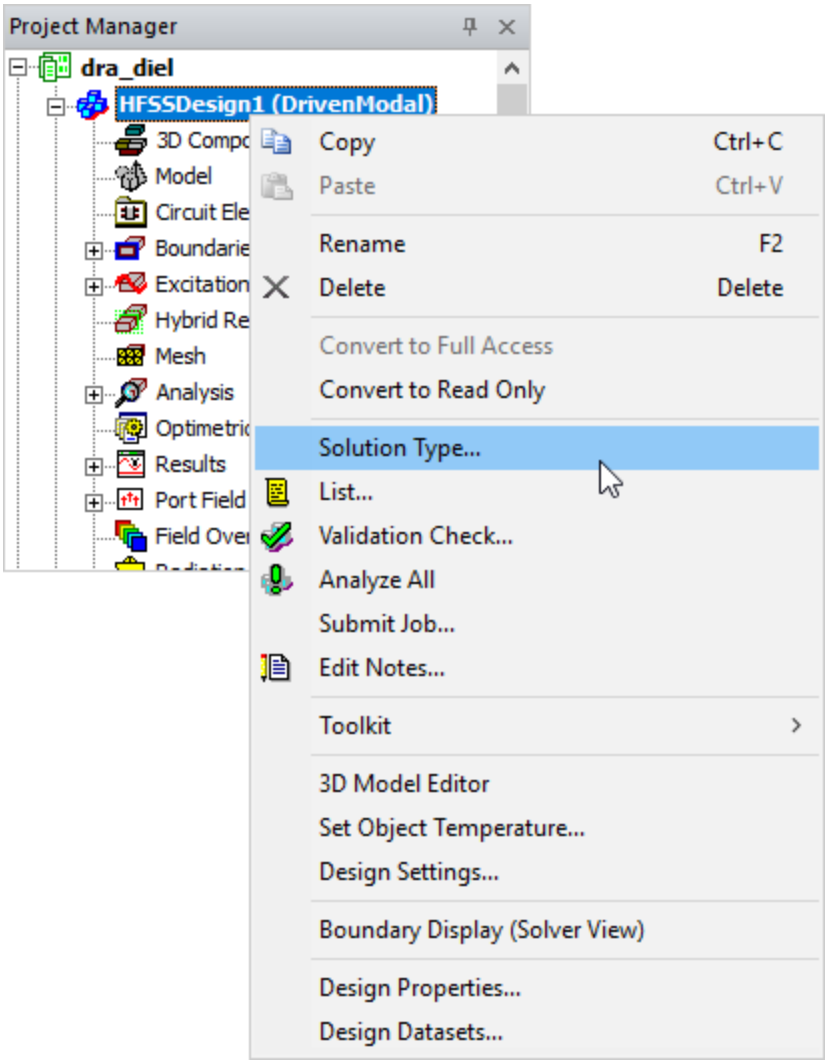
information for all the boundary assignments for the active design; remove all boundary assignments; show or hide a boundary's geometry, name, or vectors; change the priority of a previously assigned boundary; and use the PML Setup wizard to create a perfectly matched layer (PML) boundary.



Other nodes have shortcut menus appropriate for the context.

For example, the following figure shows the shortcut menu when you right-click the design.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Customizing Ansys Electronics Desktop Menus

Ansys Electronics Desktop comes with a Menu Bar. You can modify the menu bar by using customized User Interface (UI) setups to add, remove, rename, and relocate commands. You can also add commands that execute external scripts or configure right-click shortcut menus for folders and items in the Project Tree, which allows for extensive customization of the Ansys Electronics Desktop Menus.

Customized setups are implemented using named subfolders that contain XML files used to configure customized menu settings. The subfolders included with the Ansys Electronics Desktop application are:

- config/UI/ElectronicsDesktop/EM
- config/UI/ElectronicsDesktop/RF

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- config/UI/ElectronicsDesktop/RF.0
- config/UI/ElectronicsDesktop/SI
- config/UI/ElectronicsDesktop/SI1.0
- config/UI/ElectronicsDesktop/SI2.0
- config/UI/ElectronicsDesktop/Twin Builder

Each customized UI type has its own subfolder. Within these subfolders, XML files hold the menu configurations as well as the right-click menus for that UI type.

Select Different UI Types

To change the menu display from the default UI to a different UI type:

- Click **Tools** > **Options** > **General Options** > **General** > **Desktop Configuration**.
- Choose the UI type that you wish to use from the **Custom Menu Set** drop-down menu and click **OK**.

To switch from a customized UI type to the default UI:

- Click **Tools** > **Revert To Default UI**.

Add a New Customized UI Type

A new folder needs to be added to the config/UI/ElectronicsDesktop folder for any new customized UI type. All XML files for this UI type must go in this folder. The new UI type will appear in the **Custom Menu Set** drop-down menu (**Tools** > **Options** > **General Options** > **General** > **Desktop Configuration**).

For any UI type, Ansys Electronics Desktop displays the default UI menus for any products/contexts that are not in the xml files. If there is error processing any XML file, the default UI menu is displayed for that product. Check the message window for the names of problematic XML files and suggestions on how to fix them.

XML files are only processed once when you first switch to that UI type. After you make changes to any XML file, in order for it to be reprocessed, navigate to **Tools** > **Options** > **General Options** > **General** > **Desktop Configuration** and reselect the UI type from the **Custom Menu Set** drop-down menu. Then click **OK**.

Names of the XML Files

Below is the list of XML files that can be placed in a folder for a new UI Type. If any of these XML files does not exist in this folder, the default menu setting displays for that product.

- 2D Extractor.xml – used for 2D Extractor projects.
- Circuit Design.xml – used for Circuit Design projects.
- Circuit Netlist.xml – used for Circuit Netlist projects.
- EMIT.xml – used for EMIT projects.
- HFSS 3D Layout Design.xml – used for HFSS 3D Layout Design projects.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- HFSS.xml – used for HFSS projects.
- HFSS-IE.xml – used for HFSS-IE projects.
- Icepak.xml – used for Icepak projects.
- Maxwell 2D.xml – used for Maxwell 2D projects.
- Maxwell 3D.xml – used for Maxwell 3D projects.
- Maxwell Circuit.xml – used for Maxwell Circuit projects.
- NoDesignUI.xml – used for menu settings when no project is selected.
- Q3D Extractor.xml – used for Q3D projects.
- RightClickMenu.xml – used for all right-click menu settings.
- Twin Builder.xml – used for Twin Builder projects.

Valid XML Elements and Attributes

1. Root Element is **DesignerMenu**:
    - **xmlns** – required attribute which needs to be set to the following:

      <DesignerMenu xmlns="http://www.ansys.com/uiConfigMenu">

    - **UseProjectWindowSelectionContext** – child of DesignerMenu, appears zero or one time.

      > **Note:**
      >
      > This is only meaningful if it is used in NoDesignUI.xml. If this is set to true, then clicking in the Project Window on the Project icon or the Definitions icon (or a subitem) will show “Project” context in NoDesignUI.xml instead of active design context. No setting or setting this to false makes this UI type behaves the same as the default behavior of Ansys Electronics Desktop. The menus and toolbars are always shown for the active design unless there’s no design at all.

    - **Context** – child of DesignerMenu. It appears at least one time and has a required “name” attribute. For details on setting the context name, see the “Context Name” section below.
2. Child elements of **Context**:
    - **TopMenu** – For RightClickMenu.xml, do not use this element. You can specify child elements of TopMenu listed below under Context. For any other xml files, you need at least one TopMenu child element for any Context.

      Child elements of TopMenu:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **MenuName** – required, appears only one time.
- **popupMenu** – may appear one or multiple times; child elements are the same as those of Topmenu.
- **LeafMenu** – may appear one or multiple times.

  Child elements of **LeafMenu**:

  - **MenuName** – optional string.

    Pay attention to the character reference "&" in XML. In order for the name of the menu to be displayed as “Tools”, the XML syntax must be:

    <MenuName>&Tool</MenuName>

    “&“ is the character reference for ”&” in XML, while “&Tool” tells Ansys Electronics Desktop that Alt+T is the shortcut key for this menu.

  - **MenuID** – optional number.

    For a list of valid MenuIDs, please see ***Command IDs for Customizing AEDT Menus.xlsx*** under <*install_dir*>/v<*release_number*>/[Win64 or Linux64]/Help.

  - **ShowBitMap** – optional, “Yes” or “No” (default)
  - **Accelerator** – optional string (Example: Ctrl+N)

  > **Note:**
  >
  > To add a new LeafMenu, both “MenuName” and “MenuID” are required.

- **CustomMenu** – may appear one or multiple times; used to add a customized menu to run an external script (vbs or python).

  Child elements of **CustomMenu**:

  - **MenuName** – see LeafMenu for Usage
  - **ShowBitmap** – see LeafMenu for Usage
  - **Accelerator** – see LeafMenu for Usage
  - **ScriptPath** – required string; used to supply path to the script (use of $PROJECTDIR, $PERSONALLIB, $USERLIB, $SYSLIB variables is also allowed).

    Example:

    <ScriptPath>C:/Users/jwei/Python/HelloWorld.vbs</ScriptPath>

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

or

<ScriptPath>$PERSONALLIB/HelloWorld.py</ScriptPath>

> **Note:**
>
> To add a new CustomMenu, both MenuName and ScriptPath are required.

- **Separator** – may appear one or multiple times; has no child element.

3. All elements have an optional attribute of **action**.

   Valid values are:

   - **add** – can be applied to all elements other than “DesignerMenu” or “Context” . This element and all its child elements will be added to its parent. An optional attribute of “position” can be followed here to specify the position of this newly added menu. Position starts from 1. If no position is specified, this menu will be appended to its parent menu.
   - **useDefault** – default menus will be used for this menu(matched by MenuName or MenuId). Any of it’s child menu will be processed according to its action setting.
   - **delete** – can be applied to all elements other than “DesignerMenu” or “Context” . This menu (matched by MenuName or MenuID) will be deleted from its parent.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

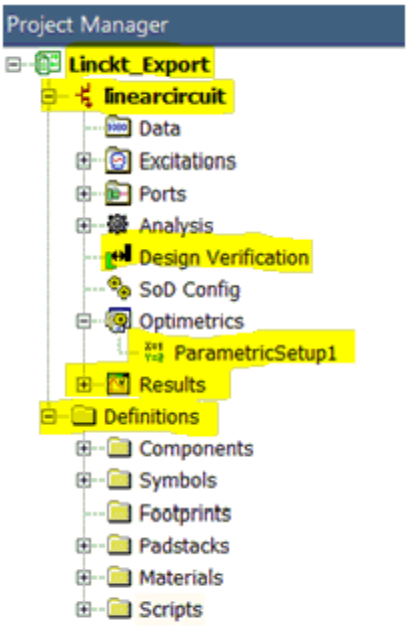
- For “DesignerMenu” and “Context”, only no action setting or “useDefault” is valid.
- The “DesignerMenu” action setting is significant because it is how you specify whether you want to construct your own menus or modify existing default menus. If this is set to “useDefault”, default menu settings are used for any menus not listed in your XML files. If no action is set, only the “File”, “Window” and “Help” menus display; no other default menus display. Menus listed in the XML file display after the “File” menu and before the “Window” menu. In this case, no action is needed for any other elements in the XML file. If any action is specified for any element, you will receive an error message and a default menu will be displayed for that context. A “Revert To Default” menu will be appended to the “Tools” menu. If no “Tools” menu is specified as a TopMenu, a “Tools” menu is created before the “Window” menu with one “Revert To Default” menu item.
- For any other elements, a missing action setting means that the parent action will be used.
- If any element’s action is set to “add”, then no action is needed for any of its child elements. If any action is specified, it will be ignored. This menu and all of its child menus will be added to the default menu.
- If any TopMenu or pop-upMenu’s action is set to “delete”, you don’t need to list any of its child menus. The menu will be deleted with all child menus.

Context Name

1. The “name” attribute specifies the name of the context for this product. You can have an "All" context if you set the action of DesignerMenu to UseDefault to specify a menu setting that you want to apply to all contexts for that product.
2. Valid context name for different xml files:
    - 2D Extractor.xml — “All”, “3d modeler”, “report2d”
    - CircuitDesign.xml — “All”, "SchematicEditor", “Layout”, “Netlist”, “report2d”
    - Circuit Netlist.xml —“All”, “Design”, “Netlist”
    - HFSS.xml — “All”, “3d modeler”, “report2d”
    - HFSS 3D Layout Design.xml — “All”, “Design”, “Layout”, “Layout3D Editor”, “report2d”
    - HFSS-IE.xml — “All”, “3d modeler”, “report2d”
    - NoDesignUI.xml — “All”, “No Context”, “project”, “FilterDesign”
    - Q3D Extractor.xml — “All”, “3d modeler”, “report2d”

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- RightClickMenu.xml — the following figure illustrates valid context names and how they display in Ansys Electronics Desktop.



  - **Project Folder** – "Project"; in the above figure, “Project” represents the “Linckt_Export” project folder.
  - **Design Instance Folder** – “Circuit Design”, “HFSS 3D Layout Design”, “HFSS”, “Q3D Extractor”, “Circuit Netlist”, “HFSS-IE”, “2D Extractor”
  - **Definitions Folder** – “Definitions”; in the above figure, the Definitions folder is highlighted.
  - For any Folder or Item under the Design Instance or Definitions folder, use the “/” notation to specify the path to the Folder or Item for context name. In the above figure: Use “Circuit Design/Results” for the Results folder. Use “Circuit Design/Design Verification” for the Design Verification folder.
    - For any folder or item under this, use the context name for the folder appended with “/Item”. In the above figure, “Circuit Design/Optimetrics/Item” represents ParametricSetup1.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> For the Report folder under Results, please use the context name for the Results folder appended with "Report"; for example: "Circuit Design/Results/Report".
>
> For the Trace folder under Report, please use the context name for the Report appended with "Trace"; for example: "Circuit Design/Results/Report/Trace"

Sample XML Files

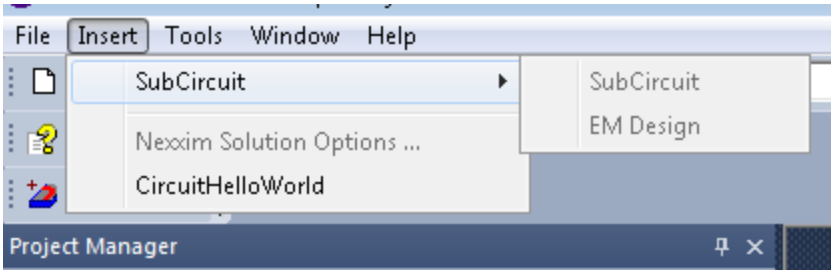
1. **Circuit Netlist.xml Sample**

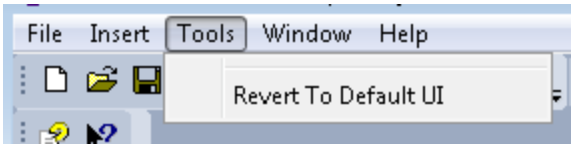
```
<DesignerMenu xmlns="http://www.ansys.com/uiConfigMenu">
   <Context name="Design">
      <TopMenu>
         <MenuName>Insert</MenuName>
         <pop-upMenu>
            <MenuName>S&amp;ubCircuit</MenuName>
            <LeafMenu>
               <MenuName>&amp;SubCircuit</MenuName>
               <MenuID>55500</MenuID>
               <ShowBitMap>No</ShowBitMap>
               <Accelerator></Accelerator>
            </LeafMenu>
            <LeafMenu>
               <MenuName>&amp;EM Design</MenuName>
               <MenuID>55502</MenuID>
               <ShowBitMap>No</ShowBitMap>
               <Accelerator></Accelerator>
            </LeafMenu>
         </pop-upMenu>
         <Separator></Separator>
         <LeafMenu>
            <MenuName>Nexxim Solution &amp;Options ...</MenuName>

            <MenuID>38460</MenuID>
            <ShowBitMap>No</ShowBitMap>
            <Accelerator></Accelerator>
         </LeafMenu>
         <CustomMenu>
            <MenuName>CircuitHelloWorld</MenuName>
            <ScriptPath>$USERLIB/HelloWorld.vbs</ScriptPath>
            <ShowBitMap>No</ShowBitMap>
         </CustomMenu>
      </TopMenu>
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
    </Context>
</DesignerMenu>
```

Menus created by processing the above sample, Circuit Netlist.xml





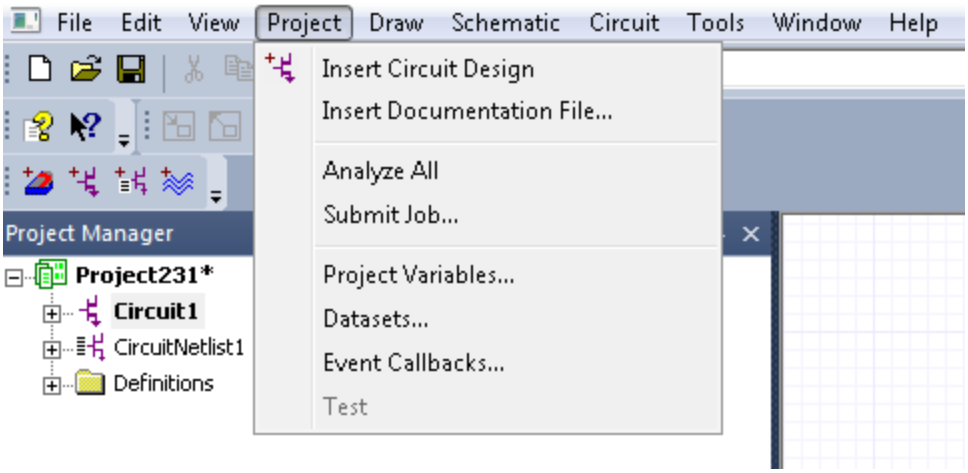
2. **Circuit Design.xml Sample**

```
<?xml version="1.0" encoding="UTF-8" standalone="no" ?>
<DesignerMenu xmlns="http://www.ansys.com/uiConfigMenu"
action="useDefault">
  <UseProjectWindowSelectionContext>false</UseProjectWindowSelectionC
  ontext>
  <Context name="All">
    <TopMenu>
          <MenuName>&amp;amp;Project</MenuName>
      <LeafMenu action="delete">
                <MenuName>Insert HFSS Design</MenuName>
      </LeafMenu>
      <LeafMenu action="delete">
                <MenuName>Insert HFSS 3D &amp;amp;Layout
        Design</MenuName>
      </LeafMenu>
      <LeafMenu action="delete">
                <MenuName>Insert HFSS-IE Design</MenuName>
      </LeafMenu>
      <LeafMenu action="delete">
                <MenuName>Insert Q3D Extractor Design</MenuName>
      </LeafMenu>
      <LeafMenu action="delete">
```

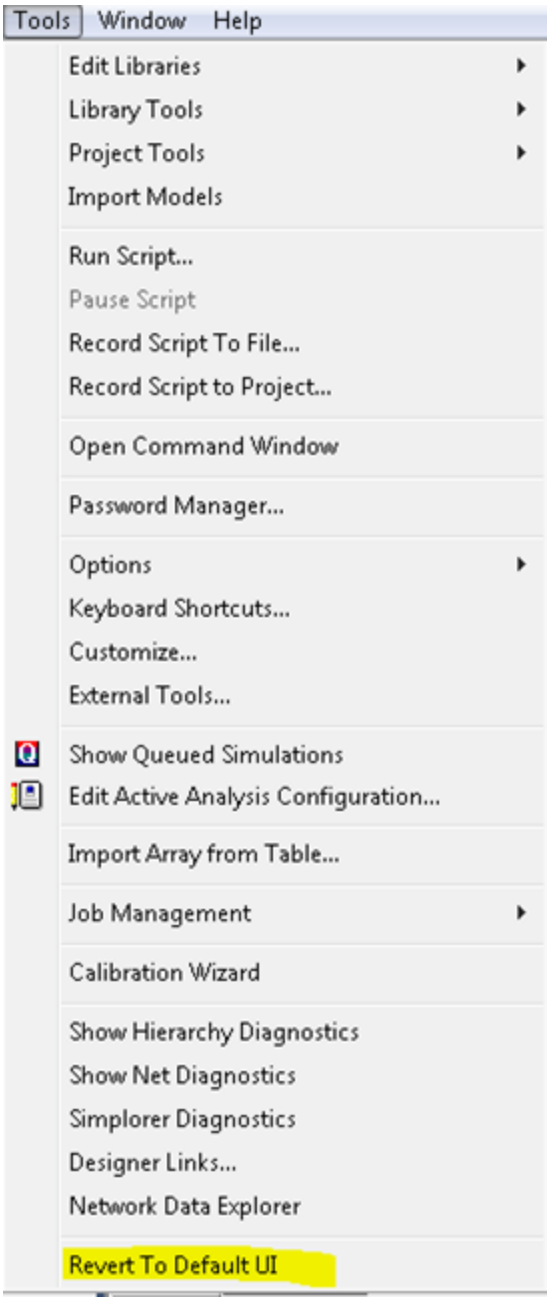

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
                <MenuName>Insert 2D Extractor Design</MenuName>
        </LeafMenu>
        <LeafMenu action="delete">
                <MenuName>Insert Circuit &amp;Netlist</MenuName>
        </LeafMenu>
        <LeafMenu action="add">
                <MenuName>Test</MenuName>
                <MenuID>3333</MenuID>
        </LeafMenu>
      </TopMenu>
   </Context>
</DesignerMenu>
```

Menus created by processing the above XML sample, Circuit Design.xml



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



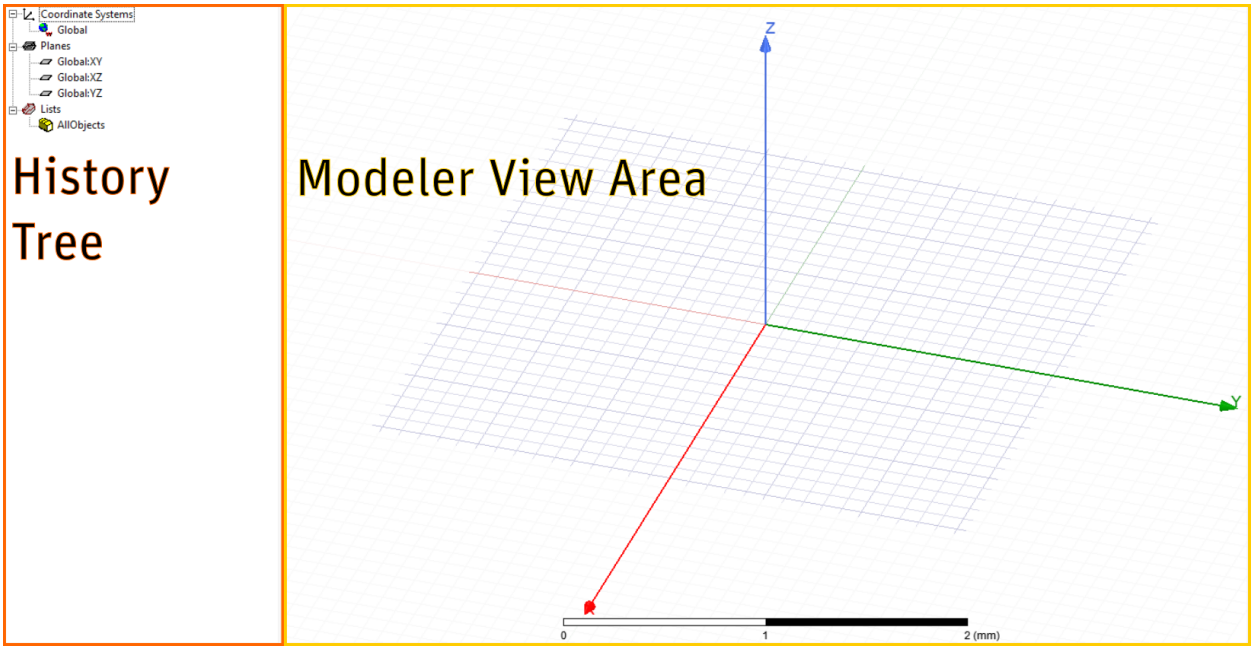
Ansys Electronics Desktop Design Area

The Design Area of the desktop can display one or more editor windows and report windows, depending on the type of designs you create or load. See the Desktop Windows topic for ways to manipulate the windows in the Design Area.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3D Modeler Window

The **3D Modeler** window is the area where you create the model geometry. It appears to the right of the **Project Manager** window after you insert an Ansys Electronics Desktop design into a project. The **3D Modeler** window consists of the modeler view area, or grid, and the History Tree. The two areas are highlighted in the following figure:



You can open a new 3D Modeler window in one of two ways:

- Insert a new Ansys Electronics Desktop design into the current project, or
- Double-click an Ansys Electronics Desktop design in the Project Tree

The model you draw is saved with the current project when you click **File** > **Save**.

Objects are drawn in the **3D Modeler** window. You can create 3D objects using Ansys Electronics Desktop's Draw menu commands, or you can draw 1D and 2D objects and manipulate them to create 3D objects. See: Drawing a Model.

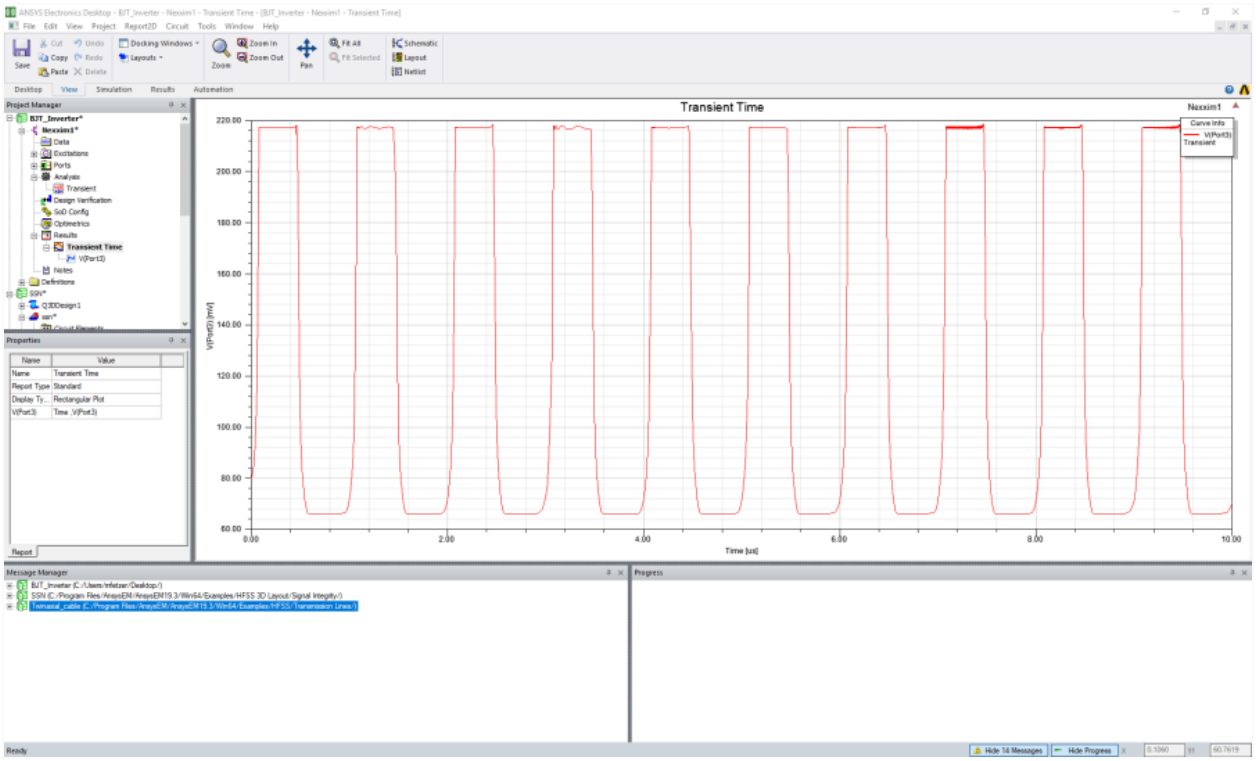
You can modify the view of objects in the **3D Modeler** window without changing their actual dimensions or positions. Besides menu and icon commands, you can also use **Alt+click**, mouse buttons, and keyboard shortcuts. See: Modifying the Model View.

Report Window

When a design has been successfully simulated, you can generate a report of results in a wide variety of forms, including XY graphs, polar graphs, 3D graphs, Smith charts, and data tables.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Various attributes of each can be customized to your liking. The following shows a 2D report window in Electronics Desktop:

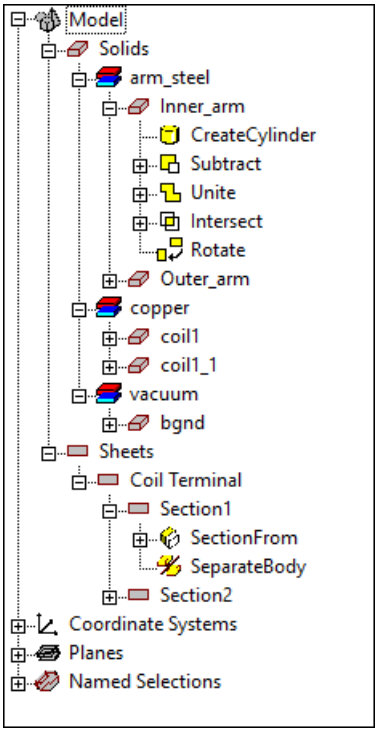


For more information, see Postprocessing and Generating Reports.

Ansys Electronics Desktop History Tree

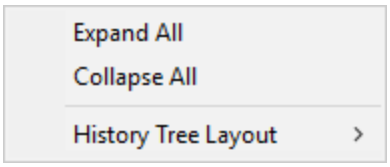
The history tree in the **3D Modeler** window lists the active model's structure and grid details.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Expand or collapse items in the tree using the **+** and **-** boxes.

Expand or collapse all items in a category by right-clicking to open a shortcut menu:



> **Note:**
>
> Some objects have more extensive shortcut menus than others, but all of them have these three options.

Selecting any item in the History Tree also selects it in the design area and populates the **Properties** window with its properties.

You can perform the following tasks with the History Tree:

- Organize the History Tree
- Control the View of Objects in the History Tree
- Use Group Commands for Modeler Objects

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select Objects in the History Tree
- View and Edit Commands on History Tree Objects

The History Tree can contain the following model details:

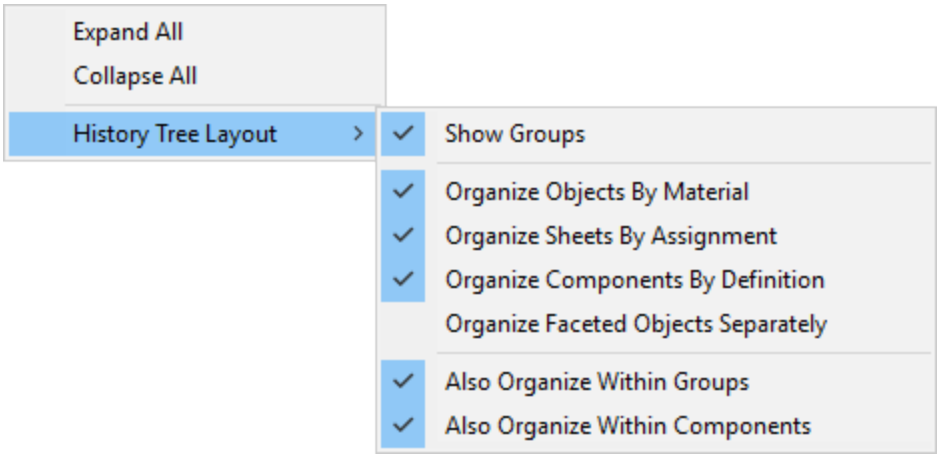
Invalid	Lists all invalid objects.
3D Component Names	Lists any 3D components added to the design. The History Tree lists their Solids, Coordinate Systems, and Planes separately from objects drawn in the modeler or imported to the modeler.
Solids	Lists the model's solid objects and a history of the commands carried out on each object.
Sheets	Lists the sheets in the model 3D design area.
Lines	Lists all line objects included in the active model. See Drawing a Straight Line for information on how to draw a line object.
Points	Lists all point objects included in the active model. See Drawing a Point for information on how to draw a point object.
Groups	Lists folders for Groups you have created or imported. See Group Commands for Modeler Objects.
Coordinate Systems	Lists all the coordinate systems for the active model. See Setting Coordinate Systems for more information on this model detail.
Planes	Lists the planes for all the coordinate systems. When you create a coordinate system, default planes are created on its xy, yz, and xz planes.
Named Selections	Lists the object or face selections for the active model. By default, a selection called *AllObjects* appears. You can add a specific group of objects or surfaces for field plot or for calculation. See: Creating an Object Selection and Creating a Face Selection.

> **Note:**
>
> While objects created in Maxwell can always be classed in the History Tree as a solid, sheet, or wire, some imported objects may have mixture of these. Maxwell places such objects in an Unclassified folder.

Organizing the History Tree

To organize and sort objects in the History Tree, right-click any object in the history tree to display the shortcut menu and select **History Tree Layout**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The shortcut menu contains the following commands:

- **Show Groups**
- **Organize Objects By Material:** This option is applicable to 3D objects and also to sheet objects that have the *Shell Element* attribute selected and a material selection specified (Mechanical - Thermal designs only). Imported solid objects that have no material specified yet are listed in a group named *Unassigned*. Non-model objects are listed in a *Non Model* group.
- **Organize Sheets By Assignment:** This option is applicable to sheet objects only (including Mechanical – Thermal shell objects with a boundary or excitation assigned). Sheet objects are grouped according to the boundary or excitation assigned. When a sheet is assigned both a boundary and an excitation, it is grouped according to the last assigned excitation (that is, the group name is the last assigned excitation). When multiple boundaries are assigned to a sheet, the sheet is grouped according to the boundary with the highest priority, and that boundary is used as the group name. The boundary priority depends on the design type, as follows:
  - **HFSS and Maxwell:** Priority is specified by the user.
  - **Icepak, Mechanical, and Q3D:** The last created boundary has the highest priority
- **Organize Components By Definition**
- **Organize Faceted Objects Separately**
- **Also Organize Within Groups**
- **Also Organize Within Components**

Controlling the View of Objects in the History Tree

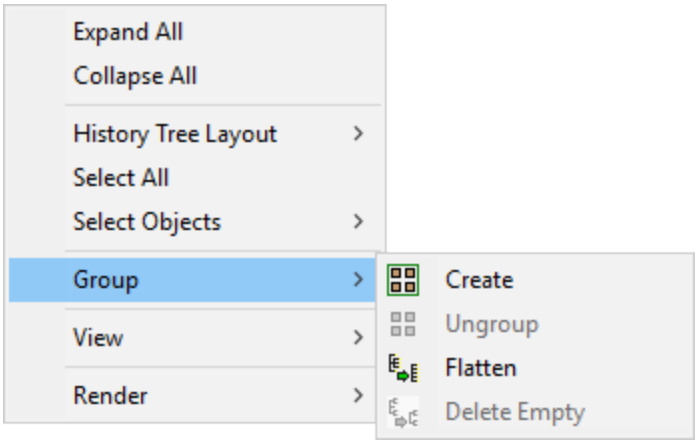
To control the view and visibility of an object such as a box or PML, right-click an object in the history tree to display the shortcut menu and select **View**. The shortcut menu contains the following commands:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Fit in Active View**
- **Hide in Active View**
- **Show in Active View**
- **Fit in All Views**
- **Hide in All Views**
- **Show in All Views**

Group Commands for Modeler Objects

The 3D Modeler allows you to group objects in the History tree. Besides predefined folders under **History Tree Layout** for solids, sheets, and material assignments, a set of **Group** commands allows you to create groups, ungroup objects, flatten a selected group's hierarchy, and delete empty groups. Groups are not compatible with 3D components, although, you can organize 3D components by definition in the History tree.

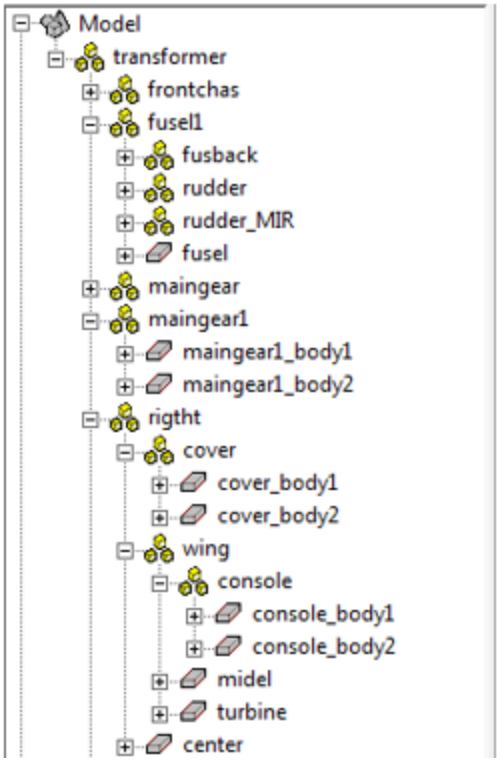


The **Group** commands support the following features:

- Group objects in the history tree. When objects are grouped, they show up under a sub-item in the history tree.
- Groups can contain sub-groups and sub-groups containing further sub-groups.
- Groups permit moving objects from one group to any other group at any time.
- Groups are purely for organization of history tree. They do not affect solution in any way.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Groups permit you to bring in MCAD assemblies and sub-assemblies as groups. See: Importing 3D Model Files.



- Groups can contain objects, submodels (for example, UDM, 3D Component, CAD links) and groups. Coordinate systems, planes etc will not have any parent group.

Object groups have following important distinctions from 3D components:

- Groups do not encapsulate history of objects. They do not encapsulate parameters used by those objects.
- Groups do not have history tree operations. For example, Arrange operations apply to all objects of the group rather than the group.
- Groups are not independent. Delete of an object in one group could cause object in another group to be also deleted.

Accessing Group Commands

You can access Group commands in several ways: via **Modeler > Group**, via the right-click shortcut menu in the History tree, via the right-click shortcut menu in the Modeler window, and via the Group icons on the Model tab.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Group Command Descriptions

Group > **Create** works with or without a selection. If the History Tree Layout > Show Groups command is disabled, a dialog box reports that groups are currently hidden, and that if you OK to continue, groups will be shown. If there is no selection of objects for new group, an empty group is created under Model. If there are selections, a new group is created under same group as all the selections, provided all selections are under one group. Once a group is created, all selections move under the new group. If selections are under different groups, then a new group is created under Model. Selections could be objects, groups, 3D Components, or User-Defined Model. The **Group** > **Create** command tries to find a more suitable group name where possible, based on names of all selections. If not, default group names are Group1, Group2, and so on. You can edit group names later.

Group > **Ungroup** is enabled only when you select one or more groups. Upon ungroup, all the contents of the selected group move under group's parent and the selected group is deleted. Note that **Group** > **Ungroup** is not recursive; it only affects the selected group is and all its children groups remain intact. Ungroup is also different from **Delete** (under **Edit** > **Delete**), which deletes a selected group and everything under it.

Group > **Flatten** is enabled when you select a group. You can select multiple groups at different levels. All of the selected group's contents (objects, sub-groups, and so forth) come directly under selected group. In a sense this is a recursive ungroup operation as all the children and grandchildren groups of selected group are ungrouped. If the History Tree Layout > Show Groups command is disabled, a dialog box reports that groups are currently hidden, and that if you OK to continue, groups will be shown.

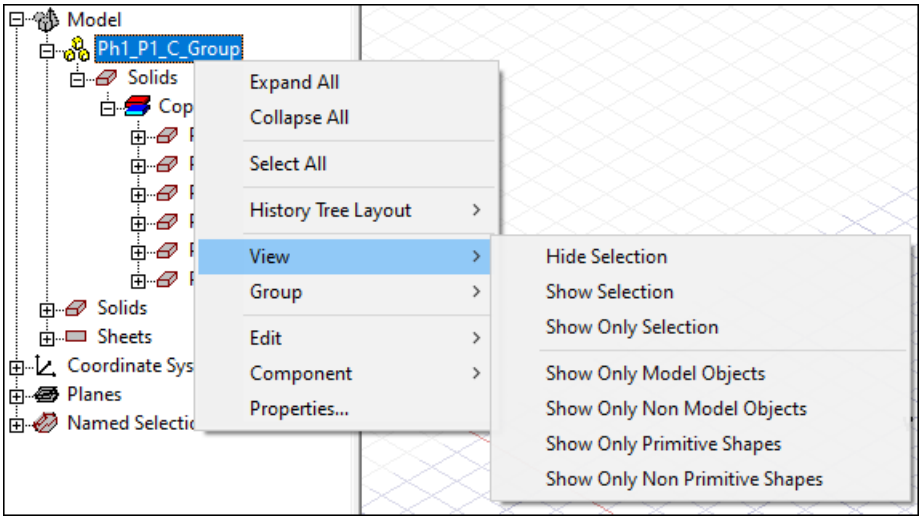
Group > **Delete Empty** deletes all empty groups under the selected group. If a selected group is empty it is deleted. This command is helpful to clean up empty groups after you have moved objects from one group to another. If the History Tree Layout > Show Groups command is disabled, a dialog box reports that groups are currently hidden, and that if you click **OK** to continue, groups will be shown.

Operations on Groups

View Options

For groups, you can hide or show the entire group. You can also opt to show only particular types of objects or shapes in that group. The right-click view options are shown below:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Edit/Delete

Delete operations delete group and all the children, including sub-groups, under the group.

Copy and Paste of Objects in Groups

When you copy and paste objects with groups to another design, group information is carried over, that is, new group in target design is created. If a group with the exact name exists, then the objects are moved under that group and no new group is created. When pasted in the same design, the pasted object moves under same parent group as original object.

Copy and Paste of Groups

When you copy and paste one or more groups in same design or to another design, a new group is created with unique name derived from original group name. Everything under that group, including any sub-groups, is copy/pasted as well.

Copy and Paste of Groups and Objects

If you multi-select a few objects and groups, all of them are pasted correctly. If you select a group and few objects under that group, copy and paste includes the complete group structure. This means that selection of objects under the selected group is ignored. If you want to only copy and paste few objects from group, you should only select those objects and NOT select the group.

Arrange Operations on Groups

You can select a group to enable arrange operations like move, rotate and mirror. Note that arrange operations works directly on objects under selected group. So after arrange operation is done, the History tree shows the arrange operation under every object of that group.

Duplicate Operations on Groups

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can select groups to enable duplicate operations. Afterward, the History tree shows a duplicate operation under every object of that group. Newly created objects appear under a new group. There will be new group for every instance of duplicate.

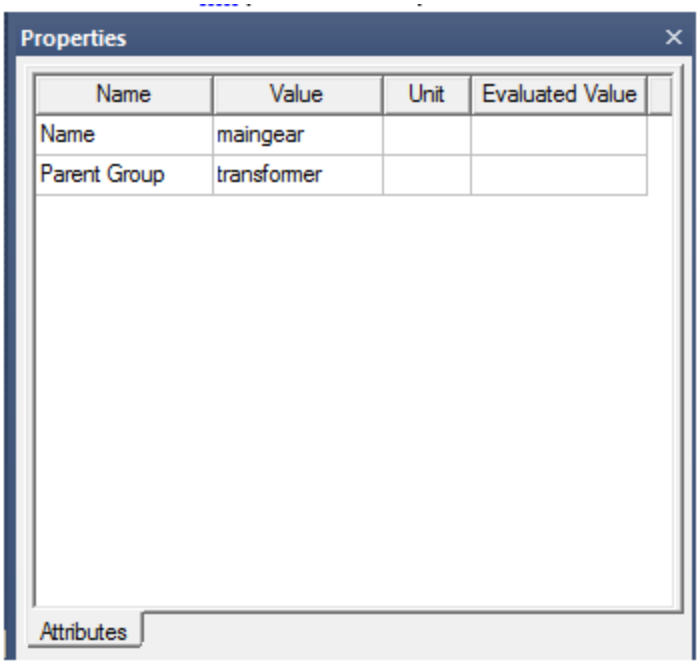
Other Operations on Groups and Objects in Groups

Modeling operations that create new objects, such as Create from Face or Edge, as well as Simplify and Separate are available for groups. The newly created objects appear under a new group created under the original object's group, based on the tool option settings. See 3D Modeler Options: Group Options.

If the group has hierarchy, you can also flatten it.

Group Properties

A Group's Property window is shown when you select a group in the history tree. The Group's properties include Name and parent Group.

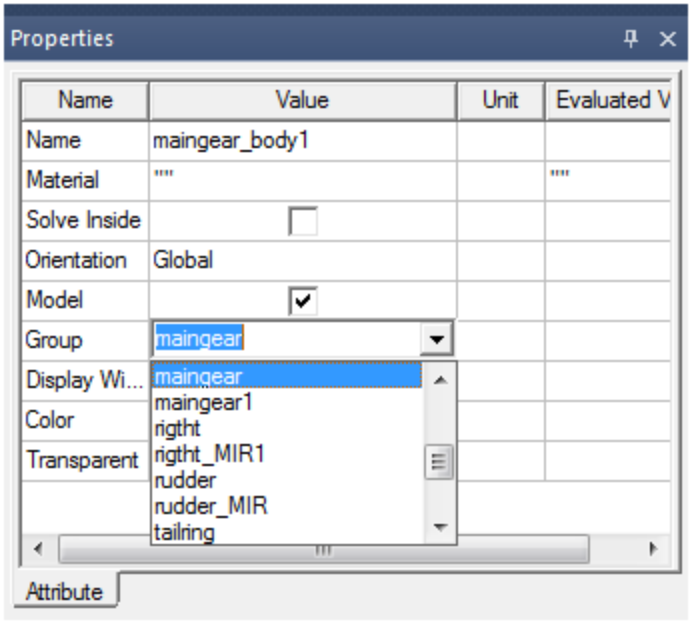


Group Property Window

Object, group and submodel properties have a Group property. You can edit the Group Property and this provides another way of setting an object's (or group's) group. You can select any of the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

existing groups listed or create a new group.



Group assignment could be changed in Object's property window

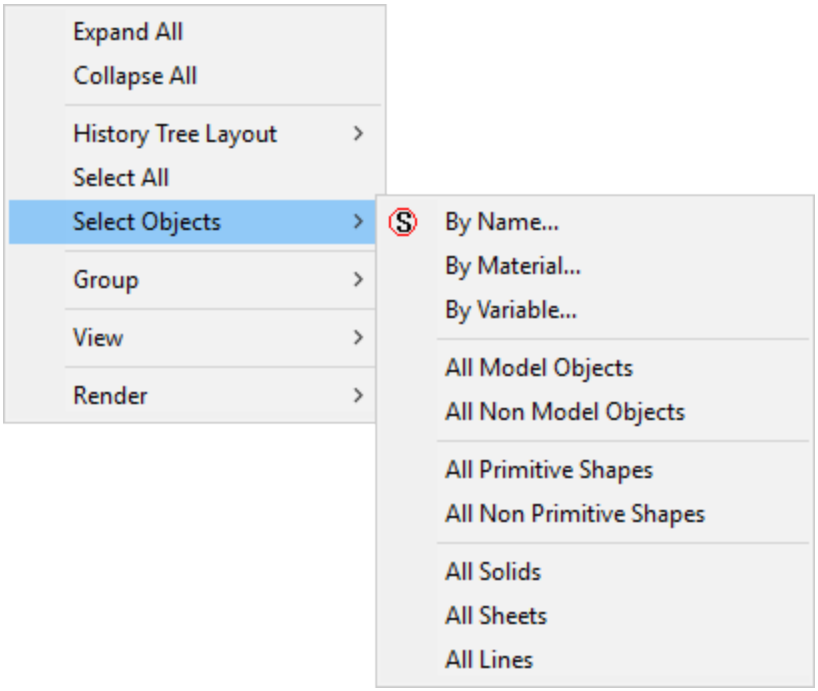
Selecting Objects in the History Tree

Selecting objects in the History Tree also selects them in the View window. This can be useful for complex objects, when it may be easier to find objects of interest by name or material because the object is inside or behind others.

Ctrl+click to make multiple selections in the History Tree. You can select a range of objects by clicking the first and using **Shift+click** to select all in the range. You can also click and drag the mouse to make rubber band selections. Only visible objects are selected. That is, if the hierarchy is closed under the selection, any operand parts are ignored and do not interfere with cut and paste operations.

Right-click **Model** in the History Tree to see additional selection options:

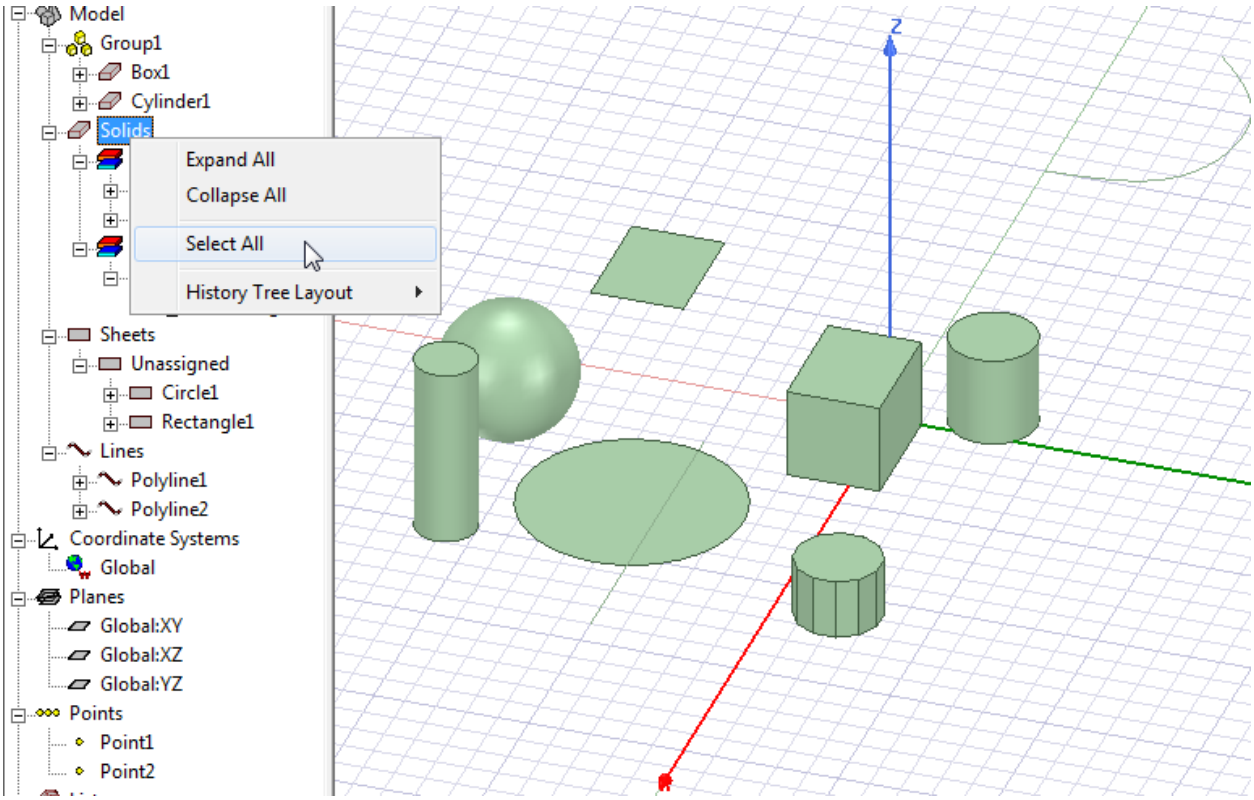
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can right-click any History Tree object or grouping to display a shortcut menu with a **Select All** option. This lets you select all non-grouped solids, sheets, lines, non-model objects, or all unclassified objects, as well as all non-grouped objects assigned to specific materials.

For example, right-click Solids and then **Select All**:

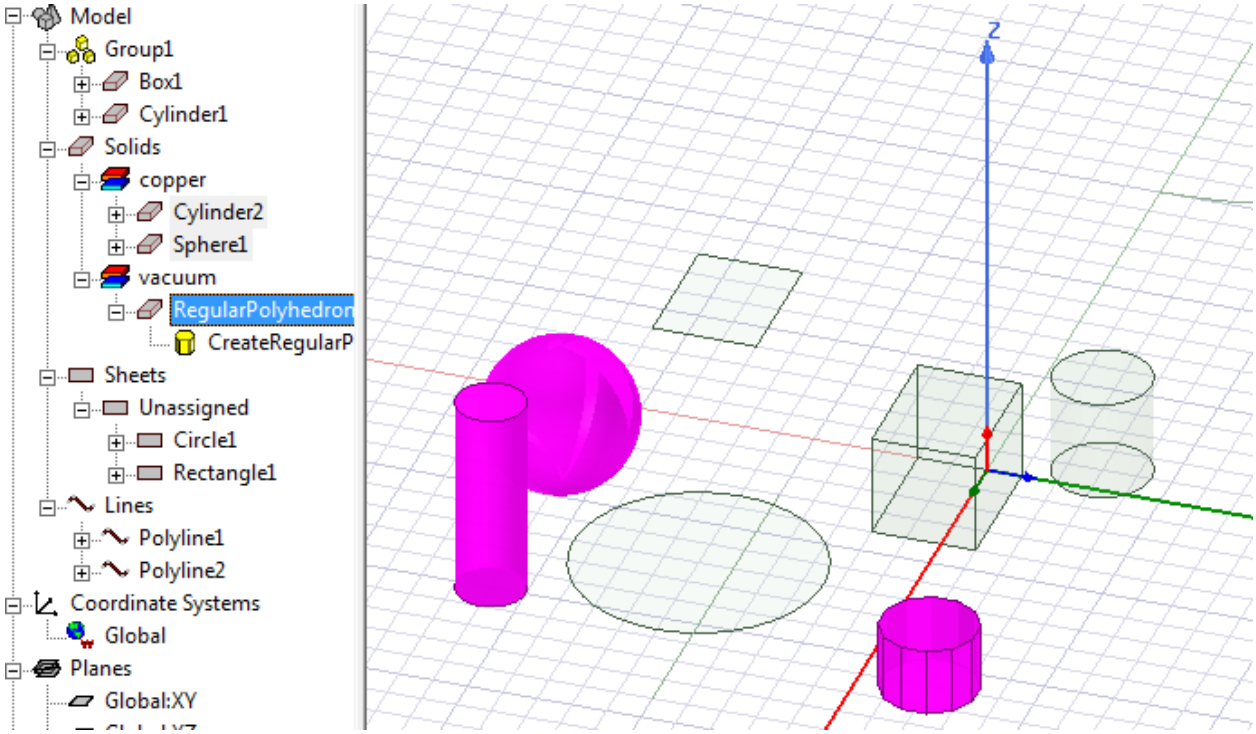
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After clicking **Select All**, the History tree and the **Model** window highlight the selected Solid Objects.

Notice that Box1 and Cylinder1 are not selected because they are in Group1.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



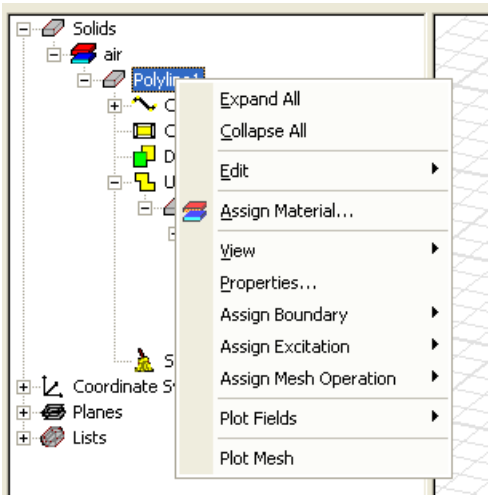
The right-click for Model and **Select All** does select the members of Group1 because they are part of the model.

If you select a geometric object, such as a Box or Cylinder, the shortcut menu **Select** command offers choices to select **All Faces**, **All Edges**, or **Sheet Edges** for that object.

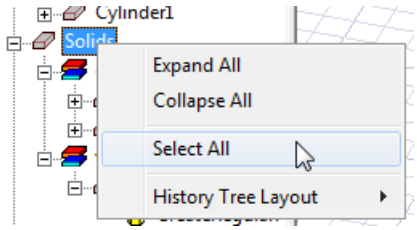
Shortcut Menus in the History Tree

If you select particular objects in the history tree, right-clicking displays a shortcut menu that lists the commands that you can apply to the selected command, object, or objects. The shortcut menus for model objects are most extensive.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

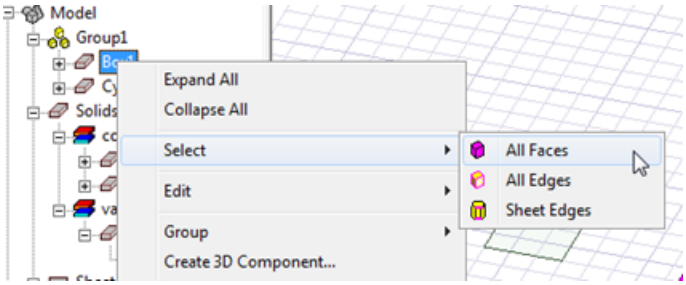


If you right-click on a model or category such as Solids, Sheets, Lines, or a material in the history tree, the shortcut menu includes a **Select All** command.



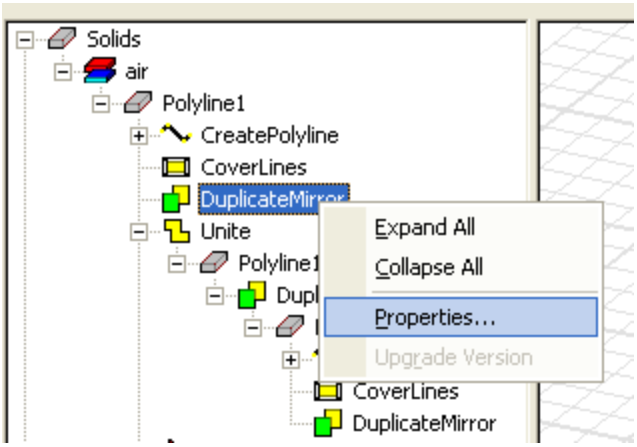
The behavior of the shortcut **Select All** command depends on the history tree model, object, or property icon from which you invoke the shortcut menu. For explanation and examples, see Select Objects in the History Tree.

If you select an object in the history tree, the menu offers selections for that object:



In other cases, you can only view properties, or expand or collapse hierarchy.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



View Item Properties in the History Tree

To view the properties of an item in the history tree:

- Click the item's name in the history tree.

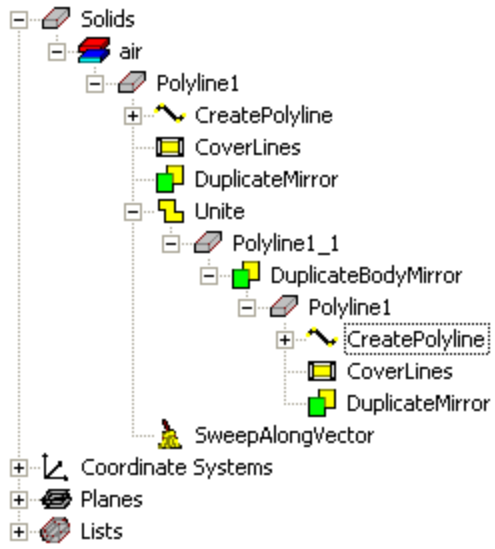
  The item's properties appears in the docked **Properties** window.

- If the Properties window is not visible, double-click on an item in the history tree to display a **Properties** window.
- Click the item's name in the history tree, and right-click to display a shortcut menu. Then select **Properties** to display the **Properties** window.

Viewing and Editing Commands on History Tree Objects

The history tree also lists the history of all commands carried out on a model's objects (for example, "CreateBox" or "Subtract"). This history is displayed in the order in which it occurred. Here is a history tree from the waveguide combiner example project:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

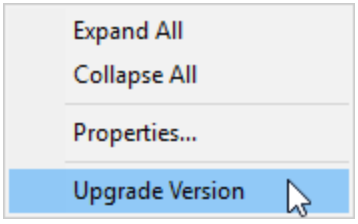


Selecting a command in the history highlights the object in the Modeler window and shows that object's properties (if available) in the docked **Properties** window. You can look at the fields in the **Properties** window to see any editable fields for that command, such as coordinate system, line type, coordinates, or units.

For some commands, such as **Edit** > **Arrange** > **Move** or boolean operations, selecting them in the history tree enables the delete icon (**X**) on the toolbar and the **Edit** > **Delete** menu. In these cases, you can delete those commands from the history tree as a way of undoing those operations. As an alternative to deletion, you can check **Suppress command** in the **Properties** window for that command. This undoes the effect of a command on an object without removing it from the History Tree.

Upgrading Version in the History Tree

Right-click an operation in the History Tree to see the **Upgrade Version** command.



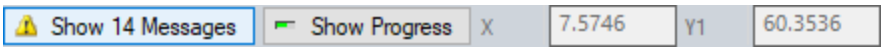
By default, the Modeler opens legacy projects using the modeling kernel version under which the project was saved, so that users don't see side effects from slight topology changes between versions. The **Upgrade Version** operation forces Electronics Desktop to use the latest modeling

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

kernel. Typically, this command would only be used when a needed fix is available for the newest version.

Ansys Electronics Desktop Status Bar

The status bar is located at the bottom of the application window. By default, it contains buttons to Show or Hide the Message window and Progress window.

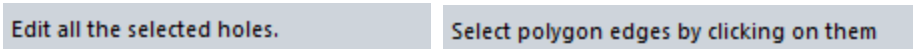


Note that the Show Messages button indicates the number of messages.

When more than one progress bar is active, the top progress bar is represented on the status bar with a progress indicator:

The status bar also displays helpful information about the current selection or command.

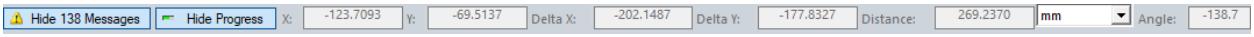
Directions for inputs appear on the left side of the status bar:



Depending on the command being performed, the status bar can display the following:

- **X**, **Y**, and **Z** coordinate boxes
- A drop-down menu for entering absolute, relative, cartesian, cylindrical, or spherical coordinates
- The model's units of measurement

These coordinate boxes and drop-down menus appear on the right side of the status bar:



To toggle status bar display:

- Click **View** > **Status Bar**.

A check mark next to this command indicates that the status bar is visible.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Keyboard Shortcuts

The following keyboard shortcuts apply to Maxwell in general:

F1	Help (context-sensitive)
Ctrl + F4	Close window
Alt + F4	Close program
Ctrl + C	Copy
Ctrl + N	New Project
Ctrl + O	Open
Ctrl + P	Print
Ctrl + V	Paste
Ctrl + X	Cut
Ctrl + Y	Redo
Ctrl + Z	Undo
Ctrl + 0	Cascade windows
Ctrl + 1	Tile windows horizontally
Ctrl + 2	Tile windows vertically

See the subtopics in this Help branch for additional information concerning shortcuts and view navigation.

To customize the shortcut assignments, use **Tools** > **Keyboard Shortcuts**. Not all shortcuts are customizable.

Desktop Shortcuts

Modifier + key	Hold down the modifier, such as Shift or Ctrl, and press the key.

The following Ansys Electronics Desktop shortcut key combinations are available at any time:

Ctrl + N	New
Ctrl + O	Open
Ctrl + P	Print
Ctrl + S	Save
Ctrl + 0	Cascade windows
Ctrl + 1	Tile windows horizontally

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ctrl + 2	Tile windows vertically
Delete	Delete
F1	Open help

3D Modeler Window Shortcuts

The following shortcut key combinations are available when the Modeler window is active. View navigation shortcuts are also applicable to 3D Plot windows. The table below is organized into four sections, with the shortcuts separated according to the following categories:

- Commands and options used for the selection of entities
- Choosing the entry mode to use when creating geometry
- View navigation (rotating, zooming, panning, choosing a view orientation)
- Choosing the geometry rendering mode (wireframe or shaded)

> **Note:**
>
> In the following table, **LMB** is an abbreviation for *Left Mouse Button*, and **MMB** means *Middle Mouse Button*.
>
> **Wheel** refers to rotation of the mouse wheel that is integrated with the middle button.

Entity Selection Commands and Options:	
B	Select face or object **behind** current selection
F	Select **faces** mode
O	Select **objects** mode
E	Select **edges** mode
V	Select **vertices** mode
M	**Multi-select** mode
Ctrl + A	Select **all** visible objects
Ctrl + Shift + A	Deselect all objects
Geometry Properties Entry Mode:	
F3	Switch to the **point** entry mode (using the mouse or coordinate text boxes to draw objects). Switching takes effect only after the drawing of an object has started.
F4	Switch to the **dialog box** entry mode (draw objects by specifying properties in the Command and Attribute tab of a dialog box). Switching takes effect only after the drawing of an object has started.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

View Navigation:	
Ctrl + D	**Fit** view
Ctrl + '+'	**Zoom in** (screen center)
Ctrl + '-'	**Zoom out** (screen center)
Alt + double-click	Change the view orientation to one of nine predefined views depending on where you double-click within the Modeler window. See Changing the Model View with Alt+Double-Click Areas for more information.
Alt + *arrow keys*	**Rotate** the view about a vertical or horizontal axis. Each press of an arrow key rotates the view 2.5°.
Ctrl + *arrow keys*	**Pan** the view vertically or horizontally. Each press of an arrow key pans the view 10 screen pixels.
MMB + drag (no modifier)	Freely **rotate** the view orientation (except when clicking and dragging near the window border). When clicking near the window border, rotation is constrained about the screen-normal axis.
MMB + X + drag	Constrained **rotation** of the view orientation about the **X** axis
MMB + Y + drag	Constrained **rotation** of the view orientation about the **Y** axis
MMB + Z + drag	Constrained **rotation** of the view orientation about the **Z** axis
Wheel	Rotate the mouse wheel to **zoom** in and out. Rotate the top of the wheel forward (away from the mouse) to zoom **in** closer, and rotate it rearward to zoom **out**.
Shift + MMB + drag	**Zoom** in or out. Moving the cursor upward zooms in closer. Moving it downward zooms out. (* see note below)
Ctrl + MMB + drag	**Pan** the view (* see note below)
(*) **Note:** The listed actions for **Shift** and **Ctrl** keys in combination with the **MMB** are the view navigation assignments for 2025 R2. Simultaneously, legacy hot keys using **Shift** or **Alt + Shift** in combination with the **LMB** for these functions continue to be supported by default. Optionally, you can choose to disable the legacy shortcuts that were applicable to 2023 R2 and earlier versions. See Choosing the View Navigation Options for more information.	
Geometry Rendering Modes:	
F6	Render the geometry as wireframes (no shading of faces)
F7	Render the geometry with smooth, shaded faces

To customize shortcut assignments, use **Tools** > **Keyboard Shortcuts**. Not all shortcuts are customizable.

Report Shortcuts

Modifier + key	Hold down the modifier (such as Shift or Ctrl) and press the key.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following shortcut key combinations are available when a Report window is open:

Ctrl + A	Select all traces
Ctrl + C	Copy selected trace
Ctrl + V	Paste
Ctrl + X	Cut selected trace
Ctrl + Y	Redo insert report
Ctrl + Z	Undo insert report
Delete	Delete

Choosing the View Navigation Options

New mouse button and modifier key assignments (hotkeys) have been added for navigating the model view (rotate, pan, and zoom functions). Simultaneously, legacy hotkeys for these functions continue to be supported by default. Optionally, you can choose to disable the legacy view navigation shortcuts that were applicable to version 2023 R2 and earlier, as follows:

1. Access the *Options* dialog box using one of the following two methods:
   - From the menu bar, click **Tools** > **Options** > **General Options**.
   - From the **Desktop** ribbon tab, click **General Options**.
2. In the tree at the left side of the dialog box, select **General** > **User Interface**. Then:
   - To use either the legacy or current view navigation buttons and hotkey combinations, select **Enable Legacy View Navigation**. This option is selected by default.
   - To use only the current view navigation behavior and disable legacy button/hotkey support, ensure that **Enable Legacy View Navigation** is cleared.
3. Click **OK**.

   The navigation option change becomes effective immediately (no program restart required).

Current vs. Legacy View Navigation

The following table summarizes changes in mouse-button and hotkey assignments between the current scheme (version 2024 R1 and newer) and the legacy scheme. "*Legacy*" assignments are applicable to version 2023 R2 or earlier and when the *Enable Legacy View Navigation* option is selected in newer versions):

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The *"Current"* assignments are available whether or not the *Enable Legacy View Navigation* option is selected:

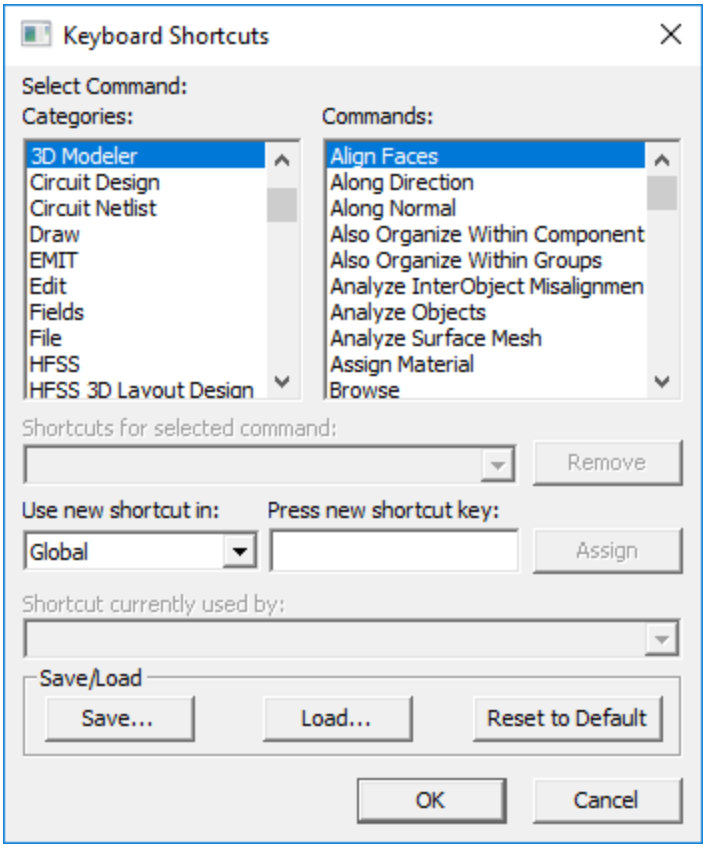
Function	Key / Mouse Button Assignment	
	Current	**Legacy**
Pan (all contexts)	Ctrl + MMB + drag or Ctrl + *arrow keys*	Shift + LMB + drag or Ctrl + *arrow keys*
Rotate (3D contexts)	MMB + drag or Alt + *arrow keys*	MMB + drag or Alt + LMB + drag or Alt + *arrow keys*
Zoom (all contexts)	Wheel or Shift + MMB + drag or Ctrl + "+", Ctrl + "-"	Wheel or Alt + Shift + LMB + drag or Ctrl + "+", Ctrl + "-"

- "**LMB**" = Left Mouse Button
- "**MMB**" = Middle Mouse Button
- "**Wheel**" refers to rotation of the mouse wheel.
- In some 2D contexts, where rotation is not applicable, you can use **MMB + drag** to pan the view for either scheme.
- For **Alt + *arrow keys***, rotation is about a vertical or horizontal axis, regardless of the model view orientation.
- For zooming, if using the "=\+" key, instead of the "+" key on the numeric keypad, do **not** press Shift.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Custom Keyboard Shortcuts

Click **Tools** > **Keyboard Shortcuts** to open the **Keyboard Shortcuts** window. Here, you can view existing assignments, create new shortcuts, and save or load assignment files.



Selecting a **Command Category** populates the **Commands** list with the available commands for that category. If the command has an assigned shortcut, it is displayed in the **Shortcuts for the selected command** field. You can use the **Remove** button to disable the shortcut for the selected command. If the selected command does not have an assigned shortcut, the **Shortcuts for selected command** field and the **Remove** button are unavailable.

To create a new shortcut key:

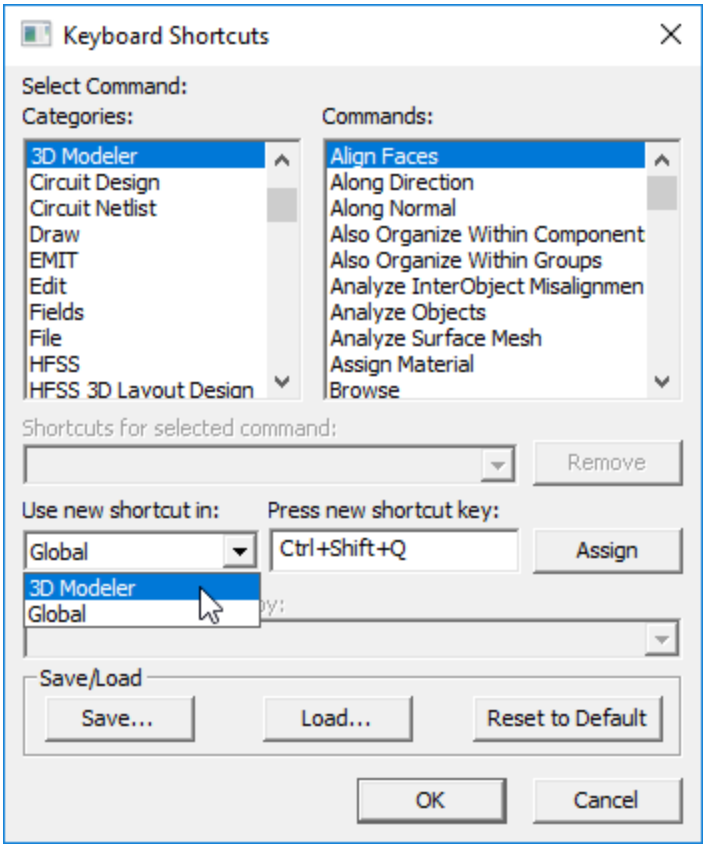
1. Select the **Category** and **Command**.
2. If you want to disable a current assignment for the selected command, click **Remove**.
3. To assign a keyboard shortcut, place the cursor in the **Press new shortcut key** field.

   The field displays the keystrokes you make. When you have made keystrokes, the **Assign** button becomes available. If you combine keystrokes, these are displayed with a plus sign (+) between them. For example, Ctrl+P or Alt+O.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the shortcut you select is currently in use, that information displays in the **Shortcut currently used by** field.

4. The **Use new shortcut in** drop-down list displays **Global** by default, which means that the shortcut will apply to all applicable contexts. If a limited context exists, the menu offers a selection.



5. When you have made your desired assignments, you can save them to a named file.

   Click **Save...** to save the assignments to your desired location in Ansoft Keyboard Shortcut (*.aks) format.

   If you have an existing *.aks file, use the **Load...** button to locate and select the file.

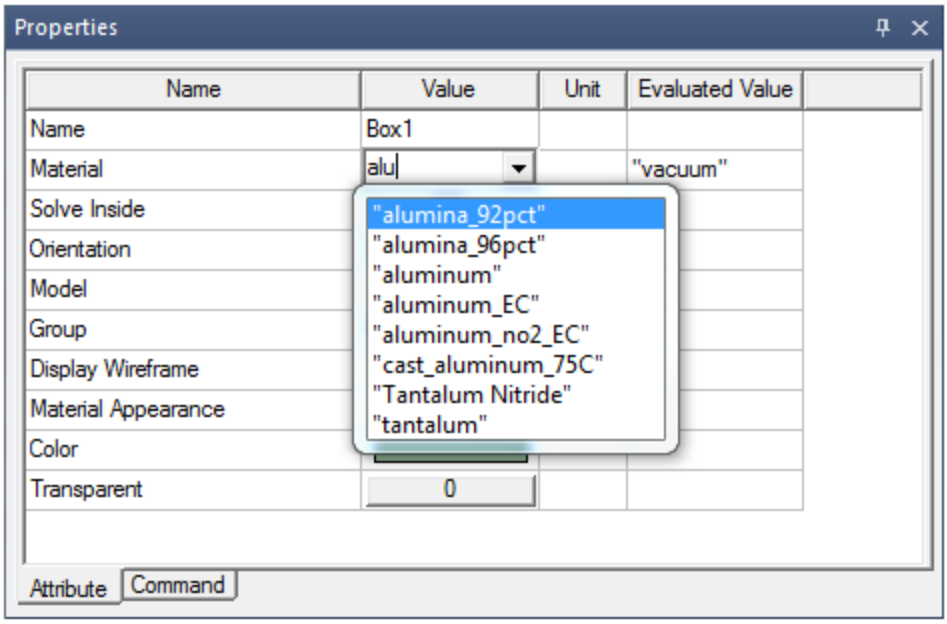
6. Click **OK** to save the current settings, or **Reset to Default**.

Auto-Complete for Variables and Properties in Electronics Desktop

When you edit a properties or variable text field, Electronics Desktop can display possible matches for what you type. This can help if a variable or material name is long. You can save

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

time by selecting a pre-determined match rather than typing out the entire name. The following figure shows an example of auto-complete for material names:



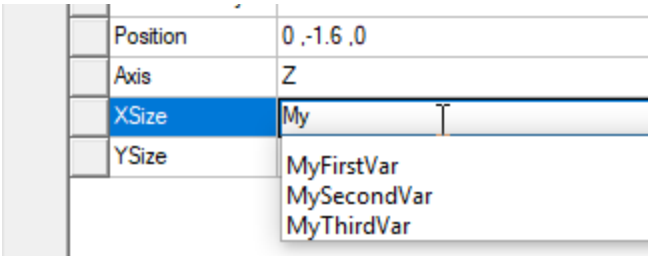
Using Auto-Complete

Certain commonly used text fields have auto-complete configured. When you start typing in these cases, matches display in a list below the text field. If there is no matching text, then no list displays. The list is automatically sized, but you can resize it. Electronics Desktop remembers the new size when doing additional auto-complete matching in the same text field. Switching to a different text field resets the sizing.

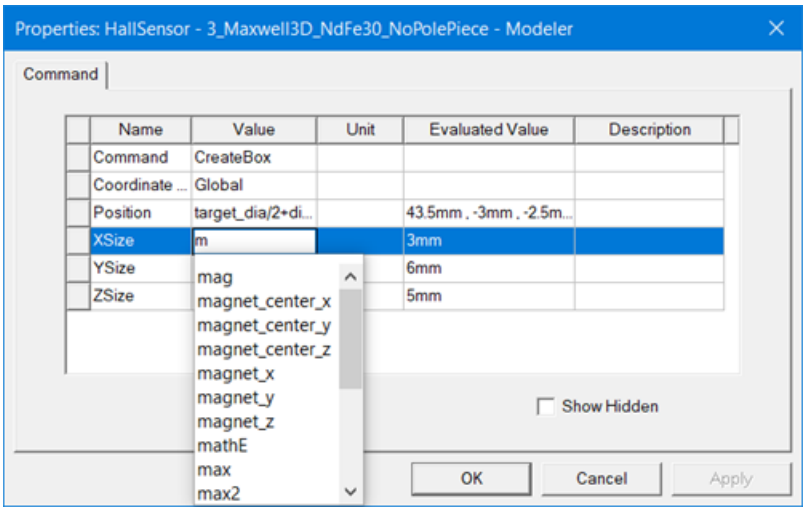
Hitting **Tab** or **Enter** accepts the current selection for auto-complete, which replaces the text typed with the full auto-complete match and hides the list. Otherwise, hitting the escape key **Esc** hides the auto-complete list. Typing more letters causes auto-complete matching to resume.

Where you select design properties, auto-complete works with expressions for the values, providing matches for the names of constants, intrinsic variables, functions, project variables (prefixed with “$”), and design properties. When project variables are displayed (that is, you select the project in the Project tree), auto-complete for value expressions also works, except without matching design properties. The variable auto-complete list does not include Separator Variables or Hidden variables since these could cause invalid formulas.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

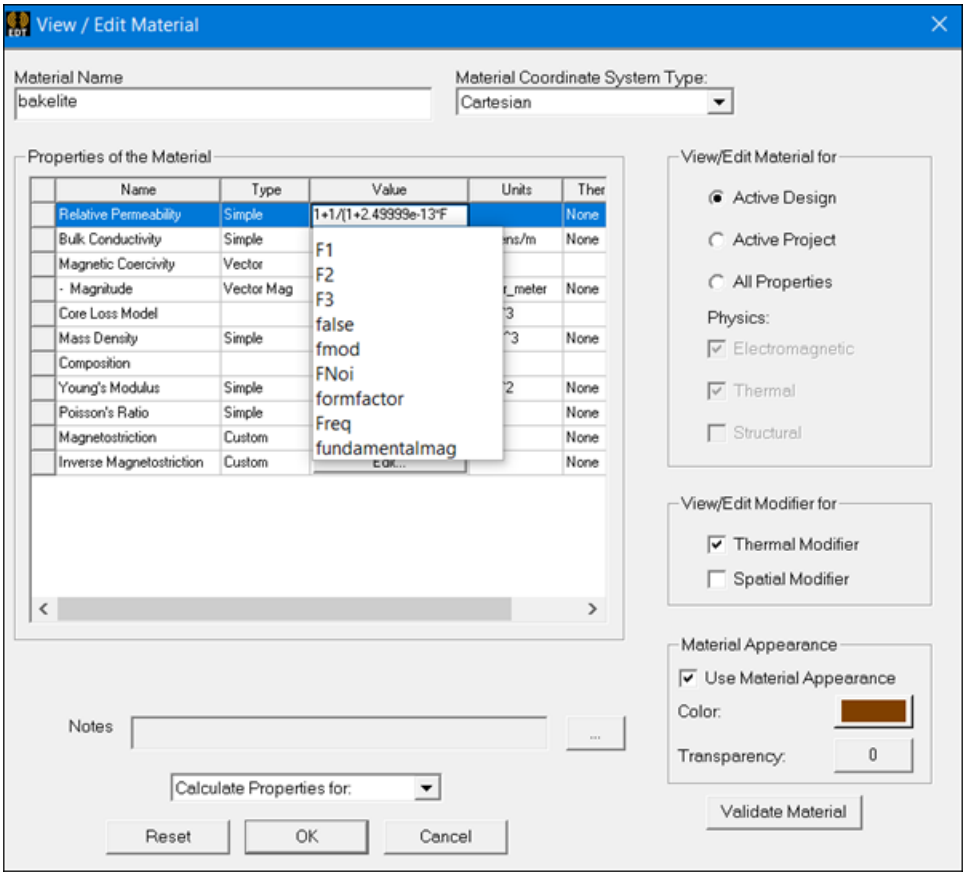


Properties of other items may also have auto-complete configured to work with value expressions (for example, properties of a CreateBox command or of a circuit component). The following shows a design property with auto-complete matching:



Auto-complete also works with value expressions for material properties:

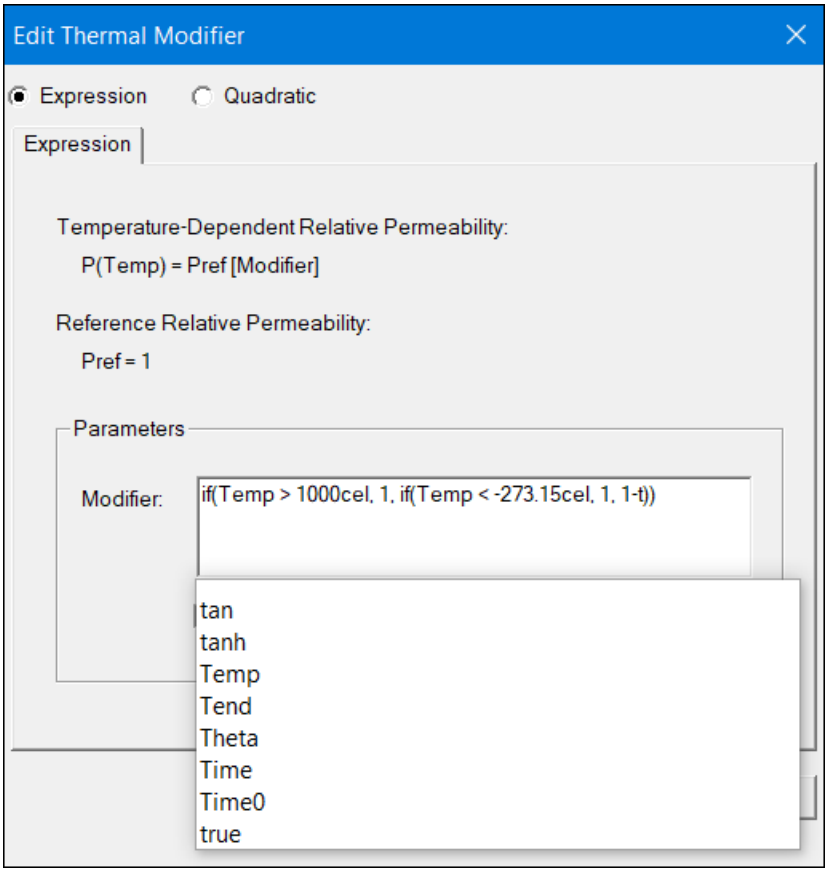
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note the scroll bar, indicating that there are more matches than those currently displayed.

Auto complete also works with thermal modifier expressions for Materials:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Using the Password Manager to Control Access to Resources

Ansys Electronics Desktop lets you specify library resources that require password access, and encryption of those resources. For convenience, the same password can apply to multiple resources. To access the **Password Manager,** click **Tools** > **Password Manager**. This displays the **Password Manager** window.

To Specify a New Password Protected Resource

1. Click **Tools** > **Password Manager**.
2. Click **New**.

   This opens the **New Encrypted Resource** dialog box.

3. Specify the name of the resource that you want to protect and click **OK**.

   This displays the **Enter Passwords** window, which contains radio buttons to let you:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Enter Password and confirm for **Full Access** or for **Execute Only Access**.
- **Use Ansys Password** (for execute only). This option does not require you to enter a password, but still encrypts the library.

4. Once you have selected a radio button and specified any necessary passwords correctly, click **OK**.

   The **Password Manager** updates to show the resource.

To Encrypt a Resource

1. Click **Tools** > **Password Manager**.
2. The **Password Manager** window appears.
3. In the Password Manager, select an existing resource to highlight it.
4. Click **Encrypt File**.

   A file browser window appears.

5. Use the drop-down menu to select the appropriate **Files of Type** filter.

   Any existing resources in the selected directory will appear.

6. When you have selected the appropriate resource, click **OK**.

   This encrypts the resource.

> **Note:**
>
> The **Expire** resource on option lets you select the date when the password expires for the relevant resource.

Running Ansys Electronics Desktop from a Command Line

Ansys Electronics Desktop includes line arguments that can be included when launching from a command line or terminal prompt. All command-line arguments are case-insensitive. The commands associated batch options can also be used with a Job Management Interface for submitting jobs to Ansys or RSM and other supported schedulers.

Command-line Syntax

```
ansysedt <options> <run command(s)> <project file/script file>
```

It is good practice to put quotation marks around the path to the solver executable, and around the full path to the project. This ensures that spaces in the path or project will not be an issue. The same is true of the design name, if there are spaces.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Run Commands

The following command line run commands are available for Ansys Electronics Desktop. Of the commands (BatchSave, BatchSolve, BatchExtract, RunScript, RunScriptandExit), one or none must be specified as arguments after the solver executable. When none is specified, you may specify a project or archive to open when Electronics Desktop launches, and can only use the **-Iconic** and **-Help** options. The commands are further described below:

- **-BatchSave** [options] **<***file or folder specifier***>**

  Saves a named project or folder containing one or more project files or folders to the current version. You can run this command with the **-Iconic**, **-Logfile**, and **-ng** options.

  If no migration is necessary (*e.g.* a project containing a circuit design): The original file will be overwritten with the new version, and no backup will be made.

  Special behavior when opening backed up files from the user interface: Files located in the AnsysEM_Backup directory would be the original non-migrated files that were backed up during an earlier –batchsave or open. Opening a file located in the AnsysEM_Backup directory via the open file dialog box will invoke the Parasolid migration, and the migrated file will be saved in the parent folder of the AnsysEM_Backup folder with a unique name so as not to overwrite any existing files.

  If a ***file*** is given, **-batchsave** will run on the specified file.

  If a ***folder*** is given, **-batchsave** will run on all files underneath the specified folder.

  .aedt, .aedtz, and .a3dcomp files will be processed.

  Allowed options:

  norecurse: only process files in top level of input folder

  forcearchive: project files are converted, then saved as archive files.

  previewonly: don't perform actual conversion, instead write preview of filenames to batchlog.

  outputfolder=<outputfolder>: save converted files to <outputfolder>, preserving subfolder structure.

- **-BatchSolve <***project file name***>**

  By default, this run command solves all adaptive setups, Optimetrics setups, and sweeps found in the project file. You can run this command with the **-Iconic**, **-Logfile**, **-ng**, and **-WaitForLicense** options. If parallel solve is possible, you can use the **-Distribute** option.

  If you wish to specify which setups **-BatchSolve** completes, you can use additional parameters:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **[*designName*]** – batch solve all setups for the specified design in the project file.
- **[*designName*]:Nominal** – batch solve all nominal setups for the specified design in the project file.
- **[*designName*]:Optimetrics** – batch solve all Optimetrics setups for the specified design in the project file.
- **[*designName*]:[*Nominal/Optimetrics*]:[*SetupName*]** – batch solve the specified Nominal or Optimetrics setup in the specified design.

If you wish to specify whether **-BatchSolve** setups are completed locally or remotely, you can use the following options:

- **-Local** – performs the **-batchsolve** on the local machine.
- **-Remote -machineList** – performs the **-batchsolve** on a remote machine. The <*machineList*> should provide a single hostname.
- **-Distributed -machineList** – performs a distributed **-batchsolve** using a specified machine list.

  The **-machineList** parameter for a **-Distributed** setup can be formatted three ways:

  - **-MachineList list= "<*machine1*>, <*machine2*>, ..."** – machine names (either by IP address or hostname) are separated by commas. If the list contains any whitespace, it must be enclosed in quotation marks. The number of distributed COM engines run on each host is equal to the number of times the hostname appears in the list. That is, if host1 appears in the list once, and host2 appears twice, then one COM engine will run on host1 and two COM engines will run on host2.

    list= accepts the following additional modifiers:

    <*MachineName***>:<***TasksOnMachine***>:<***CoresOnMachine***>:<***RAM Limit in %***>:<***GPUsOnMachine***>**

    Duplicate machine names are not permitted. The integer for <*CoresOnMachine*> must be greater than the integer for <*TasksOnMachine*>. RAM% must be of the form "<num>%" where num is an integer between 1 and 99. Please be sure to leave sufficient RAM for the operating system and applications. If -auto is specified with a machine list, the number of tasks for each machine must be -1.

    Example:

    ```
    list="Orion:4:8:90%:1, Aries:3:12, Pluto:6:12"
    ```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Duplicate machines are not allowed when specifying these additional modifiers. The number of cores must be greater than the number of tasks.

- **-MachineList file= "<***machineListFilepath***>"** – machine names (either by IP address or hostname) are listed in a file (one per line), and you specify the filepath. The number of distributed COM engines run on each host is equal to the number of times the hostname appears in the file. That is, if host1 appears in the file once, and host2 appears twice, then one COM engine will run on host1 and two COM engines will run on host2.

  file= accepts the following additional modifiers, in the file itself:

  <*MachineName***>:<***TasksOnMachine***>:<***CoresOnMachine***>:<***RAM Limit in %***>:<***GPUsOnMachine***>**

  Duplicate machine names are not permitted. The integer for <*CoresOnMachine*> must be greater than the integer for <*TasksOnMachine*>. RAM% must be of the form "<num>%" where num is an integer between 1 and 99. Please be sure to leave sufficient RAM for the operating system and applications. If -auto is specified with a machine list, the number of tasks for each machine must be -1.

  Example:

  ```
  "Orion:4:8:90%:1",
  "Aries:3:12",
  "Pluto:6:12",
  ```

  > **Note:**
  >
  > Duplicate machines are not allowed when specifying these additional modifiers. The number of cores must be greater than the number of tasks.

- **-MachineList num= "<***numberofDistributedEngines***>"** – This format is used when a scheduler (such as LSF, PBS, SGE or HPC) is used to manage the jobs sent to a cluster of hosts. In a scheduler environment, you can specify the number tasks for distributed processing. In this case, you do not specify the machine names after the flag because the names are provided by the scheduler. For example, in the Windows HPC environment, you can write the number of tasks as follows:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
-MachineList num=4
```

Distributed setups can also take the following optional arguments. When these are not present, the behavior defaults to single-level distributed solutions with no change in order of precedence among possible distribution types.

The arguments are:

- **includeTypes= <***default***>|<***distribution type 1, distribution type 2***, ...>** – If included distribution types are specified, only the listed distribution types are enabled. If default is specified, the default set of enabled distribution types is used. To see valid distribution types for your design, click **Simulation** > **Analysis Config** to open the **Analysis Configuration** window and view the types on the **Job Distribution** tab.

  If the list contains any whitespace, it must be enclosed in quotation marks. For example:
  `"includeTypes=Frequencies,Mesh Assembly"`
- **excludeTypes= <***default***>|<***distribution type 1, distribution type 2***, ...>** – If excluded distributed types are specified, all distribution types except those listed will be enabled. If default is specified, the default set of enabled distribution types is used. To see a valid distribution types for your design, click **Simulation** > **Analysis Config** to open the **Analysis Configuration** window and view the types on the **Job Distribution** tab.

  If the list contains any whitespace, it must be enclosed in quotation marks. For example:
  `"excludeTypes=Frequencies,Mesh Assembly"`
- **maxLevels= <**1 | 2**>** – the maximum number of levels of job distribution (the current maximum is 2). See: Selecting Optimal Configurations for Distributed Analysis.
- **numLevel1=** – when two-level distribution is selected (**maxLevels=2**), this specifies the number of level 1 tasks.

-Auto [**NumDistributedVariations=<***num***>**]

This flag enables automatic HPC settings and must be used with one of the following options:

- **-machinelist list=**<*machine list*>, with tasks for each machine set to -1
- **-machinelist numcores=**<*num*>, under a scheduler

All design types being solved must support **-auto** or the solve will be aborted.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The NumDistributedVariations option can be used to specify the number of optimetrics variations to solve simultaneously. The default is to solve optimetrics variations sequentially.

Arguments with **-auto** in a scheduler environment:

- **numcores=**<*total number of cores*>

  Total number of cores, and can be used only with -auto and in a scheduler environment.

- **file=**"<*tmachine list file path*>"

  The specified file can contain line delimited machine specifiers as described above.

- **num=**<*tnum distributed tasks*>

  This is the total number of tasks and used only in a scheduler environment.

- **numgpus=**<*tnumber of GPUs to use*>

  This is the total number of GPUs and used only in a scheduler environment. numgpus must be combined with either **num=** or **numcores=**.

  You can also specify how a **-BatchSolve** distributes Optimetrics variations:

- **-auto** – Without additional parameters, the batch log file will specify that Optimetrics variations be solved sequentially. If -auto is specified with a machine list, the number of tasks for each machine must be -1.
- **-auto NumDistributedVariations=<***num***>** – You can specify an integer value greater than 1. This is the number of variations that will be solved in parallel.

- **-BatchExtract <***BatchExtract script file name***> <***project file name***>** – allows the following commands to be executed non-graphically via script and without checking out any GUI licenses:
  - ExportProfile
  - ExportConvergence
  - ExportMeshStats
  - ExportNetworkData
  - ExportNMFData
  - ExportEigenmodes
  - ExportTransientData
  - Update Reports
  - ExportToFile

  A project file *must* be specified when **-BatchExtract** is used. Commands in the script file will only be executed in the specified project.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Important:**
>
> - **-ng** must be used with **-BatchExtract**, or it will fail.
> - Only the scripts listed above are supported for **-BatchExtract**. Including unsupported script commands will terminate script execution.

- **-RunScript <***script file name***>** – runs the specified script. You can use the **-ScriptArgs** option to add one or more arguments to this command, and the **-Iconic** option.
- **-RunScriptAndExit <***script file name***>** – runs the specified script and exits Electronics Desktop. You can use the **-ScriptArgs** option to add one or more arguments to this command. You can also use **-Iconic**, **-Logfile**, and **-WaitForLicense**.

> **Note:**
>
> **-BatchSolve <***DesignName***>** is mutually exclusive with **-RunScriptAndExit <***ScriptName***>**.

- **-monitor** – during non-graphical analysis, you can monitor progress and messages. Progress, warning and info messages are logged to the standard output stream. Error and fatal messages are logged to the standard error stream. Schedulers intercept these streams and provide commands for display of this output. See individual scheduler documentation for specifics.
- **-grpcsrv** – For external cpython scripting support. If not specified with **ansysedt.exe**, the Start GRPV server listens on a port in the default range 50051:51051. If specified, Start GRPV server listening on the the specified port number. An error is issued if the port is already used. For example:

```
ansysedt.exe // Start GRPV server listening on a port in
the default range 50051:51051.
ansysedt.exe -grpcsrv portnumber // Start GRPV server
listening on the the port number. error if the port
already used.
ansysedt.exe -grpcsrv 50051:50150 // Start GRPV server
listening on a port range 50051:50150.
ansysedt.exe -grpcsrv 50051:100 // Start GRPV server
listening on a port range 50051:50151.
```

Job Management from the Command Line

- **-showmonitorjob** – Launch the monitor job dialog.
- **-showsubmitjob** – Launch the submit job dialog.
- **-showselectscheduler** – Launch the select scheduler dialog.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Run Command Examples

A distributed **-BatchSolve** of a specified design's Optimetrics setups, with a specified machine list:

```
C:\Program Files\ANSYS Inc\v252\AnsysEM\ansysedt -distributed -
machinelist list="255.255.1.1,255.255.1.2" -batchsolve
myDesign:Optimetrics "C:\myProject.aedt"
```

A **-BatchExtract** operation using paths to a script file and a project file:

> **Note:** The example is for an HFSS project. The path, project and design names will vary depending on what the user specifies.

```
ansysedt -ng -batchextract exportToFile.py "C:\Program Files\ANSYS
Inc\v252\AnsysEM\Examples\ElectronicsDesktop\HFSS\RF
Microwave\OptimTee.aedt"
```

Where exportToFile.py contains:

```
oDesktop.RestoreWindow()
oProject = oDesktop.SetActiveProject("OptimTee")
oDesign = oProject.SetActiveDesign("HFSSDesign1")
oModule = oDesign.GetModule("ReportSetup")
oModule.UpdateReports(["XY Plot 1"])
oModule.ExportToFile("XY Plot 1", "exportToFilePy.csv")
```

A **-BatchSolve** of a specified design's nominal setups, run in a minimized window, with a specified log file:

```
ansysedt -Iconic -LogFile "H:\Logs\mylog.log" -BatchSolve
myDesign:Nominal "H:\Projects\MyProject.aedt"
```

Specifying Project Files

Specifying a project file opens that project when Electronics Desktop launches. If **-BatchSolve** is set, the project will also be solved.

You can specify an archive file instead of a project file. If **-Batchsolve** is set, the project will be automatically restored and solved. Otherwise, you are prompted for a restore location, and the project will be restored and opened.

When a **-Batchsolve** is being performed on an archive file, you may also specify **-archiveoptions**:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **overwritefiles** – allows non-project/results-extracted files to overwrite existing files.
- **path= <***projectFilepath***>** – extracts the project file and associated files to the specified path. If not specified, the archive will be extracted into the same directory as the archive file.
- **repackageresults** – Add batchsolve results back to archive file.
- **winpath= <***windowsProjectFilepath***>** – specifies the Windows-specific path to the extracted project file. This is used when a batch job is to be run on a Linux system, but monitored on Windows.

Options

The following options can be associated with one or more of the run commands:

- **-autoextract** – exports profile (as text), convergence (as text), and report data (as CSV) for the requested project/design/setup in a batch job. Once the solve is complete, an export directory is created (for example, "Project1.aedtexport" for a project named "Project1.aedt") that contains a subrectory for each design name. You can also specify `-autoextract "reports, fieldplots"` to also generate *.aedtplt files for each field plot and possible *.avz file (for import and display in Ensight) for all valid field plots. Export files reside within each design-name directory, and include setup name, design variation, job ID, and problem type, as applicable.

  > **Note:**
  >
  > - The -autoextract option is only valid when used with -BatchSolve
  > - The -autoextract option is automatically added for all Ansys Cloud Burst jobs submitted from Electronics Desktop. There is also an additional "reports" and/or "fieldplots" option that immediately follow "-autoextract". This causes all reports to be exported as CSV files at the end of the batch solve, after the profile and convergence have been exported.
  > - For example, you can specify `-autoextract "reports, fieldplots"` to also generate *.aedtplt files for each field plot and possible *.avz file for all valid field plots.

- **-batchoptions** – for batch jobs, specifies any of the options in **Tools** > **Options**. See additional information.
- **-batchoptionhelp** – opens a window showing -batchoptions help. The paths shown in this window can be used with batchoptions and the Update Registry Get and Set commands. See: Setting or Removing Option Values in Configuration Files: UpdateRegistry Command.
- **-distribute** – distributes a batch solve to multiple machines. This option must be combined with the **-BatchSolve** run command. See: Distributed Analysis.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **-help** – opens a window displaying command line options. This can only be used without a run command.
- **-iconic** – runs Electronics Desktop with the window iconified (minimized).
- **-logfile <***filePath***>** – specifies a log file. If none is specified, <project_name>.log will be written to the <project_name>.batchinfo directory.
- **-monitor** – enables batch job output to standard output and standard error streams.
- **-ng** – runs Electronics Desktop in non-graphical mode. This *must* be used with the **-BatchExtract** command. For the ansysedtng beta option, see: Running Ansys Electronics Desktop from a Command Line (Nongraphical) Beta.
- **-waitforlicense** – directs Electronics Desktop to wait for unavailable licenses.
- **-scriptargs** – used in conjunction with **-RunScript** or **-RunScriptAndExit**, adds arguments to the specified script. You can pass multiple arguments to **-scriptargs** by surrounding the arguments in quotation marks. For example:

  ```
  ansysedt.exe -scriptargs "Design1 Setup1" -RunScriptAndExit
  C:\temp\test.py
  ```

  In Python, the command line parameter following **-scriptargs** is passed without modification as a single string in the ScriptArgument python variable.

  In either case, **Design1** is taken into Electronics Desktop as the first argument, and **Setup1** as the second argument. If you failed to use quotation marks, **Design1** would be taken as the first argument and **Setup1** would not be understood by Electronics Desktop.

-Batchoptions

All options that are specified through **Tools** > **Options** go into the user-level registry.

> **Note:**
>
> - Options are arranged as keys and values (in a structure similar to the *Windows Registry*). However, these options are not a part of the *Windows Registry* but are separately stored and maintained by the Ansys Electronics Desktop software.
> - For access to options and functionality beyond what is directly accessible via the user interface or batch options, refer to the documentation of the **UpdateRegistry** tool. This tool is discussed in the following help topic and in the topics that follow it in the same branch of the product help:
>
>   Setting or Removing Option Values in Configuration Files: UpdateRegistry Command

You can override the option registry entries via the **-batchoptions** command line. These overrides apply only to the current Desktop session. The registry setting overrides may be specified on the command line, or may be in a file with the file pathname specified on the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

command line. Batch jobs can be submitted from the command line or through Electronics Desktop's job submission window.

Large Scale DSO offers two new batchoptions related to the redistribution ability.

- LargeScaleDSO/VarRedistribution, where 0 disables redistribution (default), and 1 enables it.
- LargeScaleDSO/RedistributionLimit, is a positive integer specifying the minimum estimated remaining time (in minutes) for variations to redistribute to another task. The default is 3.

> **Note:**
>
> **-batchoptions** is only valid for batch jobs. It is ignored if you have not specified **-BatchSolve**, **-BatchSave**, or **-BatchExtract**.

-Batchoptions Examples

The batchoption **CreateStartingMesh** is available for the following products:

HFSS, HFSS-3DLayout, Icepak, Q3D, Q2D, Maxwell3D, Maxwell2D, and Mechanical.

When this option is set, only the initial mesh and manual mesh operations are completed for the batch solution.

> **Note:**
>
> No adaptive meshing occurs for any Icepak or Mechanical solutions. Therefore, after solving, the final mesh will be identical to the starting mesh for these two design types.

This example enables CreateStartingMesh for HFSS, and runs a batch solution of the specified project:

```
ansysedt -batchoptions "'HFSS/CreateStartingMesh'=1" -batchsolve
"D:\projects\MyProject.aedt"
```

See: Additional Examples of -Batchoptions Use.

Export Options Files

The **Tools** > **Options** > **Export Options Files** command writes XML files containing the options settings at all levels to the specified directory. This feature is intended to make it easier for different users to use Ansys Electromagnetics Suite 2025 R2 installed on shared directories or network drives. See: Example Uses for Export Options Features.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Examples and Further Explanations of -batchoptions Use

This section provides further examples and explanations of -batchoptions.

- Example with registry settings specified on the command line
- Example with registry settings specified in a file
- When to Use the -batchoptions Desktop Command Line Option

The following examples use general Desktop and HFSS-specific settings. This feature is available for all desktop products.

- The registry path separator is the slash (/) character.
- Each complete registry key (that is, a registry path and option name) is enclosed in single quotes.
- Registry string values are enclosed in single quotes.
- After the -batchoptions switch, the set of registry keys and values that follows it must be enclosed in double quotes. However, if a batchoptions file is referenced (instead of listing the options directly on the command line), the double quotes are not used around the filename.
- Backslashes in registry key string values must be escaped with another backslash (\\ ), since a backslash by itself is an escape code within strings.

Example with registry settings specified on the command line

```
ansysedt.exe -batchsolve -batchoptions
"'Desktop/Settings/ProjectOptions/NumberOfProcessors'=4
'HFSS/NumCoresPerDistributedTask'=2   (* This option is not
currently available from the Add Batchoption dialog box.)
'Desktop/ProjectDirectory'='C:\\projects\test'" projectname.aedt
```

This command line overrides registry values of the NumberOfProcessors (Desktop/Settings/ProjectOptions key) and ProjectDirectory (Desktop key) options.

> **Note:**
>
> - Multiple registry settings may appear in a single -batchoptions value, separated by whitespace.
> - The -batchoptions value must be enclosed in double quotes if it contains any whitespace.

Example with registry settings specified in a file

```
ansysedt.exe -batchsolve -batchoptions <filename>
projectname.aedt
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where the referenced file, <*filename*>, contains:

```
$begin 'Config'
  'Desktop/Settings/ProjectOptions/NumberOfProcessors'=4
  'HFSS/NumCoresPerDistributedTask'=2
  'Desktop/ProjectDirectory'='C:/projects/test'
$end 'Config'
```

This command overrides the registry values of the NumberOfProcessors (Desktop/Settings/ProjectOptions key), NumCoresPerDistributedTask (HFSS key), and ProjectDirectory (Desktop key) options. These overrides apply only to the current Electronics Desktop session.

> **Note:**
>
> - The -batchoptions ***<filename>*** value must be enclosed in double quotes if it contains whitespace
> - The **$begin 'Config'** and **$end 'Config'** lines are required

For additional options you can override from the command line with **-batchoptions**, see:

- For -batchoptions Use: Project Directory and Lib Paths
- For -batchoptions Use: TempDirectory
- For -batchoptions Use: Various Desktop Settings

When to Use the -batchoptions Desktop Command Line Option

You can set analysis parameters for batch mode jobs using the graphical user interface (GUI). For example, you can set all options for a batch job using the **Add Batch Option** dialog box, which is accessed through the **Submit Job To** dialog box. These parameter settings include the following solver options (several examples, not a complete list):

- HPCLicenseType
- tempdirectory
- Desktop/AutoExtractReports
- Desktop/Settings/ProjectOptions/MumberOfProcessors
- <*design_type*>/DefaultProcessPriority
- <*design_type*>/EnableGPU and/or EnableGPUForEye – Applicable to Circuit Design, EMIT, HFSS, HFSS 3DLayout, and Maxwell 3D (also EnableGPUForSBR for HFSS only)
- <*design_type*>/MPIVendor (either “Intel” or “Microsoft”) – Applicable to HFSS, HFSS 3DLayout, Icepak, Maxwell 2D, Maxwell 3D), and Q3D

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For graphical analyses that do **not** use batch mode, you specify the analysis parameters using the GUI (**Tools** > **Options** > **General Options** or **HPC and Analysis Options**). These settings are written to the registry when you exit the Electronics Desktop program. The settings are read from the registry when the application is started. Therefore, when you start the Electronics Desktop application, all settings retain the values from the previous session of the same user on the same machine. If there was not a previous session of the same user on the same machine, then the values are obtained from other registry configuration files or from a default value.

When running a batch analysis, any setting that is not specified using the **-batchoptions** command line option is taken from the registry. This value is typically the setting from the last session of the same user on the same machine. However, the **-batchoptions** command line option allows you to override the parameter with values specified on the command line or in a batchoptions file. The values specified using the **-batchoptions** command line option only apply to the batch job, and do not affect the parameter values stored in the registry.

If important **-batchoptions** values are not specified when running a batch job, the paramaters could be affected by an interactive session running on the same host by the same user. Parameter changes can occur if the user sets an option in the GUI and then exits the program, or if another process that accesses the registry exits. To be sure of the desired batch job outcome, avoid changing options in concurrent interactive sessions or include the desired -batchoptions in the command line.

For -batchoptions Use: Project Directory and Lib Paths

The PersonalLib, syslib and userlib settings are a little different from other settings. If the final directory name is different from what is expected, then PersonalLib, syslib or userlib is appended as a final directory. In addition, these settings may come from a different registry value if the registry values shown above are not set.

Registry Key	Default Value	Units or Values	Description
Desktop/ProjectDirectory	subdirectory of user's HOME directory or “Documents” directory	Directory pathname	Directory where new projects are created
Desktop/PersonalLib	PersonalLib subdirectory of user's HOME directory or “Documents” directory	Directory pathname	Directory PersonalLib is appended if final directory is not PersonalLib
Desktop/syslib	syslib subdirectory of installation directory	Directory pathname	Directory syslib is appended if final directory is not syslib
Desktop/userlib	userlib subdirectory of installation directory	Directory pathname	Directory userlib is appended if final directory is not userlib

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

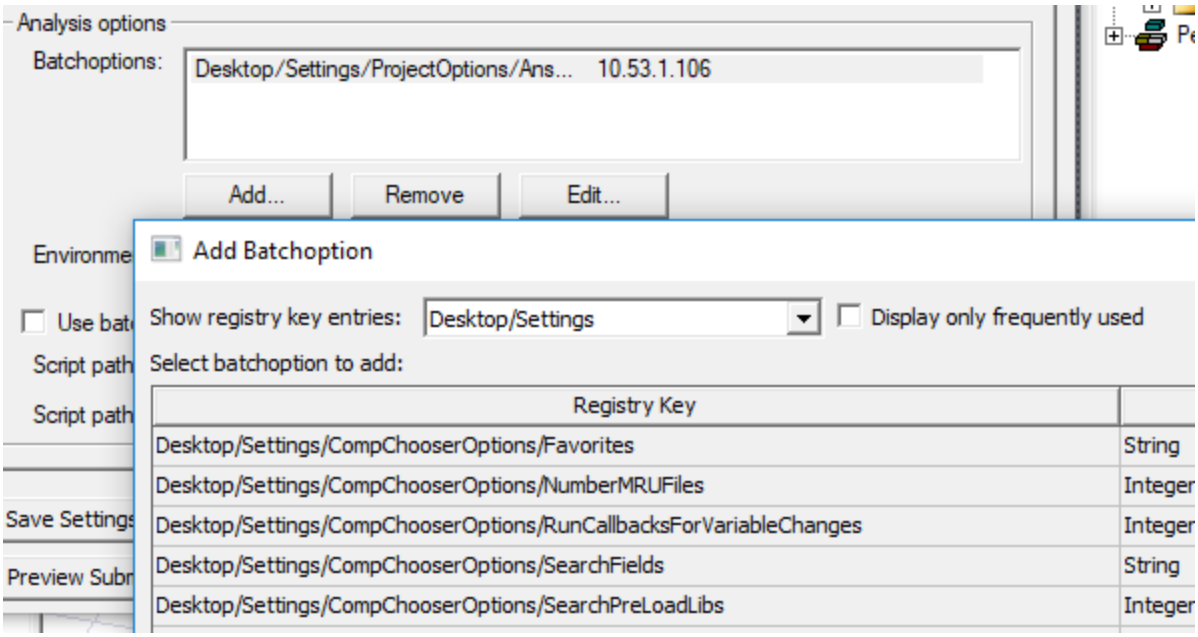
For -batchoptions Use: TempDirectory

Registry Key	Default Value	Units or Values	Description
TempDirectory	Set by installer	-	Directory for temporary files

For -batchoptions Use: Various Desktop Settings

Note that most of these options only affect the GUI. To view these options from the *Submit Job To* window:

1. In the *Analysis options* area of the *Submit Job To* window, click **Add...**.
2. In the *Add Batchoption* window that appears, select **Desktop/Settings** from the **Show registry key entries** drop-down menu.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Registry Key	Default Value	Units or Values	Description
Desktop/Settings/ProjectOptions/AnimationMemory	200	Megabytes (MB)	Stop animations when available memory falls below this value
Desktop/Settings/ProjectOptions/AnsoftCOMPreferredIPAddress (see note below table)	"" (empty string)	IP address (as a string)	IP address used to connect from COM engines to ansysedt.exe
Desktop/Settings/ProjectOptions/AnsysEMPreferredSubnetAddress (see note below table)	"" (empty string)	IP address (as a string)	Subnet used to connect COM engines to ansysedt.exe – allowed formats are: • IPv4 network prefix in CIDR notation Example: 123.123.123.0/24 • IPv4 network prefix with /subnet mask appended Example: 123.123.123.0/255.255.255.0
Desktop/Settings/ProjectOptions/AutoSaveInterval	10	edits	Number of edits to allow between autosaves
Desktop/Settings/ProjectOptions/AutoShowMessageWindow	1 (true)	0 (false) or 1 (true)	Show message window on new messages
Desktop/Settings/ProjectOptions/AutoShowProgressWindow	0 (false)	0 (false) or 1 (true)	Show progress window when starting a simulation
Desktop/Settings/ProjectOptions/DiskLimitForAbort	0	Megabytes (MB)	A warning is issued when available disk space falls below this value

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Registry Key	Default Value	Units or Values	Description
Desktop/Settings/ProjectOptions/DoAutoSave	1 (true)	0 (false) or 1 (true)	Enables autosaves if true
Desktop/Settings/ProjectOptions/DrawStateIconsInProjectTree3	1 (true)	0 (false) or 1 (true)	Change icon when selection does not match active window
Desktop/Settings/ProjectOptions/ExpandMessageTreeOnInsert	1 (true)	0 (false) or 1 (true)	Ensure that new messages are visible in the message window tree
Desktop/Settings/ProjectOptions/ExpandOnInsert	0 (false)	0 (false) or 1 (true)	Expand project tree on insert
Desktop/Settings/ProjectOptions/HighlightActiveContextInProjectTree2	1 (true)	0 (false) or 1 (true)	Emphasize active command context (menu and toolbars)
Desktop/Settings/ProjectOptions/SavePreviewImagesInProject File	1 (true)	0 (false) or 1 (true)	Save preview images in project file
Desktop/Settings/ProjectOptions/UpdateReportOnFileOpen	0 (false)	0 (false) or 1 (true)	Update reports on file open

Note:

The preferredIP address and preferred subnet address settings are mutually exclusive. If both are specified to be non-empty strings, then the preferred IP address takes precedence, and the preferred subnet address is ignored. This feature is typically used for cluster environments using batch solves. The setting can be made via batchoptions but can also be done via UpdateRegistry.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For -batchoptions Use: Maxwell 2D and Maxwell 3D Options

Here are the Maxwell Options with the registry keys (that is, the registry paths and option names). You can access a help window with information covering most available design-type-specific batchoptions using the **-batchoptionhelp** switch when launching Electronics Desktop from a command line:

Registry Key	Default Value	Units or Values	Description
Maxwell 2D (or 3D)/DesiredRAMLimitInGB	0 (No Limit)	GB	Preferred memory use in GB. Type: Floating Point, Min: 0, Max: 1000000.
Maxwell 2D (or 3D)/MaxRAMLimitInGB	0 (No Limit)	GB	This setting specifies the absolute maximum memory usage in GB. Type: Floating Point, Min: 0, Max: 1000000.
Maxwell 2D (or 3D)/HPCLicenseType	"pack"	"pool" or "pack"	For Maxwell 2D Designs: "pool" = multiprocessing enabled by HPC licensing; "pack" = multiprocessing enabled by HPC Pack licensing
Maxwell 2D(or 3D)/DefaultProcessPriority	Normal	Allowed values: "Critical", "Above Normal", "Normal", "Below Normal", "Idle"	Specifies the default priority of distributed tasks.
Maxwell 2D(or 3D)/HPCLicenseType	“pack”	“pool” or “pack” or "none"	Specifies if HPC license is used and which type.
Maxwell 2D(or 3D)/MPIVendor		“Microsoft”, “Intel”.	Which MPI libraries to use for remote communication. Type: String,
Maxwell2D(or 3D)/MPIVersion	"Default"	"Default", "2018", "2021"	This setting specifies the MPI Version. If not specified, then the default version for

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Registry Key	Default Value	Units or Values	Description
			the MPI vendor will be used. The valid values depend on the MPI vendor. Allowed values: "Default", "2018", "2021"
Maxwell 2D(or 3D)/RemoteSpawnCommand			This setting specifies the command that will be used to launch distributed solver jobs. The value Scheduler is only valid for jobs running under an LSF or SGE scheduler, and only if the MPI Vendor is Intel. Type: String, Allowed Values: "RSH", "SSH", "Scheduler".
Maxwell 2D(or 3D)/DistribLicense		"DSO", "Solver"	This setting specifies which license feature to use for distributed variations. Type: String
Maxwell 2D(or 3D)/NumCoresPerDistributedTask		Min: 1, Max: 1000	This setting specifies the number of cores that will be used per distributed task when using a machine list provided by the scheduler. Type, Integer.
Maxwell 2D(or 3D/SolveAdaptiveOnly		0 (False), 1 (True).	If true, only the adaptive portion of the setup will be solved. Applies only if a single setup is selected.
Maxwell 2D(or 3D)/CreateStartingMesh		0 (False), 1 (True)	If true, the starting mesh (with mesh operations if applicable) will be created, but none of the setup will be solved. Type: Integer
Maxwell 2D(or 3D)/RAMLimitPercent		Min: 1, Max: 99	This setting specifies the percent of total RAM to be used on each machine for solving. Please be sure to leave sufficient RAM for the operating system and

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Registry Key	Default Value	Units or Values	Description
			applications. Type: Integer
Maxwell 2D(or 3D)/RAMLimitPerCoreInGB		Min: 0, Max: 10000	This setting specifies the maximum amount of RAM used for each core allocated by the scheduler in GB. This cannot be combined with RAM limit percent and is only valid when solving in a scheduler environment. Type: Floating Point
Maxwell 2D(or 3D)TotalNumOfCores		Min: 1, Max: 1000	This setting specifies the total number of cores that will be used across all tasks when using machine list provided by the scheduler. Type: Integer
Maxwell 2D(or 3D)/SolveAdaptiveOnly	1	Allowed values: 0 (false), 1 (true)	If true, only the adaptive portion of the setup will be solved. Applies only if a single setup is selected.
Maxwell 2D(or 3D)/CreateStartingMesh	0	Allowed values: 0 (false), 1 (true)	If true, the starting mesh (with mesh operations if applicable) will be created, but none of the setup will be solved. Applies only if a single setup is selected.
Maxwell 2D/RAMLimitPercent		Min: 1, Max: 99	This setting specifies the percent of total RAM to be used on each machine for solving. Please be sure to leave sufficient RAM for the operating system and applications. Type: Integer
Maxwell 2D(or 3D)/RAMLimitPerCoreInGB		Min: 0, Max: 10000	This setting specifies the maximum amount of RAM used for each core allocated by the scheduler in GB. This cannot be combined with RAM limit

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Registry Key	Default Value	Units or Values	Description
			percent and is only valid when solving in a scheduler environment. Type: Floating Point
Maxwell 2D(or 3D)/DefaultProcessPriority	Normal	"Critical", "Above Normal", "Normal", "Below Normal", "Idle"	Specifies the default priority of distributed tasks.
Maxwell 3D/EnableGPU	0	0 (False), 1 (True).	Allow GPU to be used for AC Magnetic solves. GPU acceleration will be disabled when automatic setting is selected in Analysis Configuration. Type: Integer

Batchoptions: Override Registry Entry

Any registry entry can be overridden via a command-line option.

Example: batchoptions specified on a command line

```
ansysedt -batchsolve -batchoptions
"'Desktop/Preferences/ProjectOptions/NumberOfProcessors'=4 'HFSS 3D
Layout Design/NumCoresPerDistributedTask'=2" projectname.aedt
```

Example: batchoptions specified in a file

```
ansysedt -batchsolve -batchoptions numprocfile projectname.aedt
```

where numprocfile contains

```
$begin 'Config'
  'Desktop/Preferences/ProjectOptions/NumberOfProcessors'=4
  'HFSS 3D Layout/NumCoresPerDistributedTask'=2
$end 'Config'
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Running Ansys Electronics Desktop from a Command Line (Nongraphical) Beta

> **Note:**
>
> Running Electronics Desktop using the ansysedtng command is currently a beta feature. For information on enabling beta options, see General Options: Desktop Configuration.

Ansys Electronics Desktop includes line arguments that can be included when launching from a command line or terminal prompt. All command-line arguments are case-insensitive. The commands associated batch options can also be used with a Job Management Interface for submitting jobs to Ansys or RSM and other supported schedulers.

> **Note:**
>
> Linux users whose installations lack a MainWin home directory receive an error when running the ansysedt command. To run Elecronics Desktop from a command line successfully on such Linux machines, use the ansysedtng command and options described in this section.

Command-line Syntax

```
ansysedtng <options> <run command(s)> <project file/script file>
```

It is good practice to put quotation marks around the path to the solver executable, and around the full path to the project. This ensures that spaces in the path or project will not be an issue. The same is true of the design name, if there are spaces.

Run Commands

The following command line run commands are available for Ansys Electronics Desktop. BatchSolve must be specified as arguments after the solver executable. When none is specified, you may specify a project or archive to open when Electronics Desktop launches, and can only use the **-Help** option. The commands are further described below:

- **-BatchExtract <***BatchExtract script file name***> <***project file name***>** – allows the following commands to be executed non-graphically via script and without checking out any GUI licenses:
- ExportProfile
- ExportConvergence
- ExportMeshStats

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- ExportNetworkData
- ExportNMFData
- ExportEigenmodes
- ExportTransientData
- Update Reports
- ExportToFile

- A project file *must* be specified when **-BatchExtract** is used. Commands in the script file will only be executed in the specified project.
-
  > **Important:**
  >
  > - Only the scripts listed above are supported for **-BatchExtract**. Including unsupported script commands will terminate script execution.

- **-BatchSolve <***project file name***>**

  By default, this run command solves all adaptive setups, Optimetrics setups, and sweeps found in the project file. You can run this command with the **-Logfile** and **-WaitForLicense** options. If parallel solve is possible, you can use the **-Distribute** option.

  If you wish to specify which setups **-BatchSolve** completes, you can use additional parameters:

  - **[*designName*]** – batch solve all setups for the specified design in the project file.
  - **[*designName*]:Nominal** – batch solve all nominal setups for the specified design in the project file.
  - **[*designName*]:Optimetrics** – batch solve all Optimetrics setups for the specified design in the project file.
  - **[*designName*]:[*Nominal/Optimetrics*]:[*SetupName*]** – batch solve the specified Nominal or Optimetrics setup in the specified design.

  If you wish to specify whether **-BatchSolve** setups are completed locally or remotely, you can use the following options:

  - **-Local** – performs the **-batchsolve** on the local machine.
  - **-Remote -machineList** – performs the **-batchsolve** on a remote machine. The <*machineList*> should provide a single hostname.
  - **-Distributed -machineList** – performs a distributed **-batchsolve** using a specified machine list.

    The **-machineList** parameter for a **-Distributed** setup can be formatted three ways:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **-MachineList list= "<*machine1*>, <*machine2*>, ..."** – machine names (either by IP address or hostname) are separated by commas. If the list contains any whitespace, it must be enclosed in quotation marks. The number of distributed COM engines run on each host is equal to the number of times the hostname appears in the list. That is, if host1 appears in the list once, and host2 appears twice, then one COM engine will run on host1 and two COM engines will run on host2.

  list= accepts the following additional modifiers:

  <*MachineName***>:<***TasksOnMachine***>:<***CoresOnMachine***>:<***RAM Limit in %***>:<***GPUsOnMachine***>**

  Duplicate machine names are not permitted. The integer for <*CoresOnMachine*> must be greater than the integer for <*TasksOnMachine*>. RAM% must be of the form "<num>%" where num is an integer between 1 and 99. Please be sure to leave sufficient RAM for the operating system and applications. If -auto is specified with a machine list, the number of tasks for each machine must be -1.

  Example:

  ```
  list="Orion:4:8:90%:1, Aries:3:12, Pluto:6:12"
  ```

  > **Note:**
  >
  > Duplicate machines are not allowed when specifying these additional modifiers. The number of cores must be greater than the number of tasks.

- **-MachineList file= "<*machineListFilepath*>"** – machine names (either by IP address or hostname) are listed in a file (one per line), and you specify the filepath. The number of distributed COM engines run on each host is equal to the number of times the hostname appears in the file. That is, if host1 appears in the file once, and host2 appears twice, then one COM engine will run on host1 and two COM engines will run on host2.

  file= accepts the following additional modifiers, in the file itself:

  <*MachineName***>:<***TasksOnMachine***>:<***CoresOnMachine***>:<***RAM Limit in %***>:<***GPUsOnMachine***>**

  Duplicate machine names are not permitted. The integer for <*CoresOnMachine*> must be greater than the integer for <*TasksOnMachine*>. RAM% must be of the form "<num>%" where num is an

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

integer between 1 and 99. Please be sure to leave sufficient RAM for the operating system and applications. If -auto is specified with a machine list, the number of tasks for each machine must be -1.

Example:

```
"Orion:4:8:90%:1",
"Aries:3:12",
"Pluto:6:12",
```

> **Note:**
>
> Duplicate machines are not allowed when specifying these additional modifiers. The number of cores must be greater than the number of tasks.

- **-MachineList num= "<***numberofDistributedEngines***>"** – This format is used when a scheduler (such as LSF, PBS, SGE or HPC) is used to manage the jobs sent to a cluster of hosts. In a scheduler environment, you can specify the number tasks for distributed processing. In this case, you do not specify the machine names after the flag because the names are provided by the scheduler. For example, in the Windows HPC environment, you can write the number of tasks as follows:

  ```
  -MachineList num=4
  ```

Distributed setups can also take the following optional arguments. When these are not present, the behavior defaults to single-level distributed solutions with no change in order of precedence among possible distribution types.

The arguments are:

- **includeTypes= <***default***>|<***distribution type 1, distribution type 2***, ...>** – If included distribution types are specified, only the listed distribution types are enabled. If default is specified, the default set of enabled distribution types is used. To see valid distribution types for your design, click **Simulation** > **Analysis Config** to open the **Analysis Configuration** window and view the types on the **Job Distribution** tab.

  If the list contains any whitespace, it must be enclosed in quotation marks. For example:

  ```
  "includeTypes=Frequencies,Mesh Assembly"
  ```

- **excludeTypes= <***default***>|<***distribution type 1, distribution type 2***, ...>** – If excluded distributed types are specified, all distribution types except those listed will be enabled. If default is specified, the default set of enabled

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

distribution types is used. To see a valid distribution types for your design, click **Simulation** > **Analysis Config** to open the **Analysis Configuration** window and view the types on the **Job Distribution** tab.

If the list contains any whitespace, it must be enclosed in quotation marks. For example:

```
"excludeTypes=Frequencies,Mesh Assembly"
```

- **maxLevels= <**1 | 2**>** – the maximum number of levels of job distribution (the current maximum is 2). See: Selecting Optimal Configurations for Distributed Analysis.
- **numLevel1=** – when two-level distribution is selected (**maxLevels=2**), this specifies the number of level 1 tasks.

-Auto [**NumDistributedVariations=<***num***>**]

This flag enables automatic HPC settings and must be used with one of the following options:

- **-machinelist list=**<*machine list*>, with tasks for each machine set to -1
- **-machinelist numcores=**<*num*>, under a scheduler

All design types being solved must support **-auto** or the solve will be aborted.

The NumDistributedVariations option can be used to specify the number of optimetrics variations to solve simultaneously. The default is to solve optimetrics variations sequentially.

Arguments with **-auto** in a scheduler environment:

- **numcores=**<*total number of cores*>

  Total number of cores, and can be used only with -auto and in a scheduler environment.

- **file=**"<*tmachine list file path*>"

  The specified file can contain line delimited machine specifiers as described above.

- **num=**<*tnum distributed tasks*>

  This is the total number of tasks and used only in a scheduler environment.

- **numgpus=**<*tnumber of GPUs to use*>

  This is the total number of GPUs and used only in a scheduler environment. numgpus must be combined with either **num=** or **numcores=**.

  You can also specify how a **-BatchSolve** distributes Optimetrics variations:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

    - **-auto** – Without additional parameters, the batch log file will specify that Optimetrics variations be solved sequentially. If -auto is specified with a machine list, the number of tasks for each machine must be -1.
    - **-auto NumDistributedVariations=<*num*>** – You can specify an integer value greater than 1. This is the number of variations that will be solved in parallel.
- **-monitor** – during non-graphical analysis, you can monitor progress and messages. Progress, warning and info messages are logged to the standard output stream. Error and fatal messages are logged to the standard error stream. Schedulers intercept these streams and provide commands for display of this output. See individual scheduler documentation for specifics.

Run Command Examples

A distributed **-BatchSolve** of a specified design's Optimetrics setups, with a specified machine list:

```
C:\Program Files\ANSYS Inc\v252\AnsysEM\ansysedtng -distributed -
machinelist list="255.255.1.1,255.255.1.2" -batchsolve
myDesign:Optimetrics "C:\myProject.aedt"
```

A **-BatchSolve** of a specified design's nominal setups, run in a minimized window, with a specified log file:

```
ansysedtng -LogFile "H:\Logs\mylog.log" -BatchSolve
myDesign:Nominal "H:\Projects\MyProject.aedt"
```

Specifying Project Files

Specifying a project file opens that project when Electronics Desktop launches. If **-BatchSolve** is set, the project will also be solved.

You can specify an archive file instead of a project file. If **-Batchsolve** is set, the project will be automatically restored and solved. Otherwise, you are prompted for a restore location, and the project will be restored and opened.

When a **-Batchsolve** is being performed on an archive file, you may also specify **-archiveoptions**:

- **overwritefiles** – allows non-project/results-extracted files to overwrite existing files.
- **path= <*projectFilepath*>** – extracts the project file and associated files to the specified path. If not specified, the archive will be extracted into the same directory as the archive file.
- **repackageresults** – Add batchsolve results back to archive file.
- **winpath= <*windowsProjectFilepath*>** – specifies the Windows-specific path to the extracted project file. This is used when a batch job is to be run on a Linux system, but monitored on Windows.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Options

The following options can be associated with one or more of the run commands:

- **-autoextract** – exports profile (as text), convergence (as text), and report data (as CSV) for the requested project/design/setup in a batch job. Once the solve is complete, an export directory is created (for example, "Project1.aedtexport" for a project named "Project1.aedt") that contains a sub-directory for each design name. You can also specify `-autoextract "reports, fieldplots"` to also generate *.aedtplt files for each field plot and possible *.avz file (for import and display in Ensight) for all valid field plots. Export files reside within each design-name directory, and include setup name, design variation, job ID, and problem type, as applicable.

  > **Note:**
  >
  > - The -autoextract option is only valid when used with -BatchSolve
  > - The -autoextract option is automatically added for all Ansys Cloud Burst Computing jobs submitted from Electronics Desktop. There is also an additional "reports" and/or "fieldplots" option that immediately follow "-autoextract". This causes all reports to be exported as CSV files at the end of the batch solve, after the profile and convergence have been exported.
  > - For example, you can specify `-autoextract "reports, fieldplots"` to also generate *.aedtplt files for each field plot and possible *.avz file for all valid field plots.

- **-batchoptions** – for batch jobs, specifies any of the options in **Tools** > **Options**. See additional information.
- **-distribute** – distributes a batch solve to multiple machines. This option must be combined with the **-BatchSolve** run command. See: Distributed Analysis.
- **-help** – opens a window displaying command line options. This can only be used without a run command.
- **-logfile <*filePath*>** – specifies a log file. If none is specified, <project_name>.log will be written to the <project_name>.batchinfo directory.
- **-monitor** – enables batch job output to standard output and standard error streams.
- **-ng** – runs Electronics Desktop in non-graphical mode.
- **-waitforlicense** – directs Electronics Desktop to wait for unavailable licenses.

-Batchoptions

All options that are specified through **Tools** > **Options** go into the user-level registry.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- Options are arranged as keys and values (in a structure similar to the *Windows Registry*). However, these options are not a part of the *Windows Registry* but are separately stored and maintained by the Ansys Electronics Desktop software.
- For access to options and functionality beyond what is directly accessible via the user interface or batch options, refer to the documentation of the **UpdateRegistry** tool. This tool is discussed in the following help topic and in the topics that follow it in the same branch of the product help:

  Setting or Removing Option Values in Configuration Files: UpdateRegistry Command

You can override the option registry entries via the **-batchoptions** command line. These overrides apply only to the current Desktop session. The registry setting overrides may be specified on the command line, or may be in a file with the file pathname specified on the command line. Batch jobs can be submitted from the command line or through Electronics Desktop's job submission window.

Note:

-batchoptions is only valid for batch jobs. It is ignored if you have not specified **-BatchSolve**.

-Batchoptions Examples

The batchoption **CreateStartingMesh** is available for the following products:

HFSS, Icepak, Q3D, Q2D, Maxwell3D, Maxwell2D, and Mechanical.

When this option is set, only the initial mesh and manual mesh operations are completed for the batch solution.

Note:

No adaptive meshing occurs for any Icepak or Mechanical solutions. Therefore, after solving, the final mesh will be identical to the starting mesh for these two design types.

This example enables CreateStartingMesh for HFSS, and runs a batch solution of the specified project:

```
ansysedtng -batchoptions "'HFSS/CreateStartingMesh'=1" -batchsolve
"D:\projects\MyProject.aedt"
```

See: Additional Examples of -Batchoptions Use.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Export Options Files

The **Tools** > **Options** > **Export Options Files** command writes XML files containing the options settings at all levels to the specified directory. This feature is intended to make it easier for different users to use Ansys Electromagnetics Suite 2025 R2 installed on shared directories or network drives. See: Example Uses for Export Options Features.

Running Ansys Electronics Desktop with a JSON File

Ansys Electronics Desktop includes line arguments that can be included in a JSON file and executed by entering the -json command line option. The -json option takes a JSON file as argument, and runs AEDT with the options specified within the file. All options in the -help text is supported. Options in the JSON file can be used together with options specified in the command line. If the same option is specified both in the command line and the JSON file, the option in the command line overrides the option in the JSON file.

> **Note:**
>
> The -json option functions with both the ansysedt and ansysedtng commands.

Enter the command as follows to execute the JSON file

```
ansysedt -json <filepath to the JSON file>
```

JSON File Format

Take the following into consideration when creating a JSON file.

- The JSON file must contain an object. Each option is a key value pair where the key is the option name (no dash) and the value is the argument.
- For options that do not take any arguments, such as -monitor, the value must be a bool - true or false to turn on or off the option, respectively.
- For options that take optional arguments, such as -logfile, the value can either be a bool or some other type (in this case, a string specifying the log file) depending on the option.
- Project/archive file and folder should be specified directly on the command line when using -json.
- See the example below for a JSON file containing all options in the -help text.

> **Note:**
>
> The example does not contain a valid combination of options but is solely used to show what type of argument each option takes.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
{
    "archiveoptions":
    {
        "overwritefiles": true,
        "path": "\\\\dataserver\\home\\user\\OptimTee.aedt",
        "repackageresults": true,
        "winpath": "\\\\dataserver\\home\\user\\OptimTee.aedt"
    },
    "auto":
    {
        "numDistributedVariations": 2
    },
    "autoextract": "reports, fieldplots",
    "batchextract": "\\\\dataserver\\home\\user\\batchextract_
script.py",
    "batchoptionhelp": false,
    "batchoptions":
    {
        "HFSS/CreateStartingMesh": 1,
        "Desktop/ProjectDirectory": "\\\\dataserver\\home\\user"
    },
    "batchsave":
    {
        "autoheal": false,
        "cleanupmodel": true,
        "forcearchive": true,
        "norecurse": true,
        "outputfolder": "\\\\dataserver\\home\\user\\batchsave_
folder",
        "preservehistory": false,
        "previewonly": false
    },
    "batchsolve":
    {
        "designName": "TeeModel",
        "setupType": "Nominal",
        "setupName": "OptimTee_Analysis"
    },
    "distributed":
    {
        "excludeTypes":
        [
            "Frequencies",
            "Mesh Assembly"
        ],
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
        "includeTypes": [],
        "maxLevels": 2,
        "numLevel1": 2
    },
    "help": false,
    "iconic": false,
    "local": false,
    "logfile": "\\\\dataserver\\home\\user\\logfile.log",
    "machinelist":
    {
        "list":
        [
            {
                "machine": "sjohpcw2k16fe",
                "numTasks": -1,
                "numCores": 1,
                "RAMPercent": 90,
                "numGPUs": 1
            },
            {
                "machine": "dcuww10core01",
                "RAMPercent": 80
            }
        ]
    },
    "monitor": true,
    "ng": true,
    "runscript": "\\\\dataserver\\home\\user\\script.py",
    "runscriptandexit": "\\\\dataserver\\home\\user\\script.py",
    "scriptargs": "Design1 Setup1",
    "showmonitorjob": false,
    "showselectscheduler": false,
    "showsubmitjob": false,
    "waitforlicense": false
}
```


Running Ansys Electronics Desktop from a Windows Remote Terminal

When running Maxwell from a remote terminal, there are some performance and behavior issues to consider. These issues are due to the interaction of bandwidth/opengl drivers/remote-terminal-protocol

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Showing axes when interactively drawing objects will slow the performance.
- Remote OpenGL performance will be slower in general. Graphics card and driver quality helps.
- All 3D windows will be closed when you switch from remote PC to a console or from a console to remote. This is to avoid display/opengl instability during the switch.
- Grid will not be turned off while viewing a plot from a remote desktop. The mouse over highlights on 2D plots may appear as not totally overlapping the line color or as thin dotted lines.

Using Windows HPC Commands

HPC Integration allows you to submit jobs directly using Ansys Electromagnetics command line arguments for batch solves. Ansys Electromagnetics products must be accessible from the same directory on all machines. The Ansys Electromagnetics command line syntax is documented here. You must pass in a -distributed flag as part of the Ansys Electromagnetics command line arguments if you want to run a distributed simulation.

For information concerning the supported HPC software, go the the Platform Support webpage and click the links to the following topics:

- *Ansys 2025 R2 - Job Schedulers and Queuing Systems Support*
- *Ansys 2025 R2 - Message Passing Interface Support for Parallel Computing*

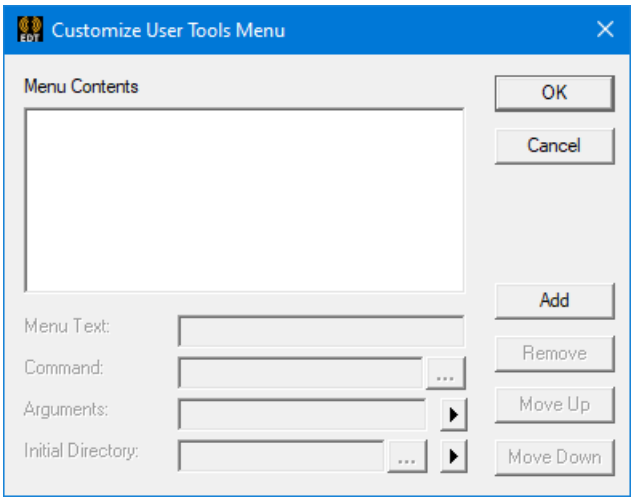
Before running a job you must click **Tools** > **Job Management** > **Select Scheduler** and use the dialog box to designate the head node of a cluster. You can then click **Tools** > **Job Management** > **Submit Job** to submit the batch commands for the job.

Adding External Tools

To add an **ExternalTools** menu to Ansys Electronics Desktop:

1. Click **Tools** > **External Tools**. This displays the **Customize User Tools Menu** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If a **User Tools** menu has been previously defined, its contents are displayed in the **Menu Contents** window. Navigation buttons allow you to **Add**, **Remove**, **MoveUp** and **MoveDown** items which are displayed in the **Menu Contents** window.

2. Click **Add**. This enables the following fields:

   Menu Text — Displays the text you want to appear in the **UserTools** menu.

   Command – Displays the external executable. Click the browse button to navigate to the file location.

   Arguments – Specifies the argument to be associated with the command. Click the right-arrow button to choose from the following: File Path, File Directory, File Name, File Extension, Project Directory, or Temp Directory.

   Initial Directory – Specifies an initial directory for the command to operate in. Click the browse button to navigate to the file location. Click the right-arrow button to navigate through your desktop or across the network.

3. Click **OK** to add the **ExternalTools** menu to Ansys Electronics Desktop, or click **Cancel** to close the dialog box without changes.

Note:

To execute a custom script using the **ExternalTools** customization, enter the name of the program that will execute the script in the **Command** field and enter the location of the script in the **Arguments** field. For example:

For Windows, enter the name of the non-graphical Windows Script Host — 'cscript' — in the **Command** field and enter the location of the script in the **Arguments** field:

Arguments: C:\scripts\myscript.py

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Customizing Ansys Electronics Desktop with Ansys ACT

With Ansys ACT, you can create custom applications or extensions to customize supported Ansys products, including Electronics Desktop.

> **Note:**
>
> The **ACT Extensions** window and the design wizards it contains (*5G Wizard, HFSS Antenna Design Toolkit, HFSS-EMA3D Link,* and *Maxwell Eccentricities*) are only available for the Windows version of the Ansys Electronics Desktop software. These items are not available when the software is installed on a Linux platform.

An ACT guided process extension enables you to leverage both the functionality of Electronics Desktop and the scripting capabilities of the Workbench/AIM framework API. You can manipulate existing features and simulation components, organizing them as needed to produce a custom automated process. A guided process extension is exposed in Electronics Desktop as a wizard that provides step-by-step simulation guidance within the application workflow.

To access ACT functionality in Electronics Desktop, open the ACT Home page by clicking **View** > **ACT Extensions**.

For more information, see ACT Simulation Wizards and Electronics Desktop Wizard in the *Ansys ACT Developer's Guide*.

Installing PyAEDT (Beta)

PyAEDT is a Python library that interacts with the AEDT API to make scripting simpler for the end user. It supports all AEDT 3D products (HFSS, Icepak, Maxwell 3D, and Q3D Extractor), 2D tools, Ansys Mechanical, EMIT, Circuit tools like Nexxim, system simulation tools like Twin Builder, and layout tools like HFSS 3D Layout and EDB. Additionally, it enables the end user to have a CPython interface with AEDT. Its class and method structures simplify operation for the end user, enabling more Pythonic code while reusing information as much as possible across the various APIs.

Documentation for PyAEDT can be found online at: https://aedt.docs.pyansys.com/version/stable/

Installing PyAEDT adds three items to the **Tools** > **Toolkit** > **PersonalLib** menu and to the **Automation** tab:

- **Console** – launches the PyAEDT console.
- **Jupyter Notebook** – launches Jupyter Notebook (a computational notebook) in an internet browser.
- **Run PyAEDT Script** – launches a file browser allowing you to select a Python script to run via PyAEDT.

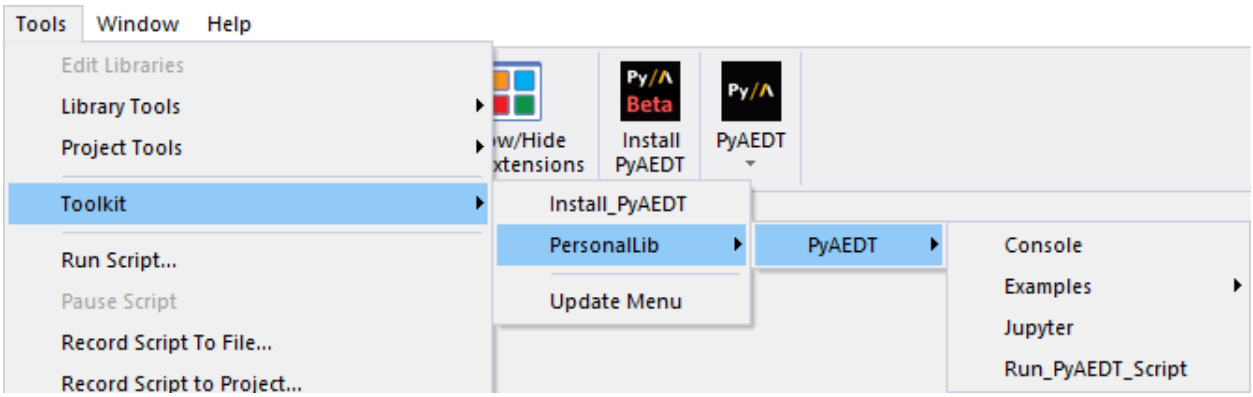
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

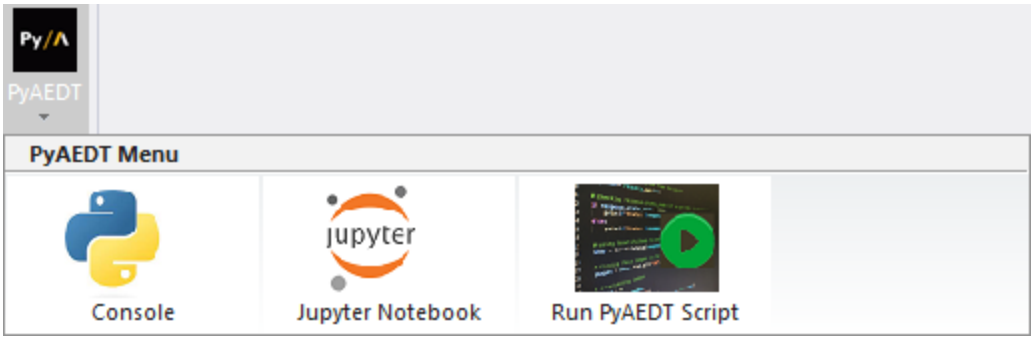
To install PyAEDT:

- From the **Automation** tab, click **Install PyAEDT**.

A web browser launches, and takes you to detailed installation instructions.

When installation is complete, the **Tools** > **Toolkit** > **PersonalLib** menu and the **Automation** tab update to display PyAEDT menu options:





Debug Logging

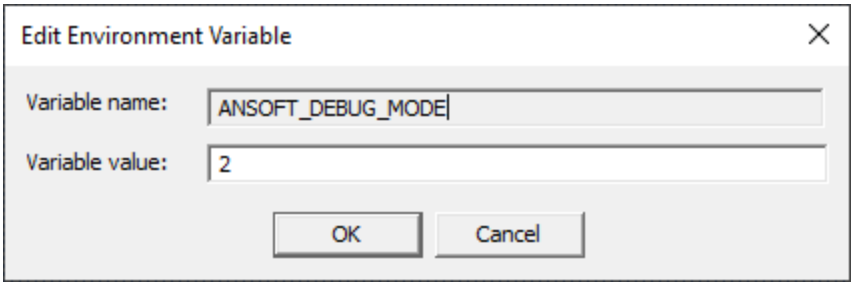
Electronics Desktop provides error logging capabilities.

To begin logging errors:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Tools** > **Debug Logging**.

   The **Edit Environment Variable** window appears, showing the ANSOFT_DEBUG_ MODE variable.

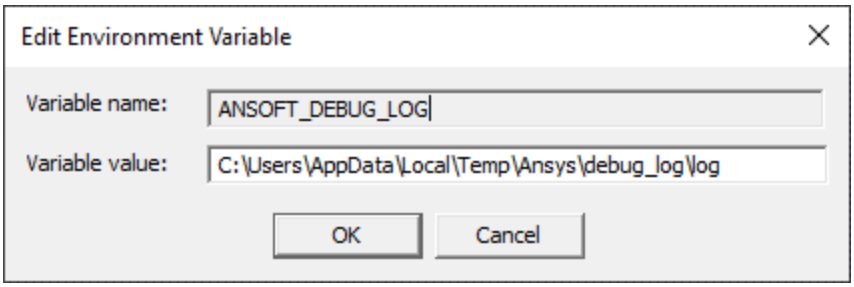


2. Enter a **Variable value**.

   The default value is 2, and should not be changed unless directed by technical support.

3. Click **OK**.

   The **Edit Environment Variable** window updates to display the ANSOFT_DEBUG_LOG variable.



4. Enter a folder path where log files (*.log) will be stored.
5. Click **OK**.

   Errors are logged to the specified folder until you disable logging.

To stop logging errors:

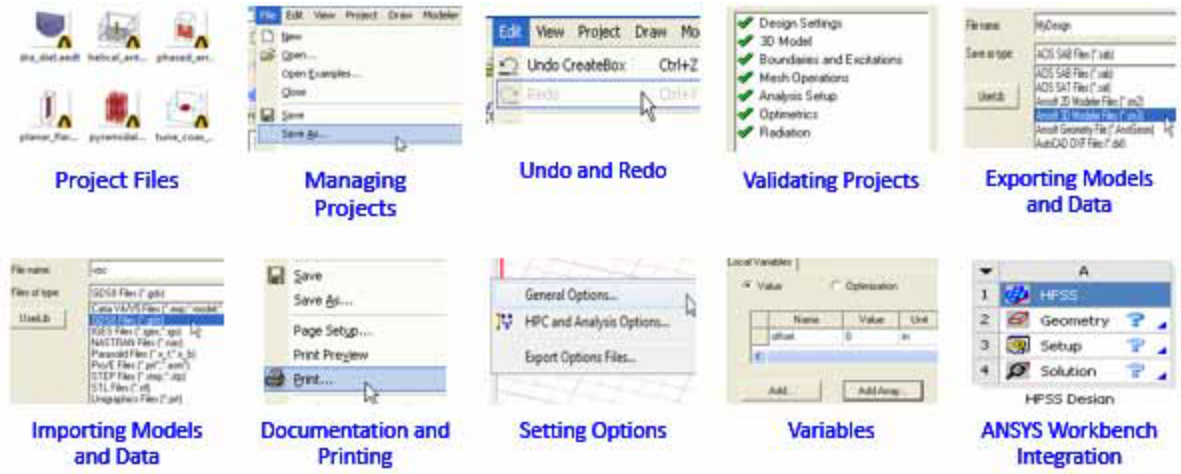
- Click **Tools** > **Stop Debug Logging**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3 - Working with Ansys Electronics Desktop Projects

An Ansys Electronics Desktop project is a folder that includes one or more models, or *designs*. Each design ultimately includes a geometric model, its boundary conditions and material assignments, and field solution and post-processing information, or a schematic or netlist.



A new project called *Projectn* is automatically created when the software is launched. By option, a design named Design*n* is automatically created for a new project. You can also open a new project by clicking **File** > **New**. In general, use the **File** menu commands to manage projects. If you move or change the names of files without using these commands, the software may not be able to find information necessary to solve the model.

> **Note:**
>
> Not all options and capabilities documented in this section apply to all design types.

Ansys Electronics Desktop Files

When you create any project in Ansys Electronics Desktop, it is given an .aedt file extension and stored in the directory you specify. For Legacy projects, the transition is handled as follows:

- When opening a legacy project in interactive mode, a dialog box appears, informing you that the legacy file extension is no longer supported and that if you continue, the project will be converted to the new .aedt extension. If you agree, the project and results directory are renamed/moved to the new file extensions immediately. The read continues with the standard code for reading previous version projects.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The Ansys Electronics Desktop 2025R1 version cannot open projects created with 2022R2 or earlier that were using ACIS modeler.
- Attempts to run batch solve or non-graphical with a legacy project return an error. There is no automatic/hidden conversion of file extensions. Note that the existing BatchSave command can be used to convert many projects to the new extension and version.
- Workbench integration has not been modified. It continues to open legacy projects without a warning and copies results from legacy to the new extension.

Maxwell Files

When you create an Ansys Electronics Desktop project, it is given an .aedt file extension and stored in the directory you specify. Any files related to that project are also stored in that directory.

Some common file and folder types are listed below:

.aedt	Maxwell project.
project_name.**aedtresults**	Maxwell folder containing results data for a project.
design_name.**results**	Maxwell folder containing results data for a design. This folder is stored in the *project_name*.**aedtresults** folder.
design_name.**asol**	Maxwell results data for a design. This file's contents may be empty if a solution is unavailable. The .asol file contains the database of all solved variations. This file is stored in the *project_name*.**aedtresults** folder.
.mxwl	Maxwell file extension before Ansys Electronics Desktop.
.pjt	Ansys Electronics Desktop version 8.5 and earlier project.
.cls	Ansoft legacy EM projects.

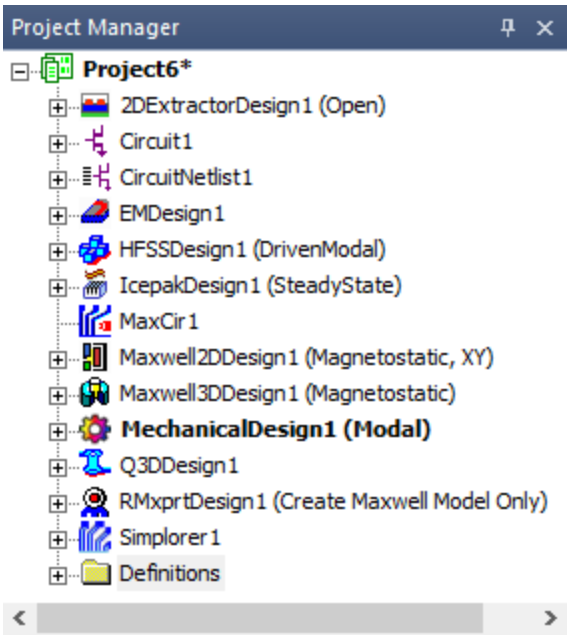
Creating Projects

To create a project, click **File** > **New**, or click the **New** icon on the **Desktop** tab of the ribbon.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A new project is listed in the project tree. It is named Project*n* by default, where *n* is the order in which the project was added to the current project folder.

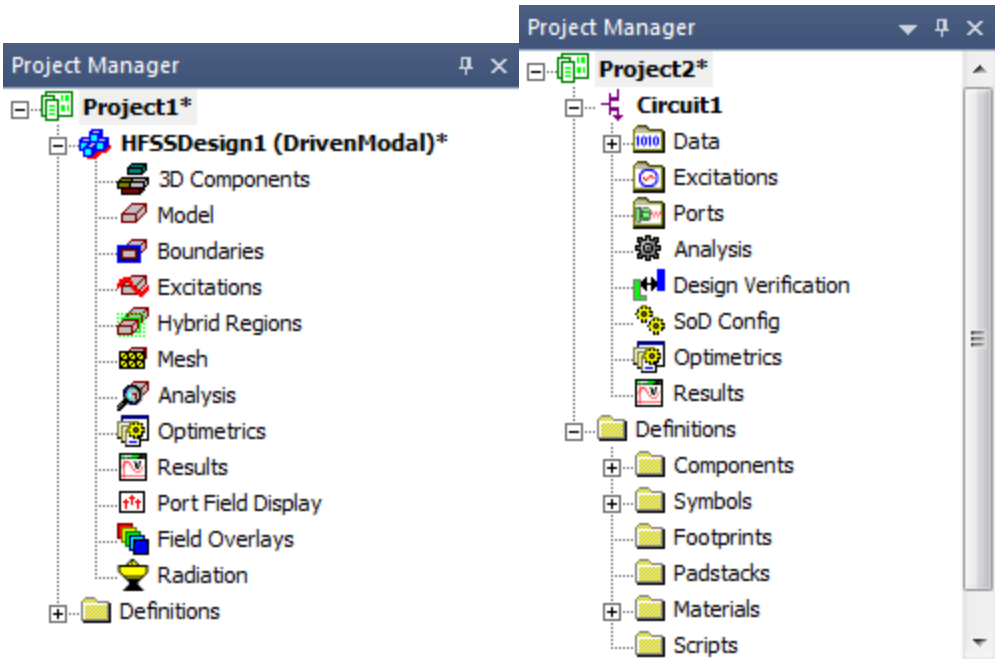
You can insert designs of any type into the project, where they are stored in the project tree.



The default name for each inserted project is <designType>*n*. You can also specify the name of the project when you save it using the **File** > **Save** or **File** > **Save As** commands.

You can view the contents of a project by clicking the plus sign (+) for each level of the hierarchy in the project or design.

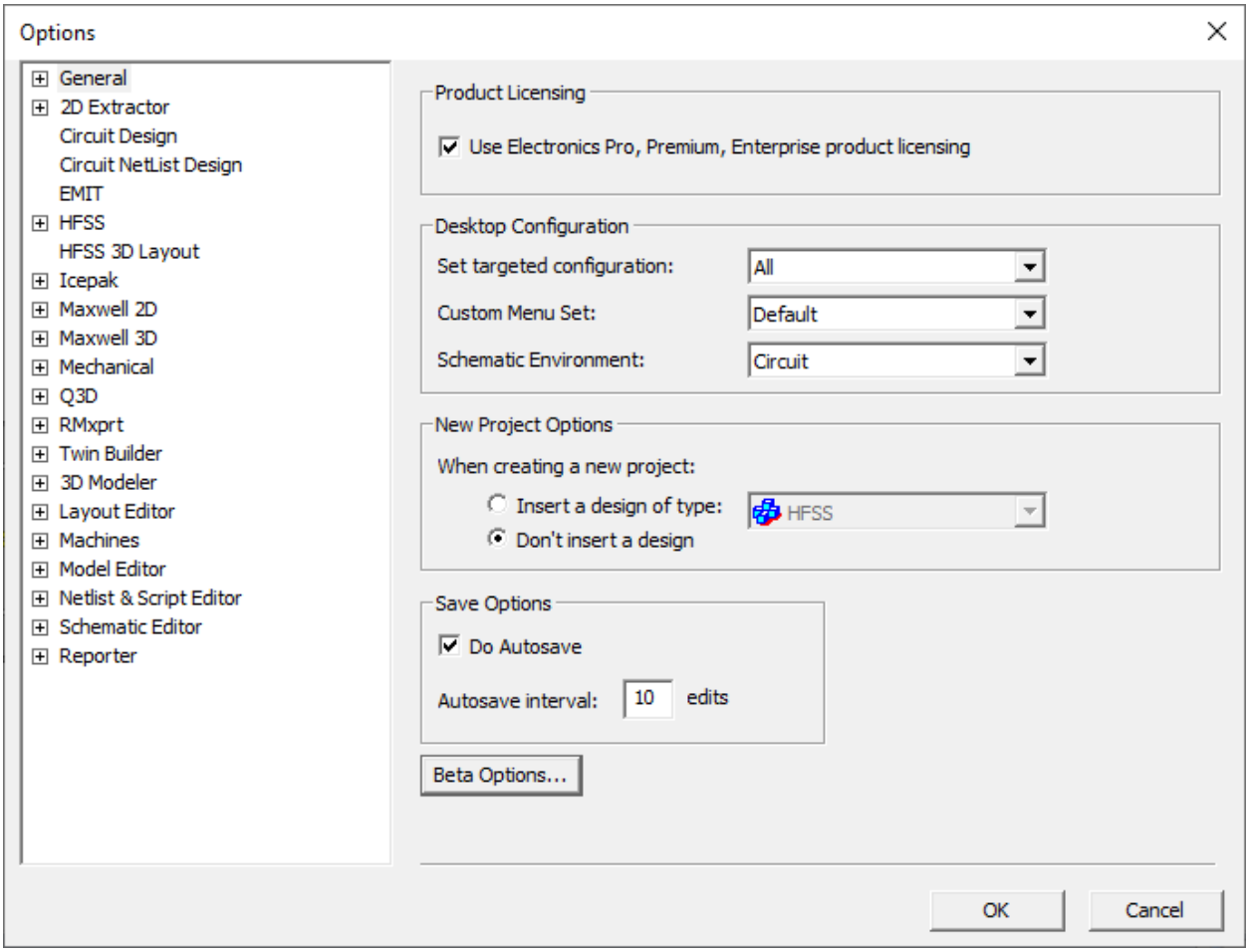
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Setting Project Options

Tools > **Options** > **General Options** opens the **Options** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The left pane provides access to the following sets of options:

- General
- 2D Extractor
- Circuit Design
- Circuit Netlist Design
- HFSS
- HFSS 3D Layout
- Icepak
- Maxwell 2D
- Maxwell 3D
- Mechanical
- Q3D
- RMxprt

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Twin Builder
- 3D Modeler
- Layout Editor
- Machines
- Model Editor
- Netlist and Script Editor
- Schematic Editor
- Reporter

Use **Tools** > **HPC and Analysis Options** to specify active configuration per design type, queuing, distributed memory vendor, HPC licensing, select a MPI vendor, and whether to enable GPU for transient solves.

The **Tools** > **Options** > **Export Options Files** command writes xml files containing the Options settings at all levels to the specified directory. The **Tools** > **Options** > **Export Options** feature is intended to make it easier for different users to use Ansys Electromagnetics tools installed on shared directories or network drives. The Example Uses for Export Options Features section outlines some use cases enabled by this feature.

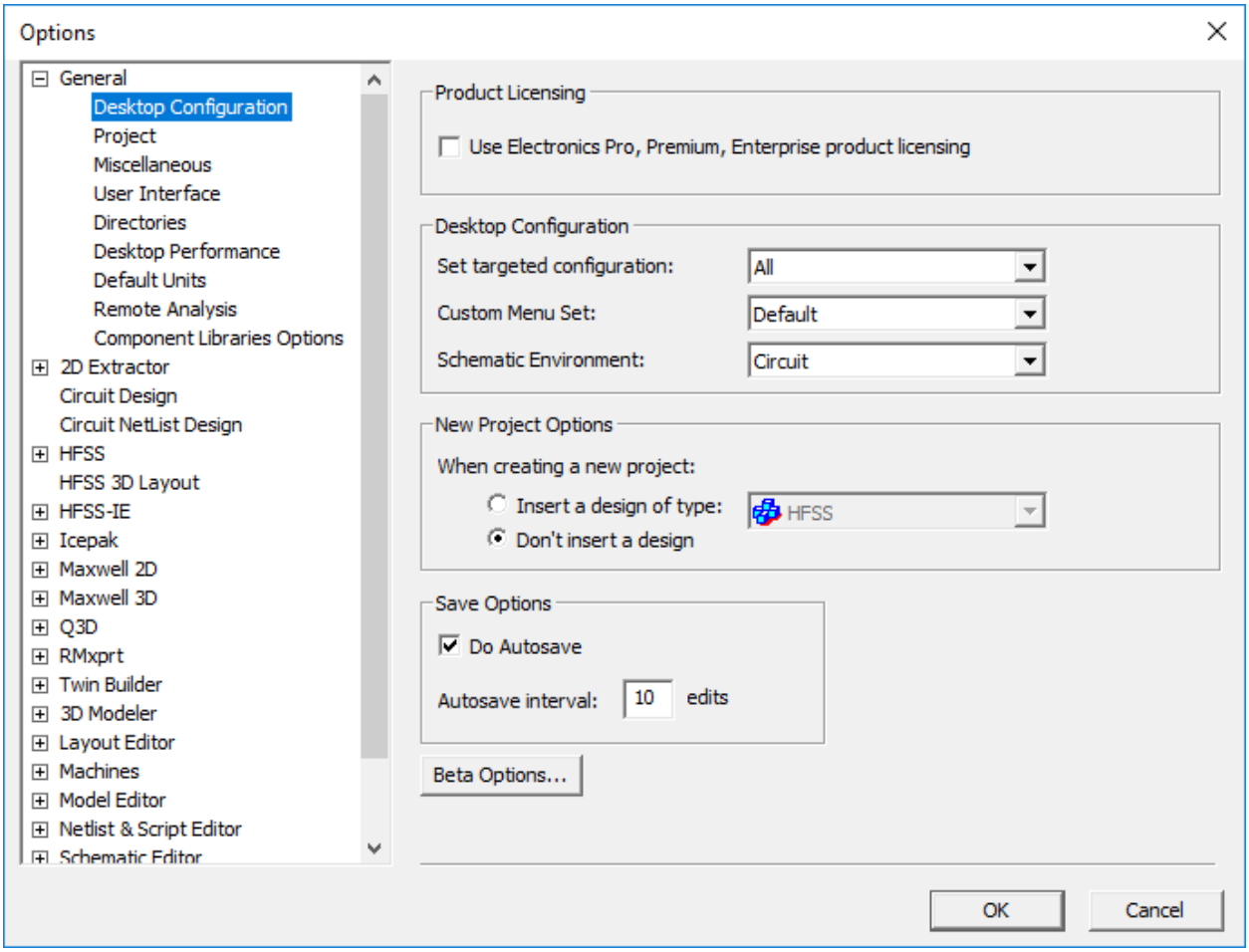
Setting General Options

To set general options in Ansys Electronics Desktop:

1. Click **Tools** > **Options** > **General Options**.

   The **Options** window opens with the Desktop Configuration options selected by default.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the left pane, select the entries below **General** to display the associated options:

- Desktop Configuration
- Project
- Miscellaneous
- User Interface
- Directories
- Desktop Performance
- Default Units
- Remote Analysis
- Component Libraries Options

2. Click each entry and make the desired selections.

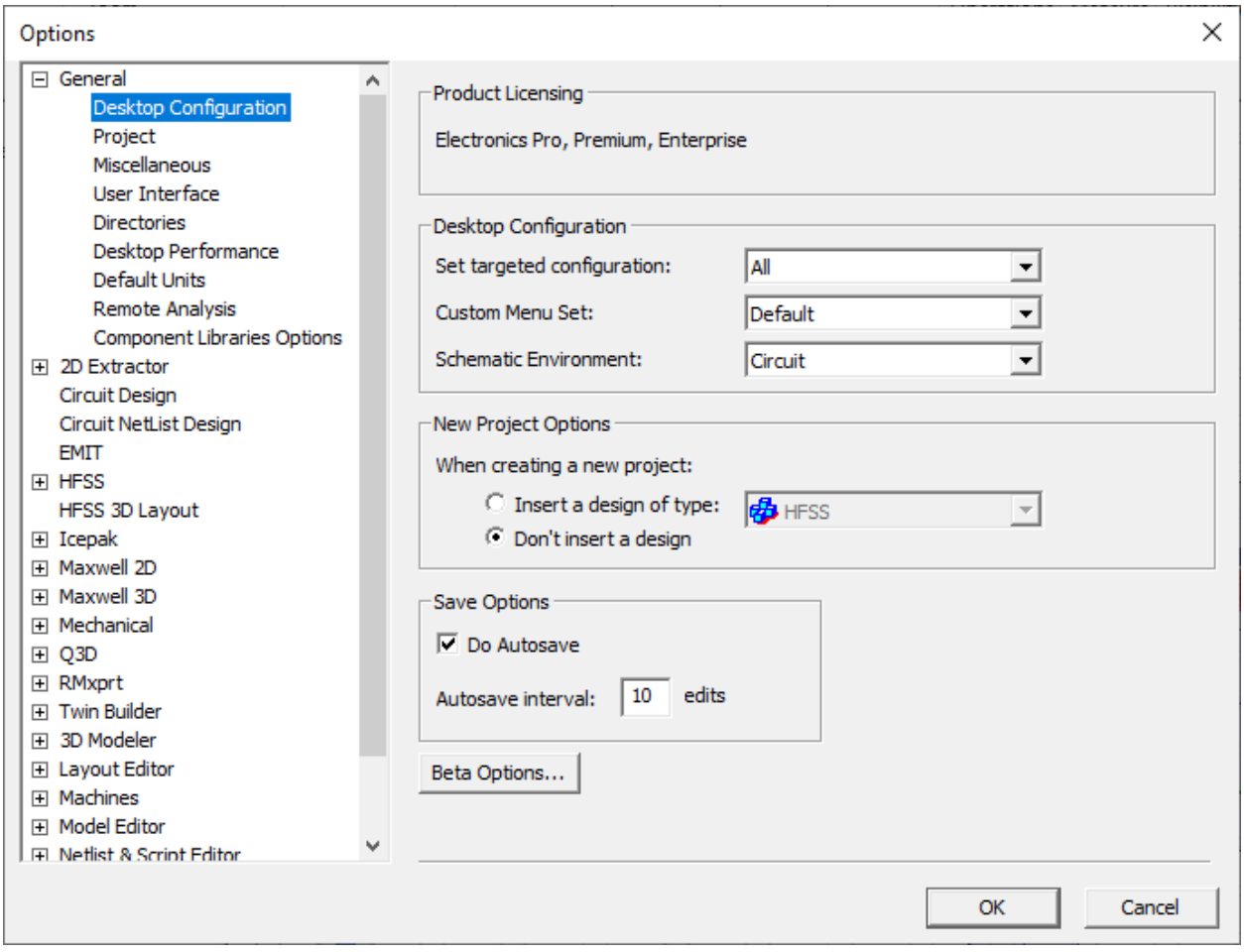
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **OK**.

> **Note:**
>
> Not all option settings are applied immediately in the current design. Some settings take effect in newly created designs.

General Options: Desktop Configuration

Under General Options, **Desktop Configuration** options allow you to customize Electronics Desktop in a way that suits your work priorities. These include options for menus, new projects, save intervals, and beta features.



Ansys Electronics Desktop now uses Electronics Pro, Premium, Enterprise (PPE) product licensing. Legacy product licensing and DSO are no longer supported.

With PPE, HPC licensing is used to enable all cores, GPUs, and distributed tasks.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For more details, see Setting HPC and Analysis Options.

In the **Desktop Configuration** area, options include:

- **Set Targeted Configuration** – Choosing a Targeted Configuration changes the Custom Menu Set and Schematic Environment for your area of focus.

  Choices are:

  - **All** – Default option. Sets the **Custom Menu Set** to **Default**, the **Schematic Environment** to **Circuit**, and the default design type to **HFSS**.
  - **EM** – Electromagnetic focus. Sets the **Custom Menu Set** to **EM**, the **Schematic Environment** to **Twin Builder**, and the default design type to **Maxwell 3D**.
  - **RF** – Radio Frequency focus. Sets the **Custom Menu Set to** RF, the **Schematic Environment** to **Circuit**, and the default design type to **HFSS**.
  - **SI** – Signal Integrity focus. Sets the **Custom Menu Set** to **SI**, the **Schematic Environment** to **Circuit**, and the default design type to **HFSS 3D Layout**.
  - **Twin Builder** – Sets everything to default to **Twin Builder**.
- **Custom Menu Set** – Changes which menu options are available in Electronics Desktop.

  Choices are:

  - **Default** – All solvers appear on the **Project** menu.
  - **EM** – Only electromagnetics solvers appear on the **Project** menu.
  - **RF** – Only radio frequency solvers appear on the **Project** menu.
  - **RF.0** – All solvers appear on the **Project** menu. The **HFSS RF Setup** menu appears.
  - **SI** – Only signal integrity solvers appear on the **Project** menu.
  - **SI1.0** – Only signal integrity solvers appear on the **Project** menu. The **Import**, **Solution Setup**, **Automation**, and **Definitions** menus appear.
  - **SI2.0** – All solvers appear on the **Project** menu. The **HFSS SI Setup** menu appears.
  - **Twin Builder** – Only Twin Builder appears on the **Project** menu.
- **Schematic Environment** – Select **Circuit**, **Twin Builder**, or **Maxwell**.

In the **New Project Options** area, options include:

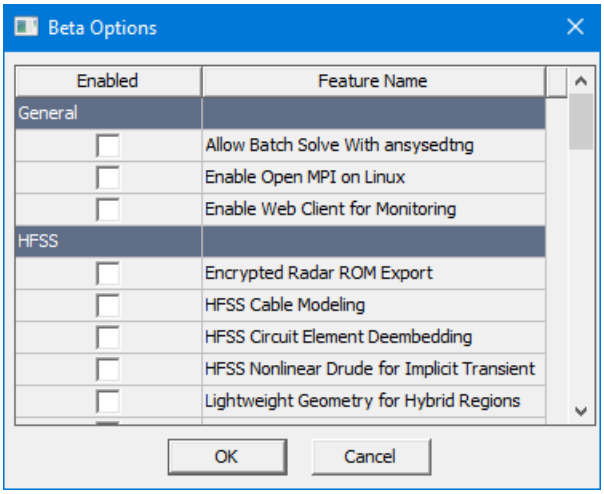
- **When Creating a New Project** – Select whether to insert a design when creating a new project. If you choose to insert a design by default, use the drop-down menu to select the default design type.

In the **Save Options** area, options include:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Do Autosave** – Electronics Desktop has autosave enabled by default. Deselect this option to disable it.
- **Autosave Interval** – When **Do Autosave** is enabled, select the number of edits at which Electronics Desktop autosaves.

Click the **Beta Options** button to open a window listing beta options. You may need to scroll or size the window to view all options.

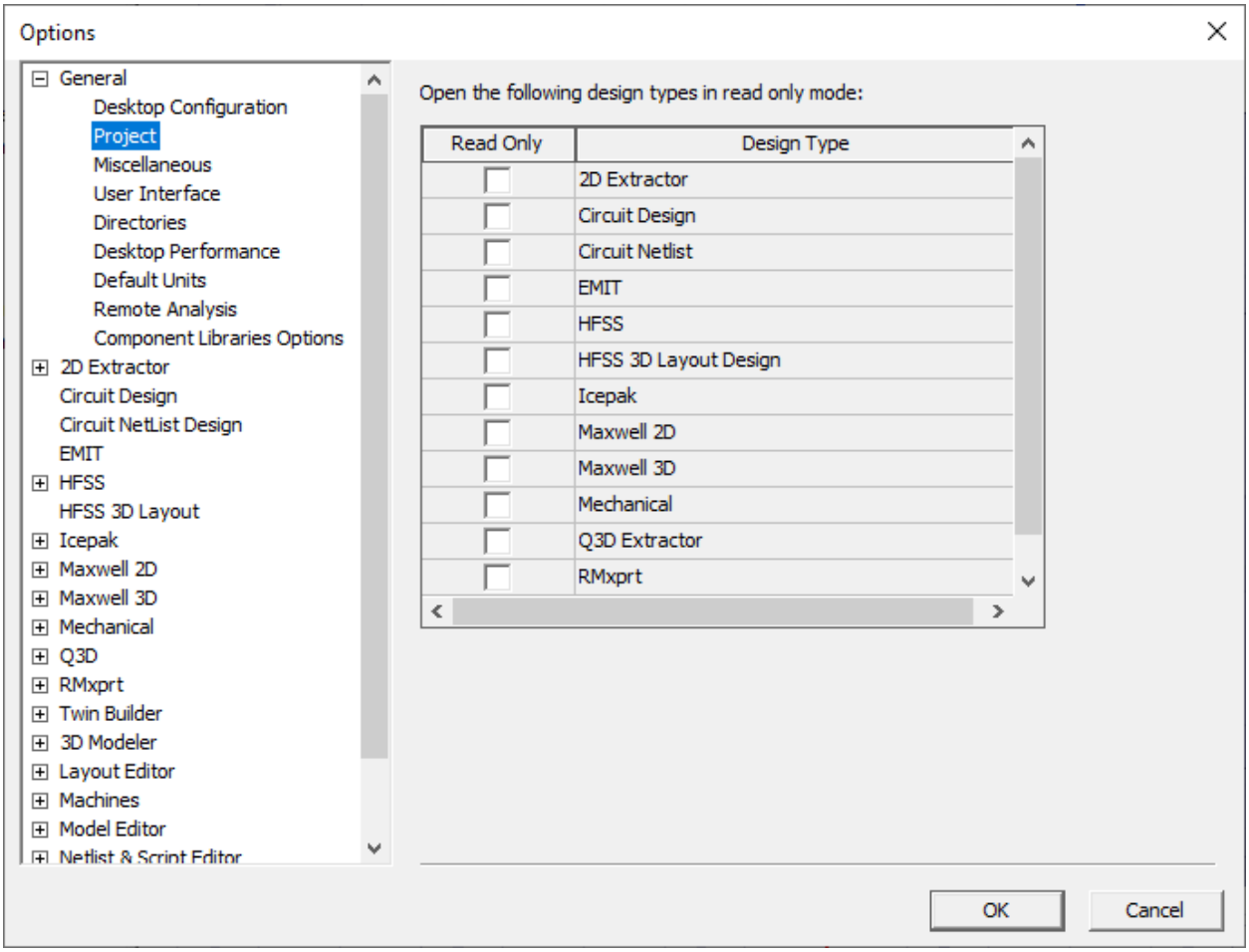


From there, enable or disable options, and click **OK**. You will need to restart to enable your selections.

General Options: Project Options

Under General Options, **Project** options allow you to select design types to open in read-only mode. This can help prevent accidental changes.

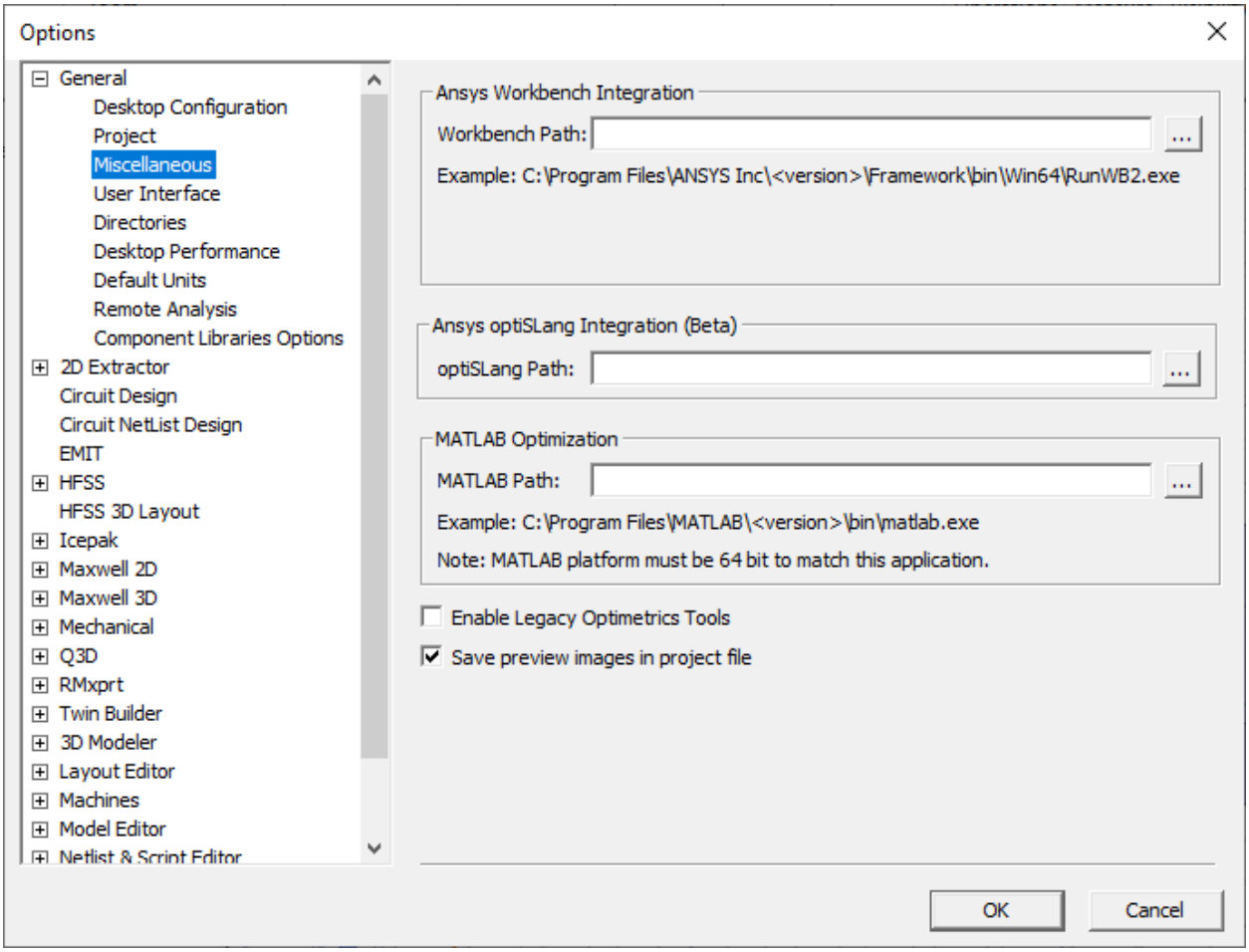
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



General Options: Miscellaneous Options

Under General Options, **Miscellaneous** options allow you to specify paths for Ansys Workbench Integration, optiSLang path, MATLAB Optimization and to enable Legacy Optimetrics Tools.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Workbench Integration allows you to specify the path to your Ansys Workbench installation. This path can be used in Optimetrics for connecting to the Design Xplorer.

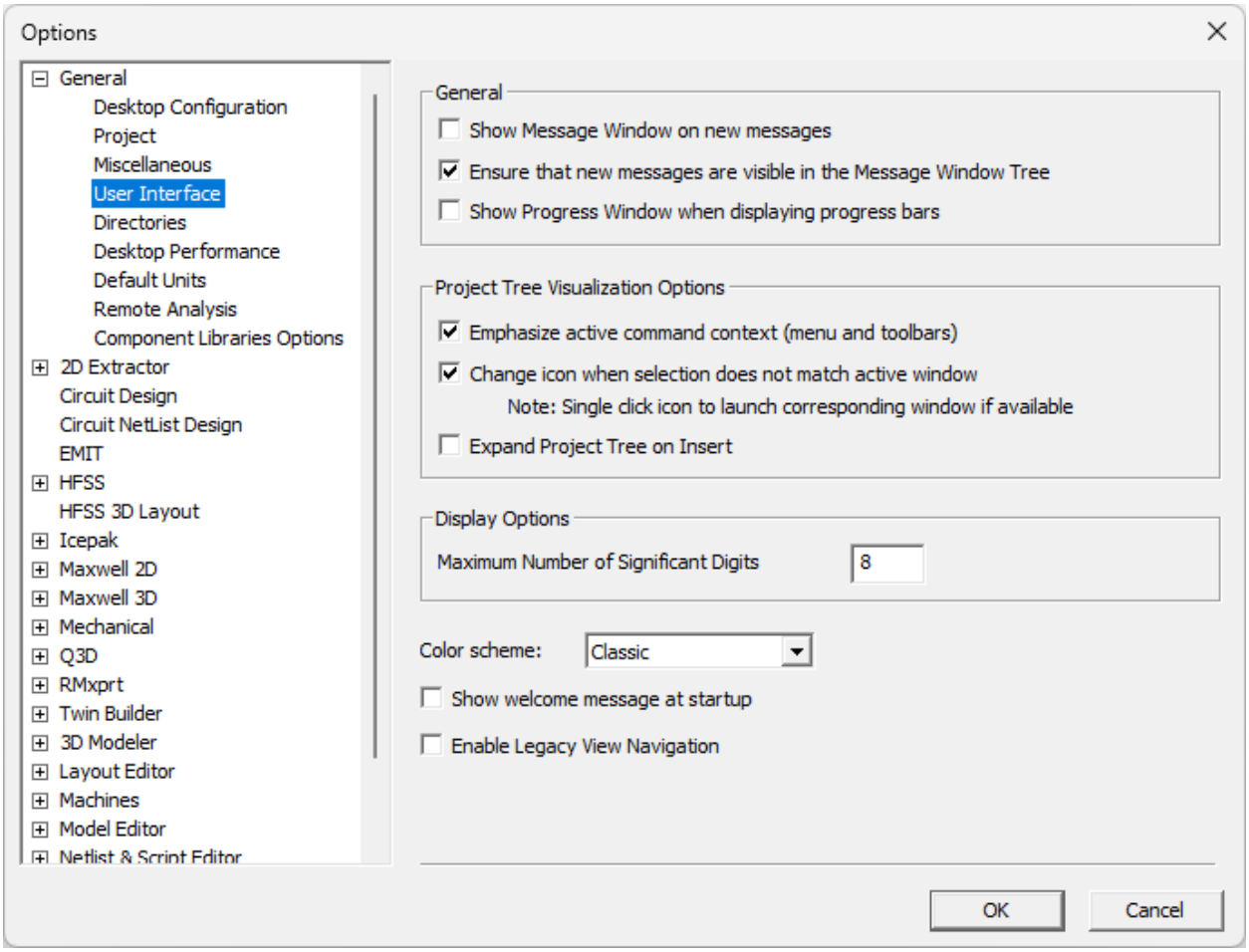
MATLAB Optimization allows you to specify the path to your installation of MATLAB. MATLAB can be used as an Optimizer.

> **Note:**
>
> The platform for MATLAB and Ansys Electronics Desktop must match (for example, a 64 bit version of MATLAB).

General Options: User Interface Options

Under General Options, **User Interface** options allow you to change how Electronics Desktop displays messages, command text, the project tree, welcome messages, and more.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the **General** area, options include:

- **Show Message Window on new messages** – when selected, the message window automatically opens if a message arrives.
- **Ensure that new messages are visible in the Message Window Tree** – when selected, the message window expands as needed to display messages.
- **Show Progress Window when displaying progress bars** – when selected, the progress window automatically opens while simulations are in progress.

In the **Project Tree Visualization Options** area, options include:

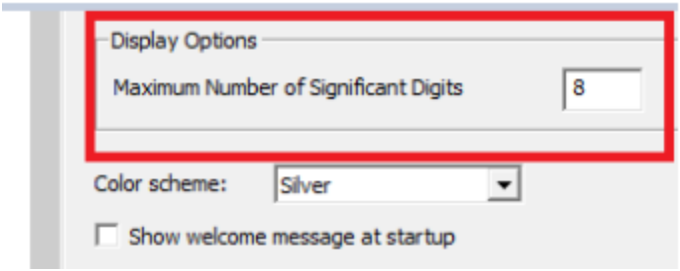
- **Emphasize active command context** – when selected, active elements in the Project Tree display in bold text.
- **Change icon when selection does not match active window** – when selected, a small, window-shaped overlay icon displays in the corner of the selected Project Tree element. This icon changes when the data in the active window is unrelated to the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

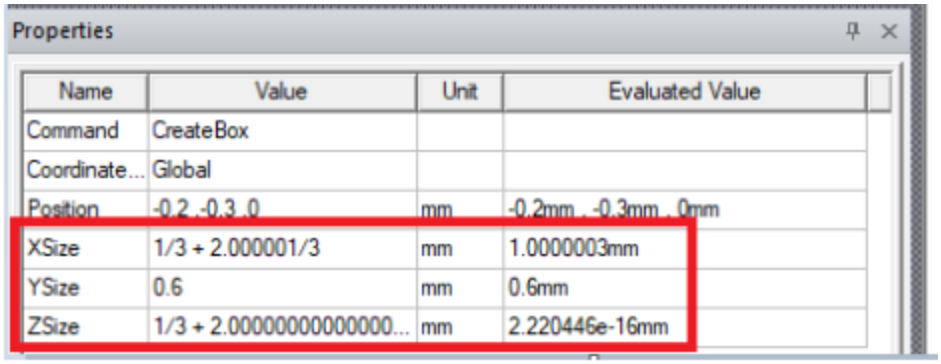
selected project item (data affecting the same model is considered to be related). Clicking the icon opens the window and brings it into focus.

- **Expand Project Tree on Insert** – when selected, the Project Tree automatically expands when you insert a new design.

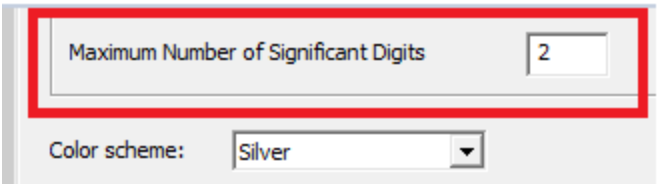
In the **Display Options** area, specify the **Maximum Number of Significant Digits** to display. The default is 8 and the maximum is 20. This affects the digits displayed in the **Solutions** dialog box, evaluated variable values, Animation dialog boxes, Optimetrics, Reports, and so forth. For example, here is the default.



So some evaluated variable values with significant digits appear as follows.

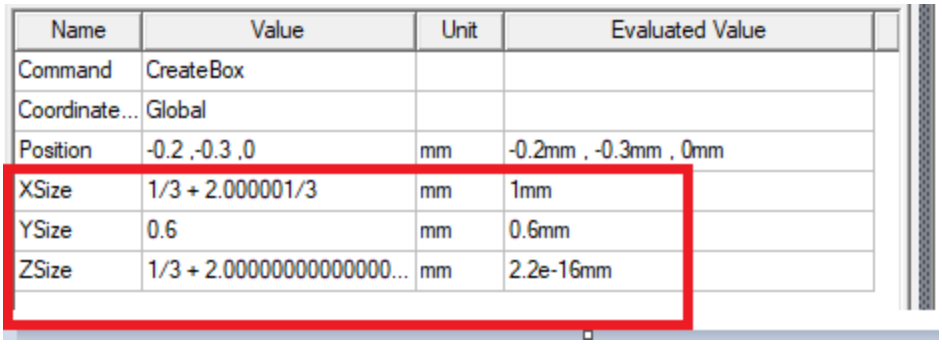


Change Precision = 2. This lets you display near-zero values in the Properties window as 0.



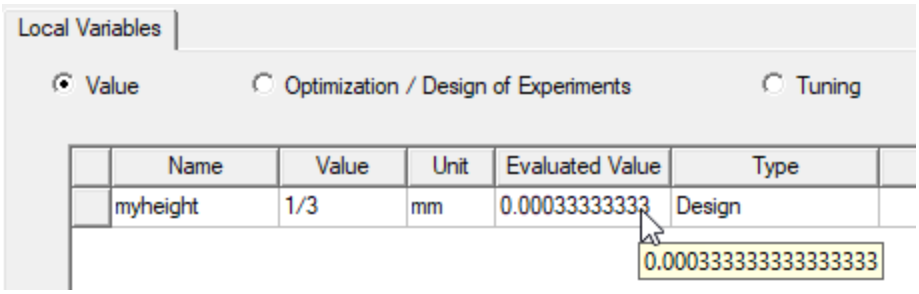
You can see that XSize is now rounded off to cleanly display 1mm. The decimal part of ZSize also shows cleaner value, but it is not rounded off to zero.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

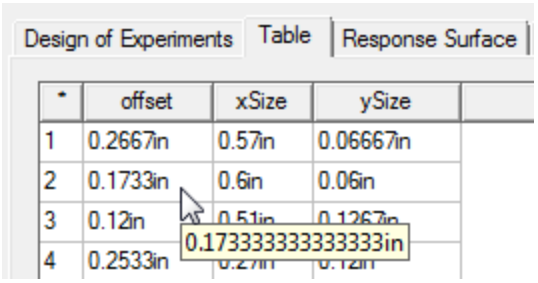


Name	Value	Unit	Evaluated Value
Command	CreateBox		
Coordinate...	Global		
Position	-0.2 ,-0.3 ,0	mm	-0.2mm , -0.3mm , 0mm
XSize	1/3 + 2.000001/3	mm	1mm
YSize	0.6	mm	0.6mm
ZSize	1/3 + 2.00000000000000...	mm	2.2e-16mm

You can still see the full precision values in tooltips by holding a cursor over the displayed value.



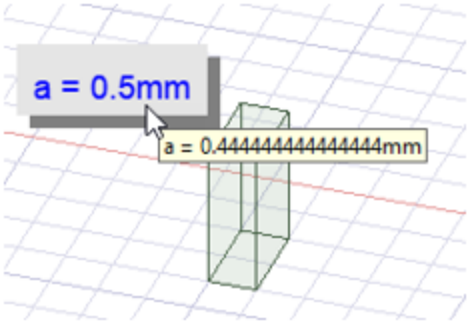
In the case of variable values, if you have assigned more significant digits, you will see these when editing the variable value. In the case of table displays of values, the tooltip display shows all available digits when the mouse pointer is over a result:

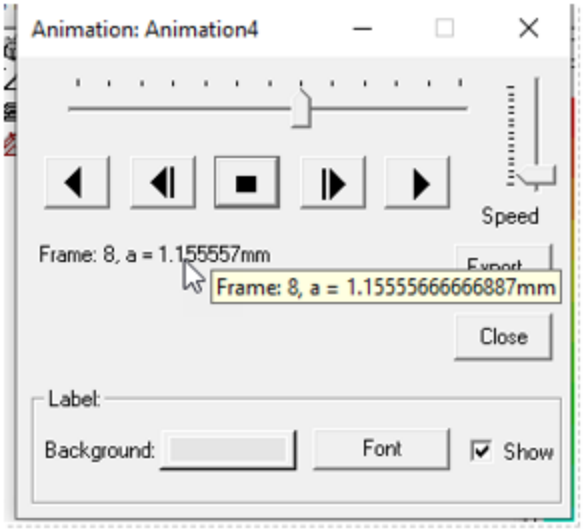


If you set the **Maximum Number of Significant Digits** to a lower value, Change Precision = 2. You can see that XSize is now rounded off to cleanly display 1mm. The decimal part of ZSize also shows cleaner value, but it is not rounded off to zero.

The tooltip functions to show internal digits in throughout the Ansys Electronics Desktop interface.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Select a **Color Scheme**. The choices are **Light (Beta)**, **Dark (Beta)**, or **Classic**: See Choosing a Color Scheme for more information and samples of each scheme.

Select **Show welcome message at startup** if you want to see a welcome message when Electronics Desktop opens.

Enable legacy view navigation enables you to choose between two schemes for view navigation keyboard shortcuts and mouse button assignments, as follows:

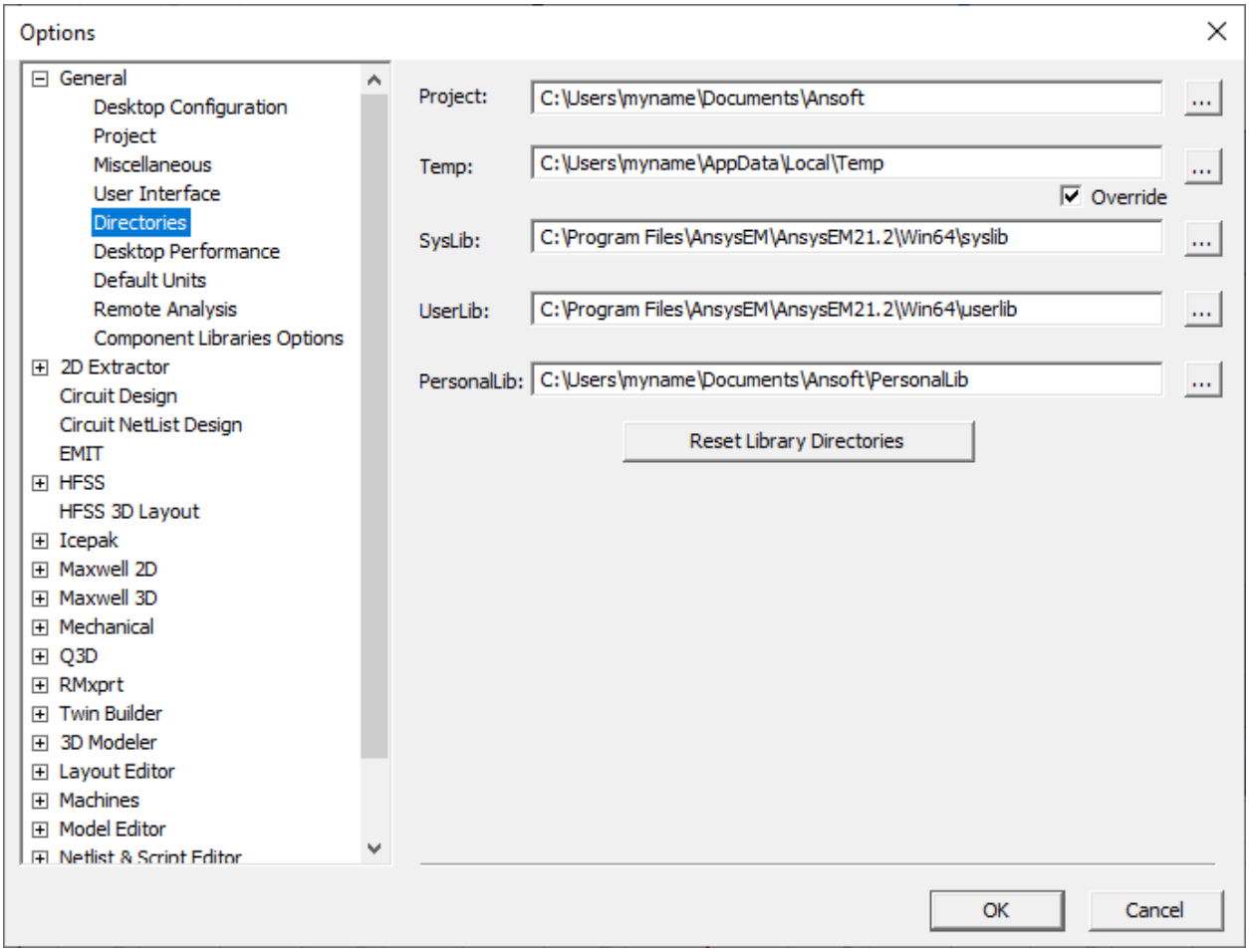
- **Cleared:** Use only the view navigation mouse-button and hotkey assignments introduced in Ansys Electronics Desktop version 2024 R1 and applicable to subsequent versions too. Legacy (2023 R2 and earlier) mouse-button and hotkey assignments will *not* be recognized when this option is cleared.
- **Selected:** Both the legacy (2023 R2 and earlier) and current (2024 R1 and newer) mouse-button and hotkey assignments are supported, and either can be used for view navigation.

See Choosing the View Navigation Options for more information, including a detailed comparison of the two schemes.

General Options: Directories Options

Under General Options, **Directories** options allow you to get the paths for Project, Temp, SysLib, UserLib, and PersonalLib directories.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Use the **[...]** buttons to browse to paths and click **OK**. You will need to select **Override** to change the Temp directory.

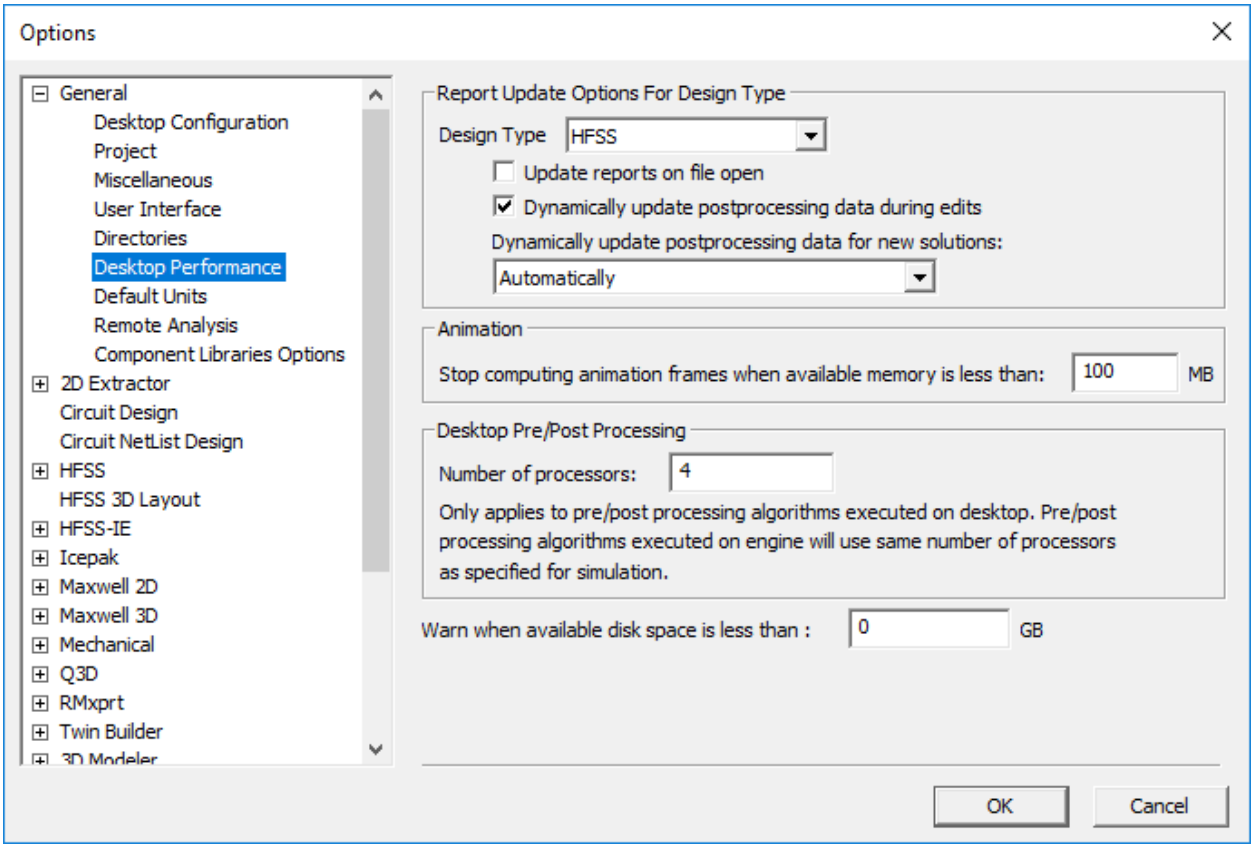
If you need to restore defaults, click **Reset Library Directories**.

> **Note:**
>
> Your changes will be reflected in the User Defined Primitives menu on next startup.

General Options: Desktop Performance

Under General Options, **Desktop Performance** options allow you to change settings for animation, pre/post-processing, disk space warnings, and updating reports.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



These options are set on the **Desktop Performance** panel under **General** in the **Options** window.

In the **Report Update Options for Design Type** area, select a **Design Type**. For each type, you can set the following options:

- **Update Reports on File Open** – when selected, reports are automatically updated whenever an existing file with solution data is opened for viewing/editing.
- **Dynamically Update Postprocessing Data During Edits** – when selected, report plots and overlays update as you edit their parameters.
- **Dynamically Update Postprocessing Data for New Solutions** – Because updating reports during analysis can impact solution time, you can specify how often your reports are updated:
  - **After Each Variation** – updates reports after analysis of each variation has been completed. Used for an Optimetric or parametric analysis.
  - **Automatically** – balances report and field plot updating with solution time.

    For Adaptive Passes, plots update at the end of each solution pass. For Last Adaptive or Transient, plots update at the end of the transient or adaptive solution.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, reports may be updated after each adaptive pass but field plots will not be updated until the solution is complete.

- **Immediately** – updates reports and plots as soon as data comes from the solver.

  This option will have the greatest impact on overall solution time, but affords the fastest updates to reports and field plots. Caution should be used in selecting this option. Some types of reports and field plots may take a long time to update, especially as mesh size increases.

- **Never** – updates reports only upon manual intervention. This prevents updates from impacting solution time.
- **On Completion** – updates reports once, when the solution completes.

> **Note:**
>
> Updates done on completion are done after the solve has been completed, and the time for that update is not included in the simulation profile.

In the **Animation** area, you can elect to **Stop computing animation frames when available memory is less than** a specified value, in MB (the default is 100MB). This setting is used to prevent problems related to low memory should an animation require large memory allocation.

In the **Desktop Pre/Post Processing** area, you can specify the **Number of Processors**. This option only affects pre- and post-processing (not solve or simulation). Pre-processing algorithms can take advantage of multiple processors for visualization and faceting of 3D models, model validation for 3D products, auto net identification for Q3D, and more. The default value is determined by the number of logical processors on the machine running Electronics Desktop:

- The default core usage per desktop session (UI + solve) is set to 2/3 of the logical processors on the machine.
- The cores for default local config is max{4, 1/3 of the logical processors on the machine}
- The default number of processors for pre/post is min{1, 2/3 of the logical processors - default_cores_for_local_config}
- When the core usage per desktop session cannot be evenly distributed between solve and pre/post, more cores will be assigned to solve.

Example 1:

- logical processors = 24
- cores usage per desktop session = 16
- local hpc set to 4 cores
- processors for pre/post set to 12

Example 2:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- logical processors = 20
- cores usage per desktop session = 13
- local hpc set to 4 cores
- processors for pre/post set to 9

Example 3:

- logical processors = 5
- cores usage per desktop session = 3
- local hpc set to 2
- processors for pre/post set to 1

You can also elect to **Warn when available disk space is less than** a specified value, in GB.

General Options: Default Units

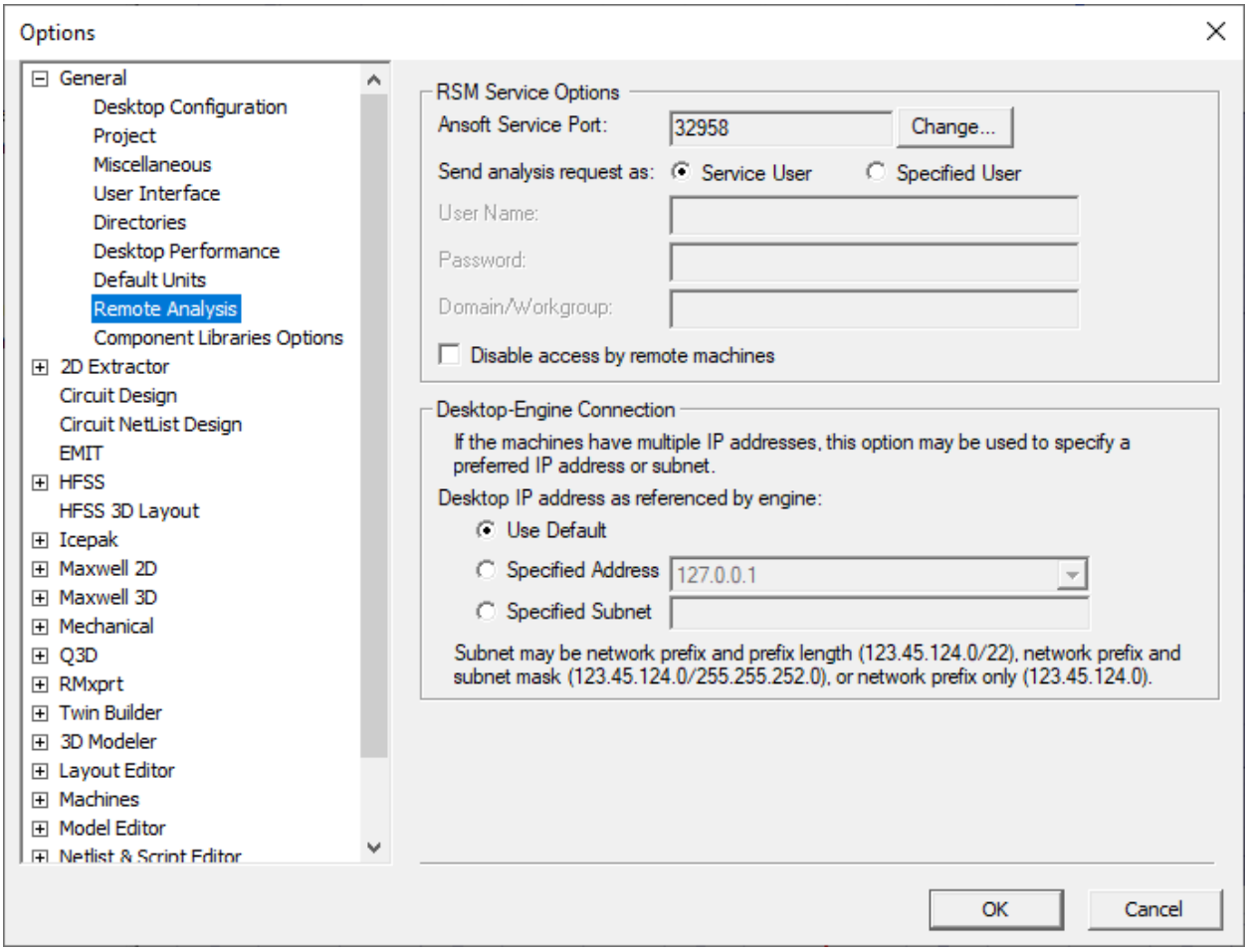
Under General Options, **Default Units** options allow you to set default units for the following:

- Length
- Angle
- Time
- Temperature
- Torque
- Magnetic Induction
- Pressure
- Frequency
- Power
- Voltage
- Current
- Speed
- Mass
- Conductance
- Resistance
- Inductance
- Capacitance
- Force
- Angular Speed
- Magnetic Field Strength

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Options: Remote Analysis Options

Under General Options, **Remote Analysis** options allow you to launch all analyses as a service or specified user rather than as the current user.



In the **RSM Service Options** area, options include:

- **Ansoft Service Port** – Click **Change** to update the port number. Ansys Electromagnetics RSM Service should be running on this port for all distributed machines.
- **Send Analysis Request As** – Select either **Service User** or **Specified User**. Selecting **Specified User** enables the **User Name**, **Password**, and **Domain/Workgroup** fields.

> **Note:**
>
> If any of the remote machines is Linux-based, you must specify the current user.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Disable Access By Remote Machines** – If desired, select to disable access for remote machines.

When multiple IP addresses are available, the **Desktop-Engine Connection** area allows you to specify the preferred IP address for communication:

- **Use Default** – your system's default IP address.
- **Specified Address** – an IP address you specify.
- **Specified Subnet** – a subnet you specify. Subnet may be network prefix and prefix length (123.12.123.0/22), network prefix and subnet mask (123.12.123.0/255.255.252.0), or network prefix only (123.23.123.0).

Changing the Listening Port used by Ansys RSM Service

To change the listening port used by the RSM Service, you must change the ansoftrsmservice.cfg file, as follows:

Specify the ListenPort within a 'CommDetails' block, which must be within a 'Default:CommDetails' block, which must be within the top level block of the file (the 'AnsoftCOMDaemon' block). The following example changes the listen port from 32958 to 32957, with these blocks at the beginning of the file:

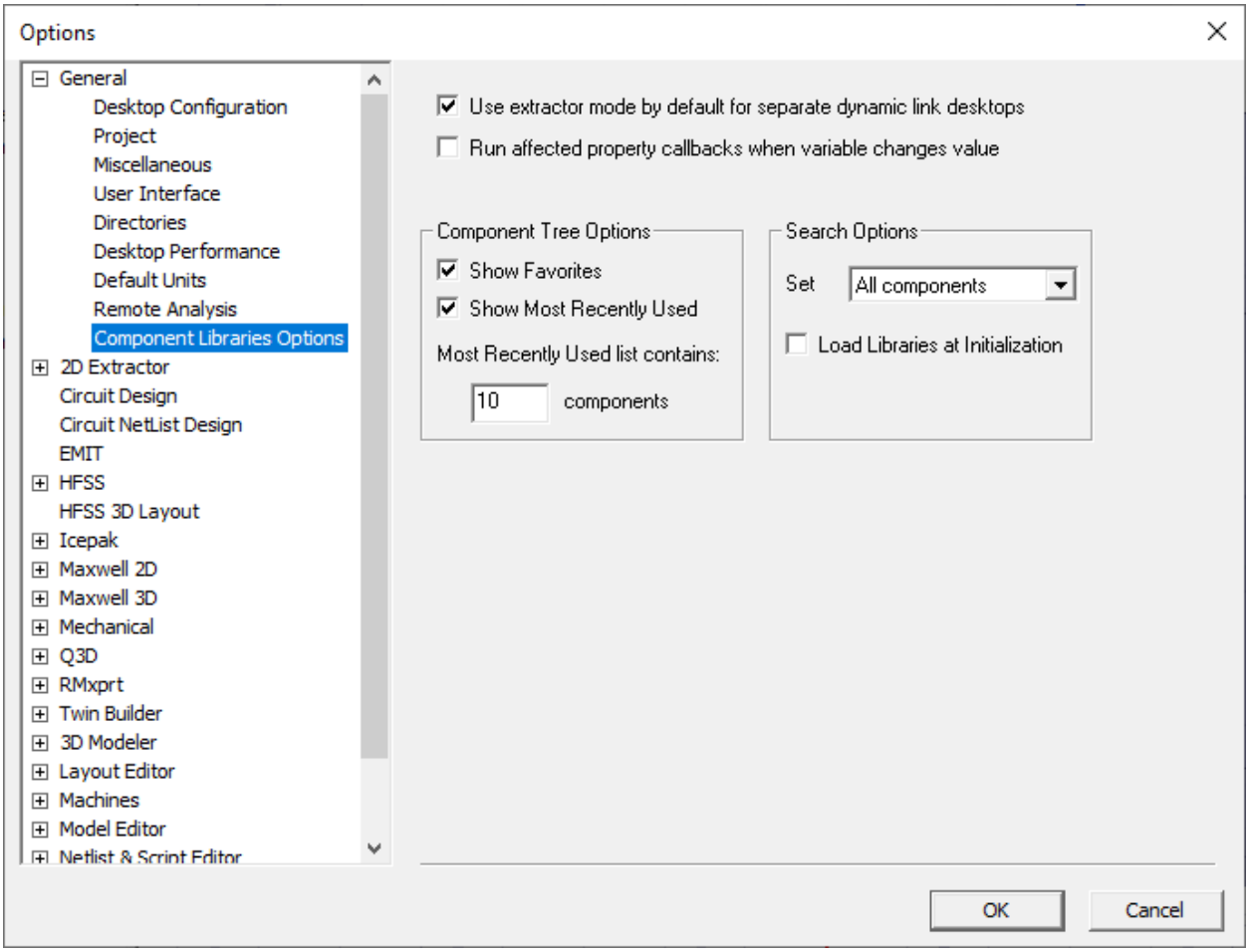
```
$begin 'AnsoftCOMDaemon'
  $begin 'Default:CommDetails'
    $begin 'CommDetails'
      ListenPort='32957'
    $end 'CommDetails'
  $end 'Default:CommDetails'
  . . . .
$end 'AnsoftCOMDaemon'
```

For the second level block, ensure that there is a single colon character and no spaces or tabs separating the two parts of the block name 'Default:CommDetails'. The third level block, with name 'CommDetails' is also required. Use caution when editing this file by hand, because any typos in the block or value names may cause the data to be ignored.

General Options: Component Libraries Options

Under General Options, **Component Libraries** options allow you to change how Electronics Desktop handles components.

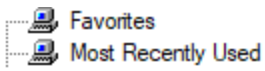
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can elect to **Use extractor mode by default for separate dynamic link desktops**, or to **Run affected property callbacks when variable changes value**.

In the **Component Tree Options** area, options include:

- **Show Favorites** – enables the **Favorites** folder in the **Component Libraries** tree.
- **Show Most Recently Used** – enables the **Most Recently Used** folder in the **Component Libraries** tree. Use the field to specify the number of components shown in the **Most Recently Used** folder. The default is 10.



In the **Search Options** area, options include:

- **Set** – specifies the search set when searching the Component Libraries. Choose to search **All components**, **Current list only**, or **Append to current list**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Load Libraries at Initialization** – this option can slow initialization.

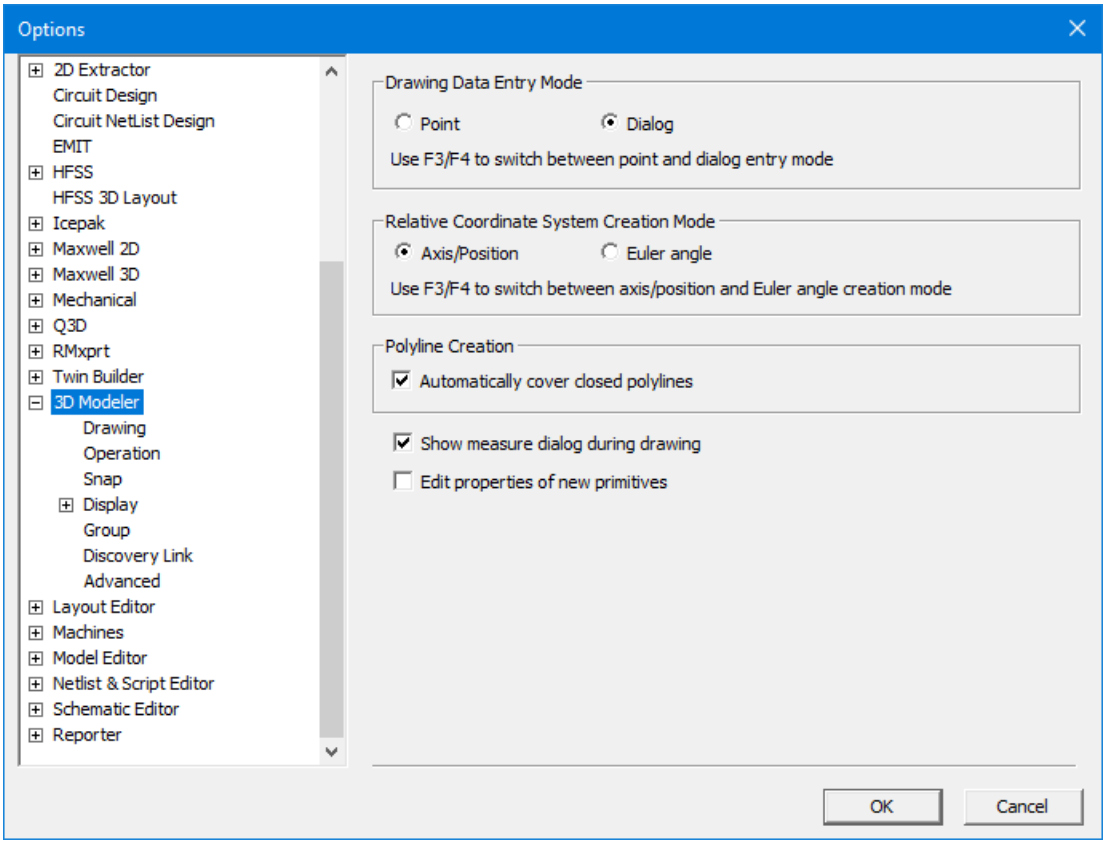
Setting 3D Modeler Options

To set 3D Modeler options in Ansys Electronics Desktop:

1. Click **Tools** > **Options** > **General Options**.

   The **Options** window opens with the Desktop Configuration options selected by default.

   In the left pane, select 3D Modeler to view modeler options:



   In the left pane, select the entries below **General** to display the associated options:

   - Drawing
   - Operation
   - Snap
   - Display
   - Group

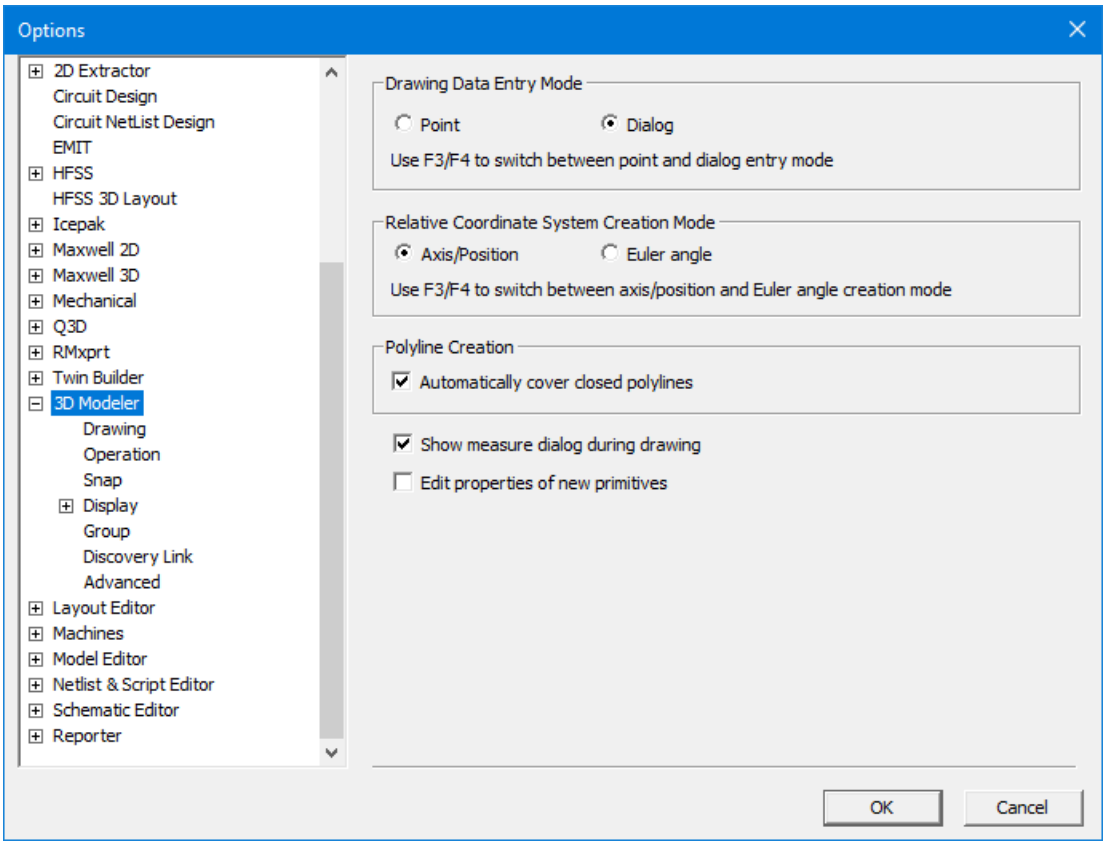
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Discovery Link
- Advanced

2. Click each entry and make the desired selections.
3. Click **OK** to apply your preferences.

3D Modeler Options: Drawing Options

Under 3D Modeler options, **Drawing** options allow you to decide how Electronics Desktop draws objects.



In the **Drawing Data Entry Mode** area, select **Point** to draw new objects directly via the mouse, or select **Dialog** to prompt a **Properties** dialog box that allows you to enter object dimensions. Dialog mode works with the equation-based line and all two- and three-dimensional objects.

In the **Relative Coordinate System Creation Mode** area, select **Axis/Position** or **Euler angle**. This determines how the relative coordinate system is created from the **Modeler** > **Coordinate System** > **Create** > **Relative CS** menu. When **Axis/Position** is selected, relative coordinate systems are created in Draw mode. When **Euler angle** is selected, they are created from values entered in a dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

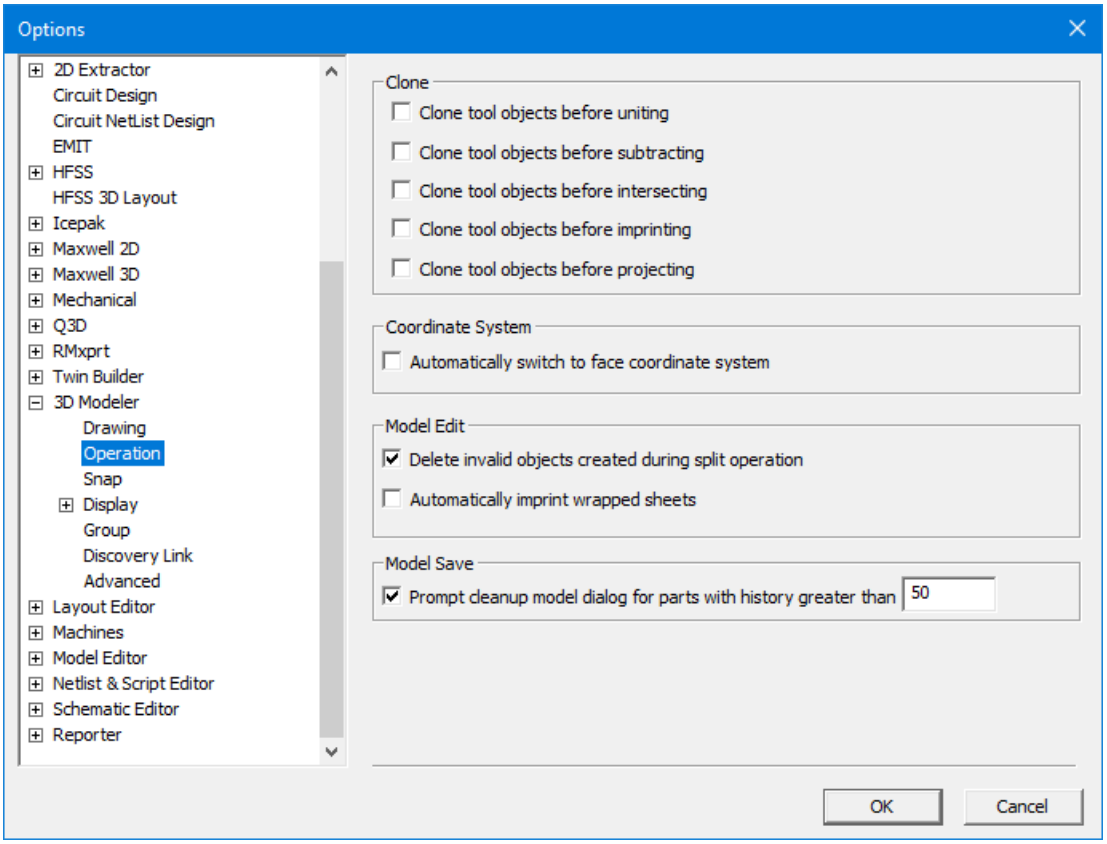
In the **Polyline Creation** area, enable or disable **Automatically cover closed polylines**. When enabled, polylines become sheet objects and are listed as such in the History tree. When deselected, closed polylines are left as uncovered objects and listed as lines in the History tree.

Other options are:

- **Show measure dialog during drawing** – specifies whether a **Measure** dialog box appears upon creation of a new primitive. The dialog box shows the coordinates of the current cursor position.
- **Edit properties of new primitives** – specifies whether a **Properties** dialog box displays when you create a new object in the modeling workspace.

3D Modeler Options: Operation Options

Under 3D Modeler options, **Operation** options allow you to select options for clone, coordinate system, model editing, and model save operations.



In the **Clone** area, enable or disable options. By default, the modeler deletes tool objects when performing operations on objects. You may specify that the modeler make a clone (copy) of the tool object before these operations:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Clone tool objects before uniting**
- **Clone tool objects before subtracting**
- **Clone tool objects before intersecting**
- **Clone tool objects before imprinting**
- **Clone tool objects before projecting**

In the **Coordinate System** area, decide whether or not to **Automatically switch to face coordinate system**.

By default, the modeler operates within the user-selected coordinate system. If this option is enabled, when you select a face and create a new object, the modeler first creates a face coordinate system consistent with the selected face and the new object is created within the face coordinate system. When this option is disabled, you must manually create a Face Coordinate System before creating an object related to it.

In the **Model Edit** area, enable or disable the following options:

- **Delete invalid objects created during split operation** – When selected, the modeler deletes invalid objects created during split operation. When deselected, invalid objects can be freely created.
- **Automatically imprint wrapped sheets** – Imprinting is a boolean operation, whereas wrapping is not.

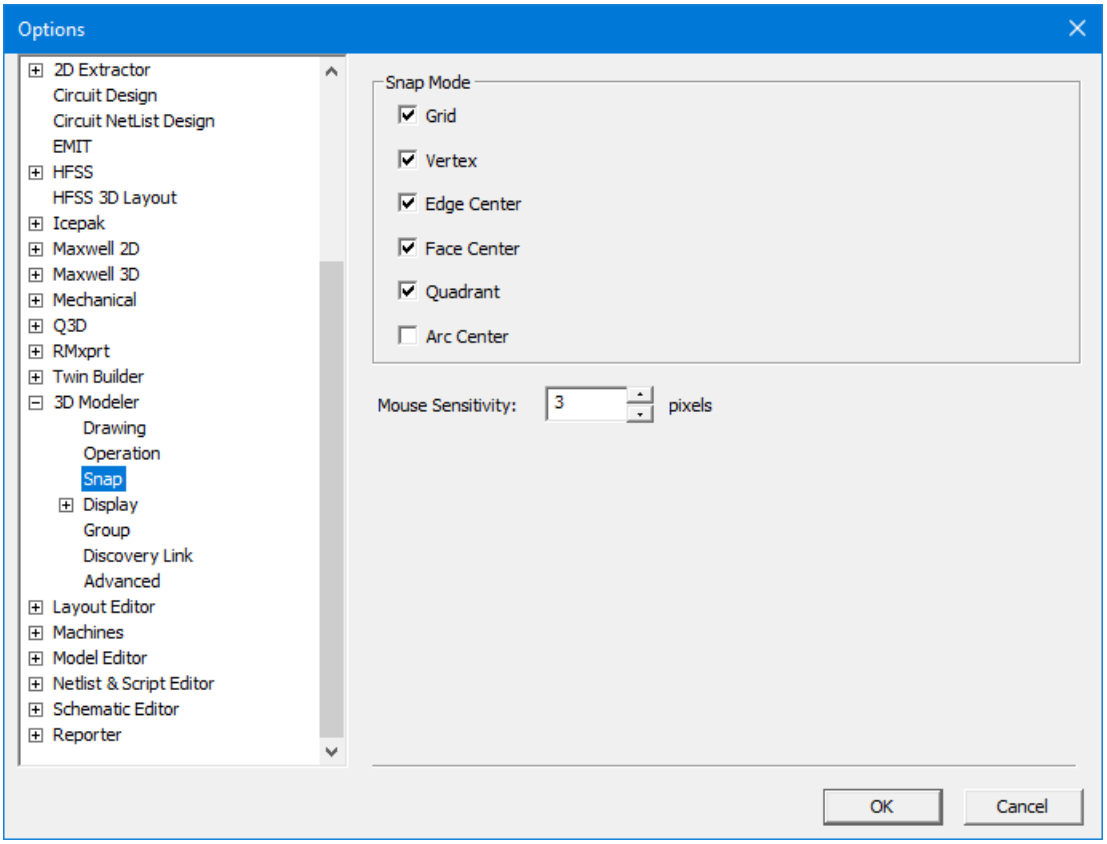
  See: Imprinting an Object and Wrap Sheet Command.

In the **Model Save** area, you can enable/disable being prompted for model history cleanup. You can also specify the number of actions on a part that will cause a prompt. The default value is 50.

3D Modeler Options: Snap Options

Under 3D Modeler options, **Snap** options allow you to decide how Electronics Desktop snaps objects. By default, most objects are set to snap (adhere) to the nearest point on the grid. The coordinates of this point are used, rather than the exact location of the mouse.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the **Snap Mode** area, select the points you would like the cursor to snap to:

- Grid
- Vertex
- Edge Center
- Face Center
- Quadrant
- Arc Center

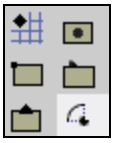
Use the **Mouse Sensitivity** option to select how many pixels you must move the mouse before a different snap point is selected. Values between 1 and 20 are accepted, with 1 being the most sensitive and 20 being the least sensitive. The default is 3.

You can also access snap settings:

- **From the Modeling Workspace** – While in draw mode, right-click and select **Snap Mode**.
- **From the Toolbar** – Select **Modeler** > **Snap Mode**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **From the Draw Ribbon** – Select and deselect the snap icons.



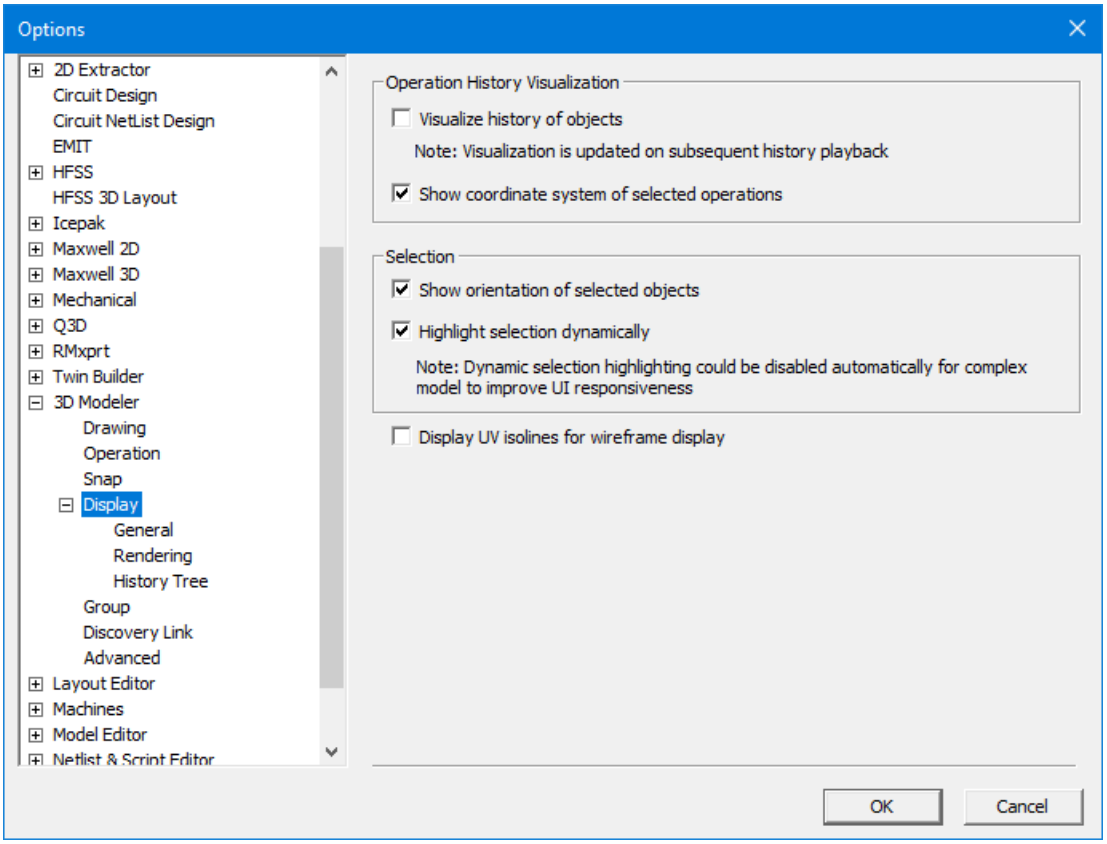
Hover over each icon for a tooltip indicating its equivalent menu option.

> **Important:**
>
> In general, you should select at least one of the snap options . If none is selected, the software is in "free mode" and selects whatever point you click, regardless of its coordinates. This can cause problems creating closed objects because the point you select may appear to be the vertex point of an open object, but may not be the exact coordinates.

3D Modeler Options: Display Options

Under 3D Modeler options, **Display** options allow you to change how Electronics Desktop displays your project tree, history, and objects.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



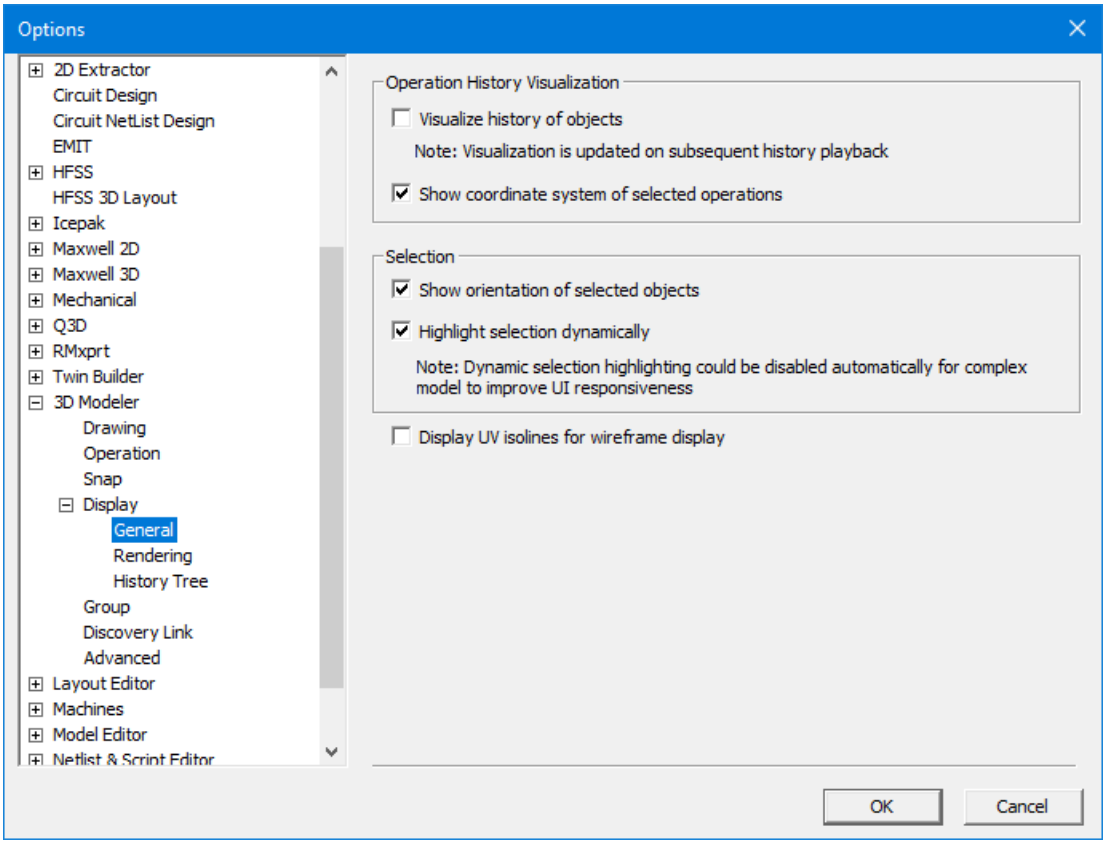
In the left pane, select the entries below **Display** to display the associated options:

- General
- Rendering
- History Tree

Display Options: General Options

Under Display Options, **General** options allow you to change how objects are selected and how their history is visualized.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

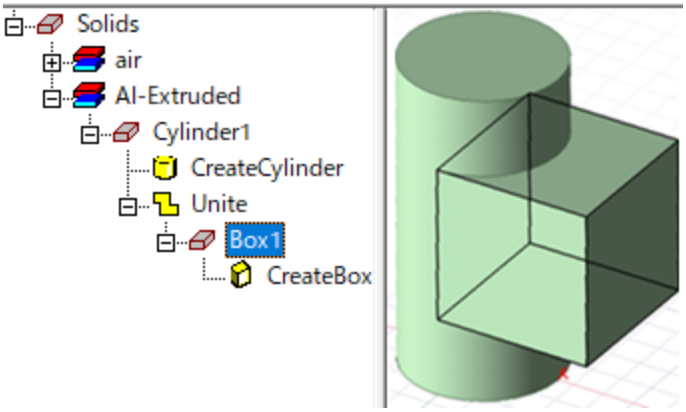


In the **Operation History Visualization** area, set the following options:

- **Visualize history of objects** – When selecting an object in the history tree, this option lets you view an outline of each part that comprises the object. This can help you visualize an object that has been merged with another object. Clearing this selection removes visualization of objects that are part of the model history. For large models, this is faster and uses less memory.

  Visualized history:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> You must restart Electronics Desktop for this option to take effect.

- **Show coordinate system of selected operations** – Select or clear.

In the **Selection** area, set the following options:

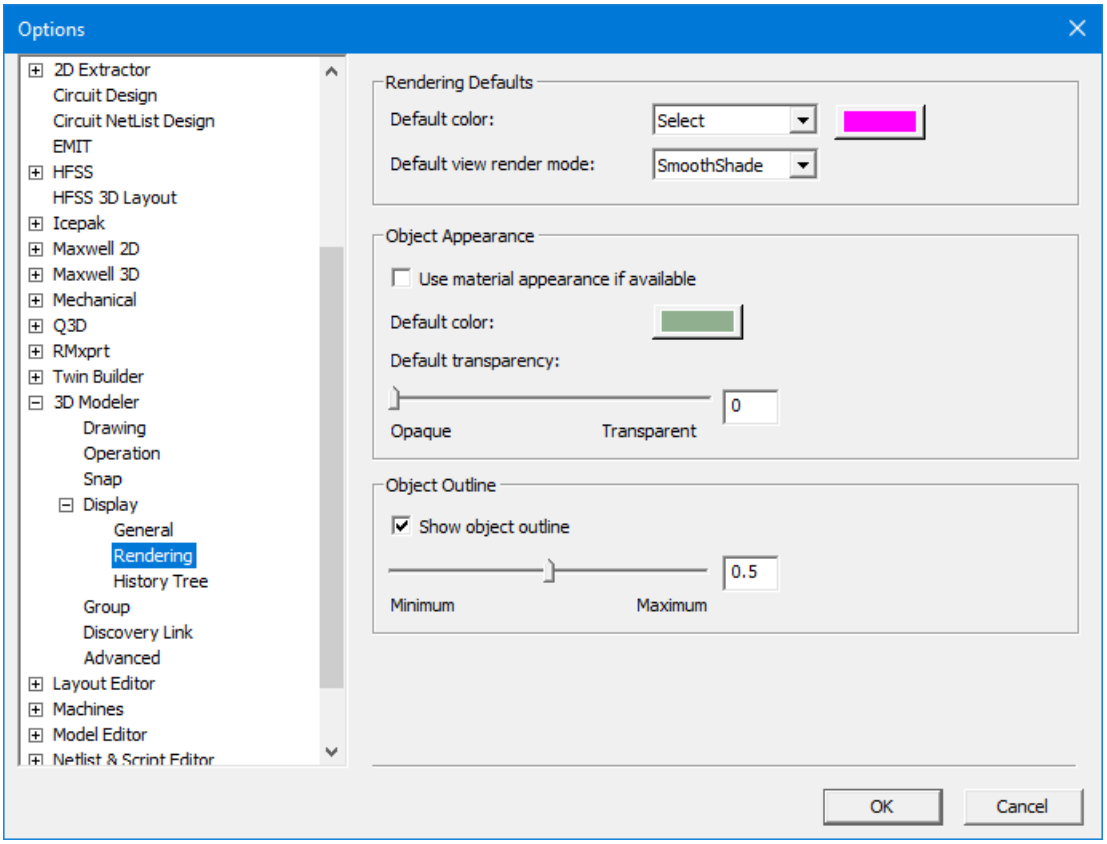
- **Show orientation of selected objects** – Select or clear.
- **Highlight selection dynamically** – This option causes objects or faces (depending on the selection mode) to be highlighted when you pass the mouse pointer over them. You may want to turn this off for complicated models as it can reduce responsiveness. If Electronics Desktop identifies the model to be large enough, dynamic highlighting is automatically ignored. In this case you will see a message about this.

You can also **Display UV isolines for wireframe display**. For models with curved faces, you may prefer to clear this option to simplify the wire-frame display and speed up rendering.

Display Options: Rendering Options

Under Display Options, **Rendering** options allow you to change rendering and object appearance defaults.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the **Rendering Defaults** area, set the following options:

- **Default color** – Use the drop-down menu to choose the **Select**, **Highlight**, or **Rubberband** action. Click the color button to select a color for each action.
- **Default view render mode** – Select either **WireFrame** or **SmoothShade**. When dealing with complicated geometries, choose **WireFrame** rendering for speed.

In the **Object Appearance** area, set the following options:

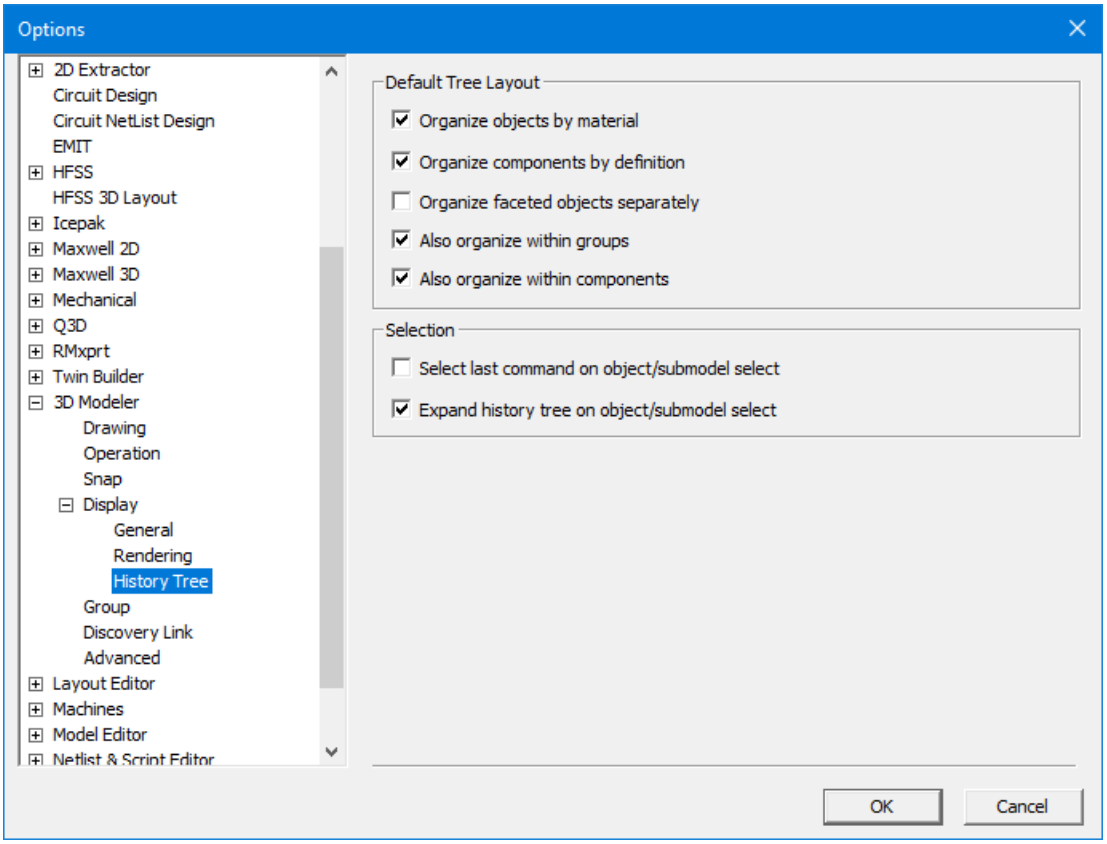
- **Use material appearance if available** – When this option is selected, default color and transparency used for newly created objects come from material settings. Some materials may not have default appearance specifications; in those cases, default color and transparency specified under **Rendering Defaults** are used.
- **Default color** – Click to select the default color.
- **Default transparency** – Use the slider to select the default transparency.

In the **Object Outline** area, select or clear **Show object outline**. If you elect to show the object outline, use the slider to determine the outline thickness. Value must be between 0 and 1.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Display Options: History Tree Options

Under Display Options, **History Tree** options allow you to change the history tree display and selection response.



In the **History Tree Layout** area, change the organization of the history tree as desired:

- **Organize objects by material**
- **Organize components by definition**
- **Organize faceted objects separately**
- **Also organize within groups**
- **Also organize within components**

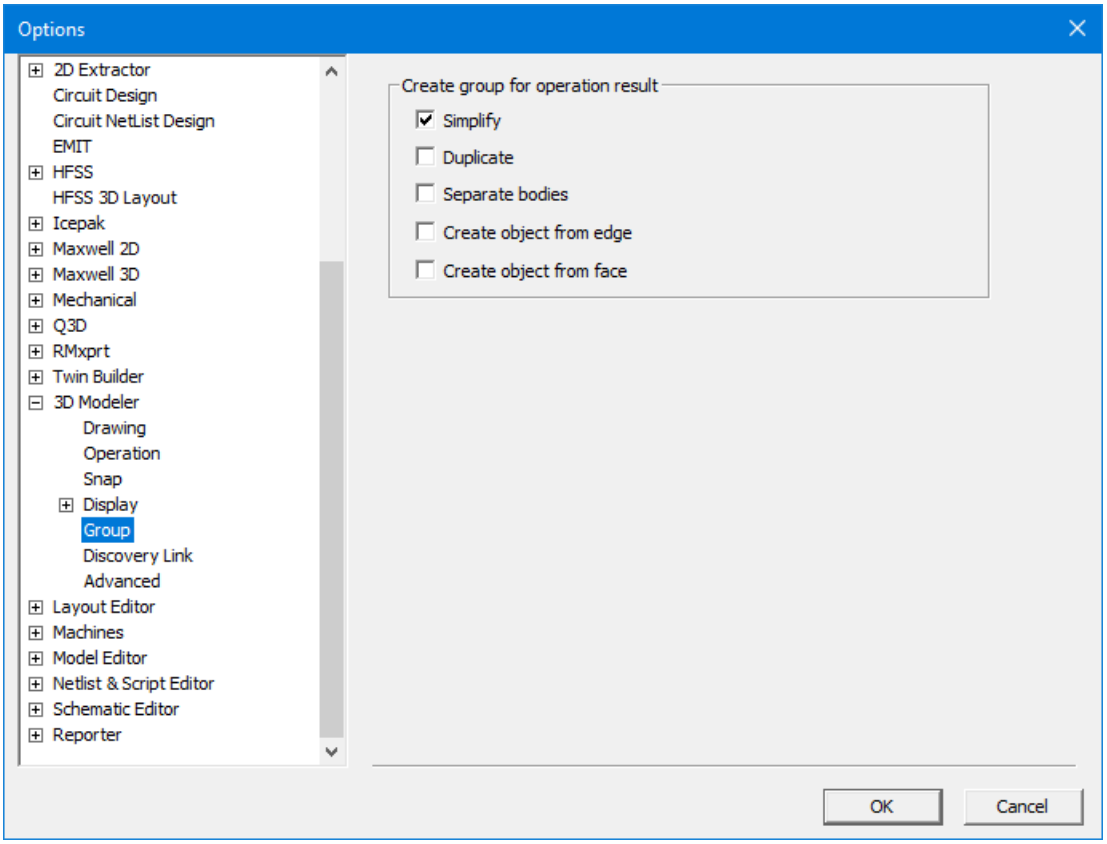
In the **Selection** area, set the following options:

- **Select last command on object/submodel select** – When selected, the history tree is expanded after operations on object properties, even if the tree is collapsed for the item. The command is selected.
- **Expand history tree on object/submodel select** – When selected, selecting an object automatically opens the history tree.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3D Modeler Options: Group Options

Under 3D Modeler options, **Group** options determine how groups are created based on operations.



In the **Create group for operation result** area, select or clear the following actions:

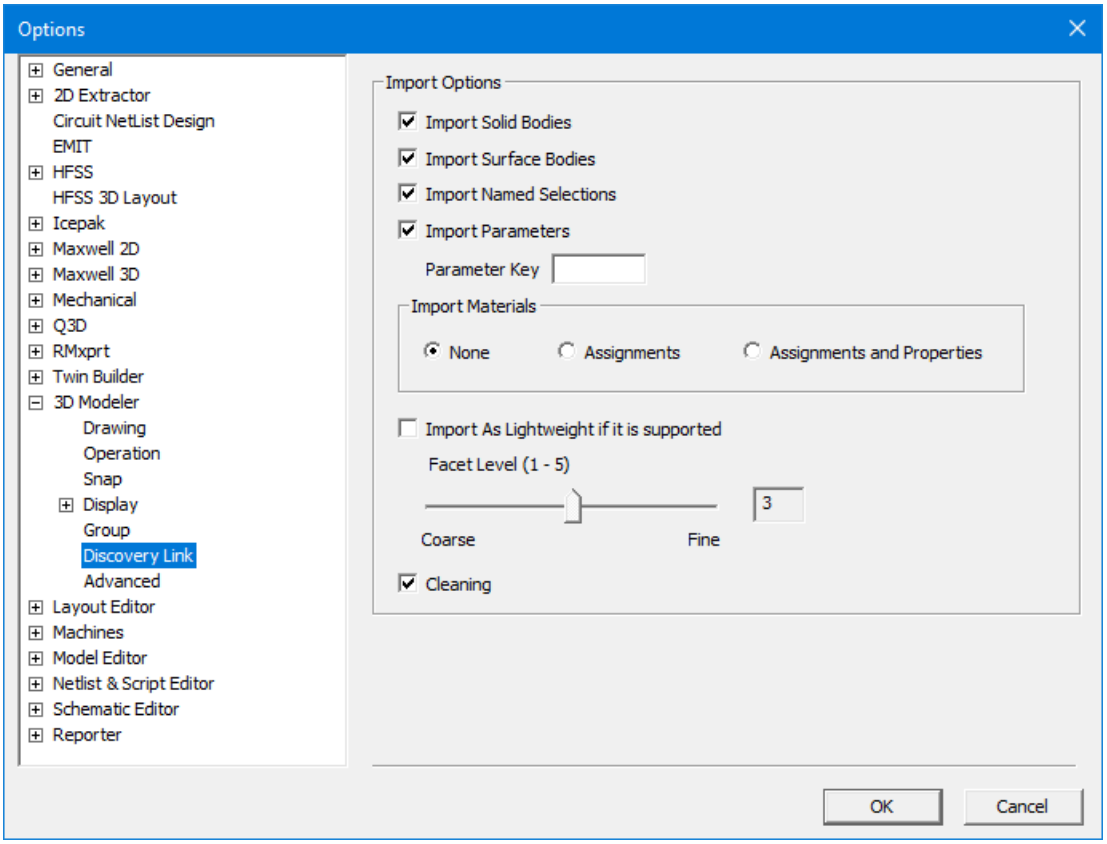
- **Simplify**
- **Duplicate**
- **Separate bodies**
- **Create object from edge**
- **Create object from face**

When any selected action is performed, a group will be created in the History Tree.

3D Modeler Options: Discovery Link Options

Under 3D Modeler options, **Discovery Link** options determine how linked Discovery projects are imported.

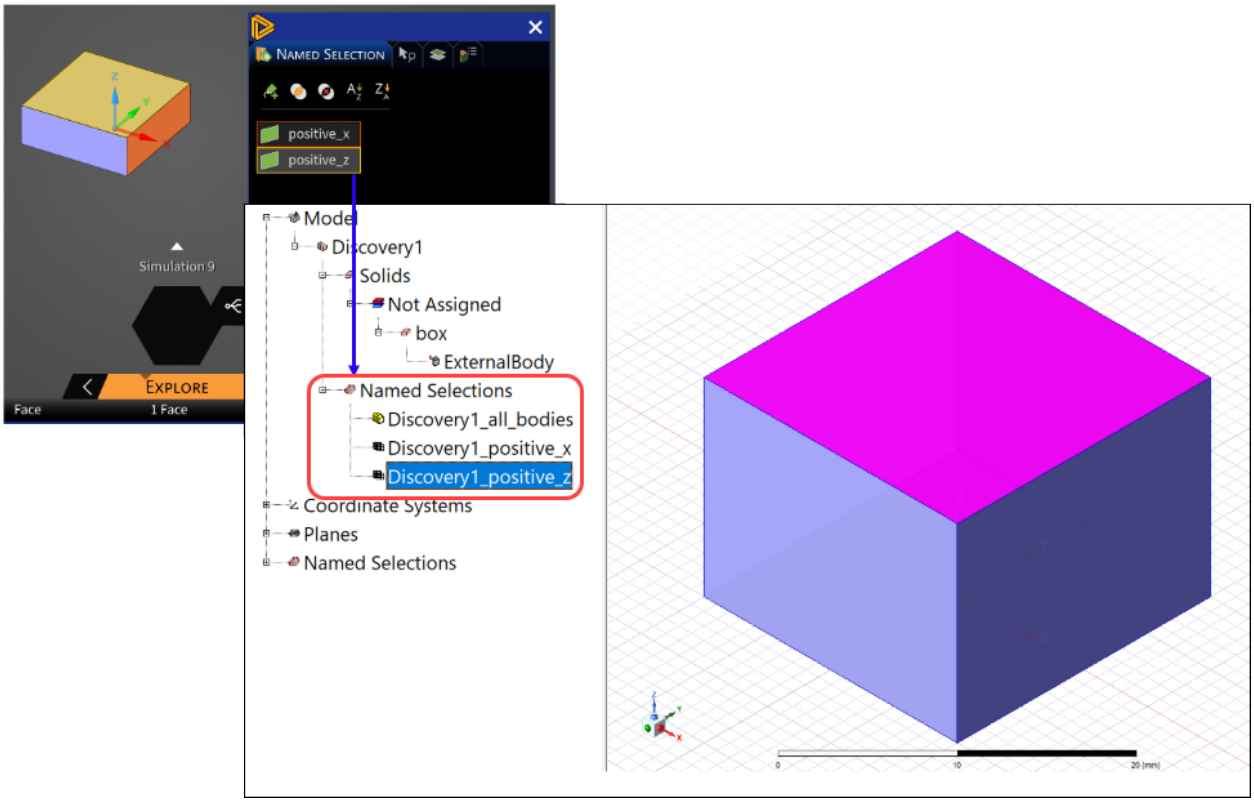
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To specify Discovery link settings, select or clear the following check boxes in the **Discovery Link** section:

- **Import Solid Bodies** – Enabled by default.
- **Import Surface Bodies** – Enabled by default.
- **Import Named Selections** – Import groups of entities. For example, faces can be grouped and named in Discovery, and if this option is checked, the named selections will be preserved during import and listed in the 3D Modeler history tree under a **Named Selections** branch.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Tip:** Named selections can be used to preserve entity IDs during parametric variations. See 3D Modeler Discovery Link for more details.

- **Import Parameters** – Enabled by default.
  - **Parameter Key** – A string used to filter out parameters. By default, no string is entered and all Discovery parameters are imported.
- **Import Material Assignments** – **None** by default.
  - **Assignments** – Imports the material names without the material properties. The material properties will be obtained from a material with the same name that exists in the user's project. With this option, the system will first search for the material within the project's materials. If no matching material is found, the search will extend to the sys libraries. (See Working with Material Libraries for details on libraries.) If a material with the same name exists in a sys library, it will be imported into the project from the library. If a material with the same name cannot be found in the project, a violation will be issued after the Discovery link is created.
  - **Assignments and properties** – Imports material with its material properties. A project material will be created in AEDT with imported properties. Note that if a

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

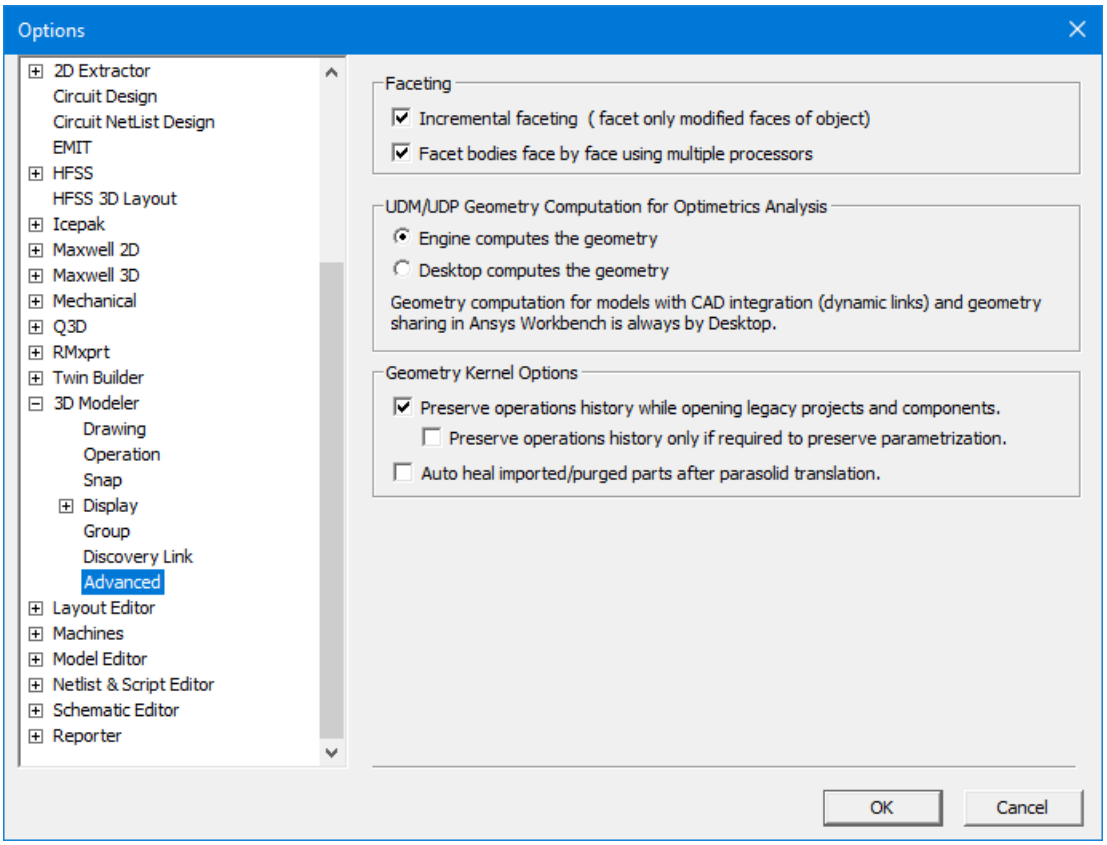
project material with same name already exists, then it will be used instead of a new material being created.

> **Note:** Discovery currently only supports exporting mechanical and thermal properties, so the **Assignment with properties** option is only supported in Mechanical and Icepak designs. For HFSS, Maxwell, and Q3D designs, this option will be ignored.

- **Import As Lightweight if it is supported** – This feature allows you to import a geometry from Discovery as a lightweight geometry into a design with SBR+ solution type. The Facet Level can be set on a 1-5 range, from coarse to fine, if the geometry in Discovery is not faceted. If it is faceted in Discovery, the facets will be imported directly and the Facet Level setting will be ignored.

3D Modeler Options: Advanced Options

Under 3D Modeler options, **Advanced** options help configure faceting and UDM/UDP.



In the **Faceting** area, select **Incremental faceting** or **Facet bodies face by face using multiple processors**. Face by face will be slower.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the **UDM/UDP Geometry Computation for Optimetrics Analysis** area, select whether **Engine** or **Desktop** computes the geometry.

In the **Geometry Kernel Options**, enable **Preserve operations history while opening legacy projects** to preserve operations history while opening legacy projects that used the ACIS kernel. When opening legacy projects, the conversion is a one-time event. A renamed copy of the original file is preserved. The default is to preserve operations history. If the option is disabled, the operations history is purged. The option to purge history is available because:

- Some operations cannot be replayed in Parasolid.
- Purging provides backup to translate geometry.
- Parameterization is lost.
- Expect a better conversion success rate with purged models.

Setting Maxwell 2D Options

To set Maxwell 2D options:

1. Click **Tools** > **Options** > **General Options** to open the **Options** window, displaying a hierarchical list of options.
2. Click the **+** button next to **Maxwell 2D** to show the following choices: **General**, **Solution Type**, **Material Threshold**, **Boundary Assignment**.
3. On the **General** panel, select or clear the following check boxes:
    - **Save before solving** – Note that when you enable this setting, the project is only saved if it has been modified since its last save.
    - **Save Optimetrics field solutions**
    - **Apply variation deletions immediately**
    - **Generate model history when importing legacy 2D projects**
4. On the **Solution Type** panel:
    - To change the default solution type when you initially insert a design, select one of the following from the **Default solution type** drop-down menu: **Magnetostatic**, **Electrostatic, AC Magnetic**, **Transient**, **DC Conduction**, **AC Conduction**.
    - To change the default geometry mode when you initially insert a project, select one of the following from the **Default geometry mode** drop-down menu: **XY**, **about Z**.
5. On the **Material Threshold** panel, enter the **Default perfect conductor** and **Default insulator/conductor** values in siemens/m.

> **Note:** Setting the material threshold affects the default setting for the current and all future projects/designs. To change the material threshold for the current design only, use the **Maxwell** > **Design Settings** window to change the material thresholds on the **Set Material Thresholds** tab.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. On the **Boundary Assignment** panel, select or clear the following check boxes:
    - **Use Wizards for data input when creating new boundaries** – When this is checked, the creation of boundaries and excitations use Wizards to guide you through the process. When this is not checked, the creation of boundaries and excitations displays a Properties dialog with tabs for different kinds of information.
    - **Duplicate boundaries/mesh operations with geometry** – When this is checked, you can duplicate a boundary or excitation when its geometry is pasted or duplicated. See Duplicating Boundaries and Excitations.

Setting Maxwell 3D Options

To set Maxwell 3D options:

1. Click **Tools** > **Options** > **General Options** to open the **Options** window, displaying a hierarchical list of options. Click the **+** button next to **Maxwell 3D** to show the following choices: **General**, **Solution Type**, **Material Threshold**, **Boundary Assignment**.
2. On the **General** panel, select or clear the following check boxes:
    - **Save before solving –** Note that when you enable this setting, the project is saved only if it has been modified since its last save.
    - **Save Optimetrics field solutions**
    - **Apply variation deletions immediately**
3. On the **Solution Type** panel:
    - To change the default solution type when you initially insert a design, select one of the following from the **Default solution type** drop-down menu: **Electrostatic**, **Magnetostatic**, **AC Magnetic**, **Transient**, **DC Conduction**, **ElectroDCConduction**, **Electric Transient**, **TransientAPhiFormulation**, **EddyCurrentAPhiFormulation**.
4. On the **Material Threshold** panel, enter the **Default perfect conductor** and **Default insulator/conductor** values in siemens/m.

    > **Note:** Setting the material threshold affects the default setting for the current and all future projects/designs. To change the material threshold for the current design only, use the **Maxwell** > **Design Settings** window to change the material thresholds on the **Set Material Thresholds** tab.

5. On the **Boundary Assignment** panel, select or clear the following check boxes:
    - **Use Wizards for data input when creating new boundaries** – When this is checked, the creation of boundaries and excitations use Wizards to guide you through the process. When this is not checked, the creation of boundaries and excitations displays a Properties dialog with tabs for different kinds of information.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Duplicate boundaries/mesh operations with geometry** – When this is checked, you can duplicate a boundary or excitation when its geometry is pasted or duplicated. See Duplicating Boundaries and Excitations.
- **Visualize boundaries on geometry –** When checked, enables visualization of boundaries on geometry.

Setting RMxprt Options

To specify settings for RMxprt options:

1. Click **Tools** > **Options** > **General Options** to open the **Options** window, displaying a hierarchical list of options.
2. Click the **+** button next to **RMxprt** to show the following choices: **General**, **Machine Type**, **Threshold**.
3. In the **General** options panel, select or clear the following check boxes:
   - **Save before solving** – Note that when you enable this setting, the project is only saved if it has been modified since its last save.
   - **Apply variation deletions immediately**
4. On the **Machine Type** panel, to change the default machine type when you initially insert a design, select one of the following from the **Default machine type** drop-down menu:
   - Three-Phase Induction Motor
   - Single-Phase Induction Motor
   - Three-Phase Synchronous Machine
   - Brushless Permanent-Magnet DC Motor
   - Adjust-Speed Synchronous Machine
   - Permanent-Magnet DC Motor
   - Switched Reluctance Motor
   - Line-Start PM Synchronous Motor
   - Universal Motor
   - DC Machine
   - Claw-Pole Synchronous Machine
   - Three-Phase Non-Salient Synchronous Machine
   - Generic Rotating Machines
5. On the **Threshold** panel, enter the **Default conductivity** and **Default permeability** values in siemens/m.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** Setting the material thresholds under **Tools** > **Options** impacts the default setting for the current and all future projects/designs. To change the material threshold for the current design only, use the **RMxprt** > **Design Settings** window to change the material thresholds on the **Set Material Thresholds** tab.

6. Click **OK** to close the window.

Setting Reporter Options

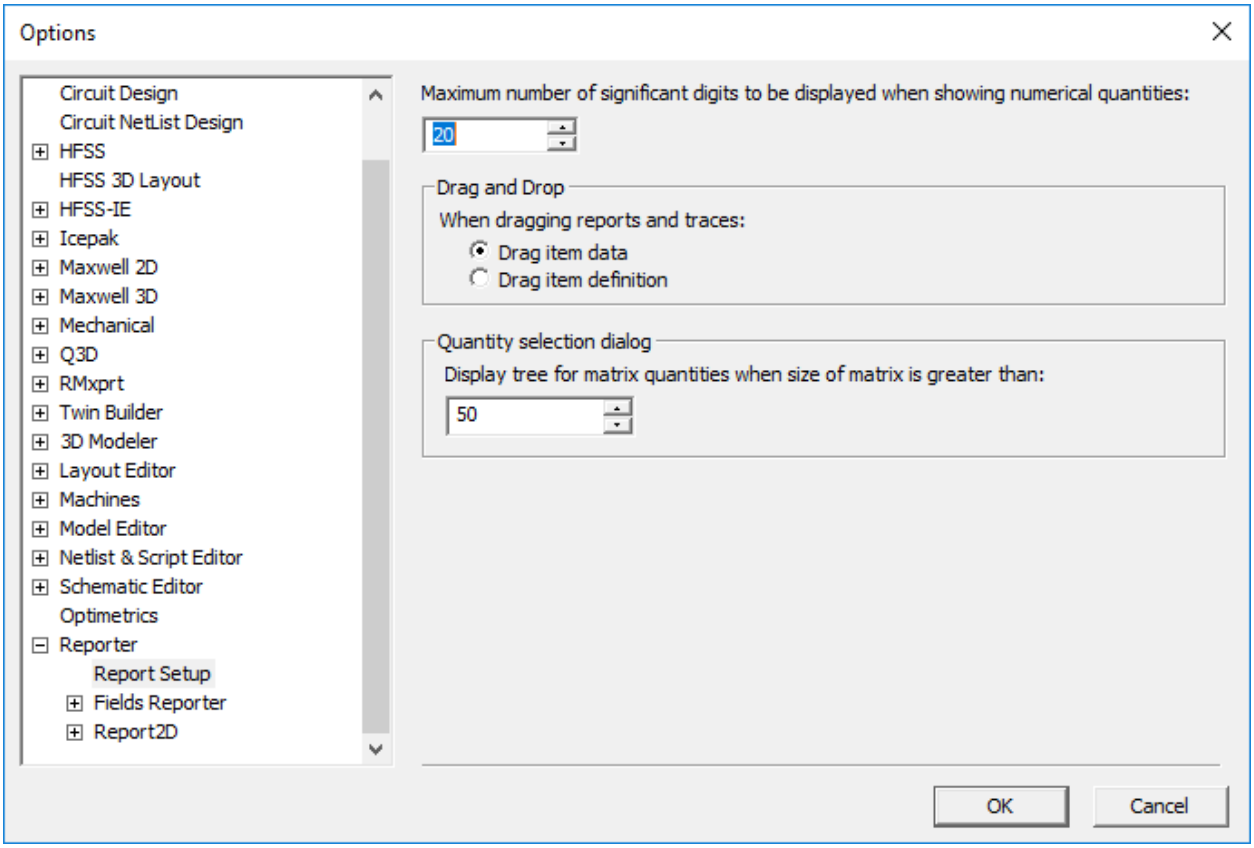
To set Reporter options in Ansys Electronics Desktop:

1. Click **Tools** > **Options** > **General Options**.
2. In the left pane, select **Reporter**.
3. Select the entries below **Reporter** to display the associated options:
   - Report Setup
   - Fields Reporter
   - Report2D
4. Click each entry and make the desired selections.
5. Click **OK** to apply your preferences.

Reporter Options: Report Setup Options

Under Reporter Options, **Report Setup** options allow you to customize significant digits, drag-and-drop, and quantity selection options for reports.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Set the **Maximum number of significant digits to be displayed when showing numerical quantities**. The value must be between 0 and 20.

In the **Drag and Drop** area, select how Electronics Desktop behaves when you drag reports and traces. You can either **Drag item data** or **Drag item definition**.

In the **Quantity Selection Dialog** area, select a **Display tree for matrix quantities when size of matrix is greater than** value. When the number of matrix elements is larger than the selected number, the Quantities field uses a tree structure to divide matrix quantities into groups by their first element name. The initial display shows groups, without initially listing group members. This is useful when dealing with large matrices.

Reporter Options: Fields Reporter Options

Under Reporter Options, **Fields Reporter** options fall into three categories:

- Animation Options
- Mesh Plot Options
- Streamline Plot Options

In the left pane, select a category to view those options.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

From the **Animation** options, you can set the following:

- **Group Field Overlays by Type** – If selected, field overlays are grouped in the Project Tree. If deselected, they are not.
- **Default Phase Animation Settings** – Select the default start and stop angles as well as the number of steps for Scalar and Vector plots.

From the **Mesh Plot** options, you can set the **Clipping of volume mesh plot**. When dragging a clip plane, the plot can update automatically. Select **Never** to disable this feature, **Always** to enable it at all times, or set a maximum number of mesh elements for automatic update. This option allows you to update plots for smaller meshes but avoid automatically updating larger plots that may consume too much memory.

From the **Streamline Plot** options, set **Streamline drawing stopping criteria** and **Streamline marker spacing**.

Reporter Options: Report2D Options

Under Reporter Options, **Report2D** options fall into several categories:

- Curve
- Axis
- Grid
- Header
- Note
- Legend
- Marker
- Marker Table
- X/Y Markers
- Stacked
- Digital
- General
- Table

In the left pane, select a category to view those options.

Report2D Options: Curve

Curve settings change how curves are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Curve**:

- **Line Style** – use the drop-down menu to select from: Solid, Dot, ShortDash, DotShortDash, Dash, DotDash, DotDot, DotDotDash, and Long Dash.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Color** – double-click a color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Width** – set the line width by editing the real value in the text field.
- **Arrows** – select the check box to use arrows on curve ends.
- **Show Symbol** – select the check box to have symbols mark the locations of data points on the curve.
- **Sym Freq** – set the symbol frequency by editing the integer value in the text field.
- **Sym Style** – use the drop-down menu to select a symbol from: Box, Circle, Vertical Ellipse, Horizontal Ellipse, Vertical Up Triangle, Vertical Down Triangle, Horizontal Left Triangle, and Horizontal Right Triangle.
- **Fill Sym** – select the check box to set the symbol display as solid. Otherwise it will display as hollow.
- **Sym Color** – double-click a color box to display the **Color** dialog box. Select a default or custom color and click **OK**.

Report2D Options: Axis

Axis settings change how axes are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Axis**:

- **Axis Name** – this describes the axis to which a row's settings apply. You cannot change this field.
- **Color** – double-click a color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Font Color** – double-click a color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Edit Font** – click **Edit Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Min Gutter %** – sets the amount of empty space (gutter) around the axis.
- **Font Description** – describes the settings applied from the **Font** window. You cannot change this field. To change the font, use **Edit Font**.
- **Axis Scaling** – Linear or Log. The default is linear.

Report2D Options: Grid

Grid settings change how grids are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Grid**:

- **Grid Name** – this describes the grid to which a row's settings apply. You cannot change this field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Line Style** – use the drop-down menu to select from: Solid, Dot, ShortDash, DotShortDash, Dash, DotDash, DotDot, DotDotDash, and Long Dash.
- **Line Color** – double-click a color box to display the **Color** dialog box. Select a default or custom color and click **OK**.

Report2D Options: Header

Header settings change how headers are displayed in reports. You can separately style the Title and Subtitle.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Header**:

- **Color** – click a color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Font** – click **Edit Title Font** or **Edit Subtitle Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Company Name** – Enter a company name to appear on all reports.
- **Show Ansys Logo** – Remove the check from the box to hide the Ansys logo that appears on all reports (checked by default).

Report2D Options: Note

Note settings change how notes are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Note**:

- **Note Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Note Font** – click **Edit Note Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Background Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Background Visibility** – select the check box to make the background color visible. Deselect to make it transparent.
- **Border Line Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Border Visibility** – select the check box make the note border visible. Deselect to make it transparent.
- **Border Line Width** – set the line width by editing the real value in the text field.

Report2D Options: Legend

Legend settings change how legends are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Legend**:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Legend Name** – Default is no name. When non-empty, a header row for the Legend in plot shows up with that string.
- **Show Trace Name** – select the check box to show the trace name; deselect it to hide the trace name.
- **Show Solution Name** – select the check box to show the solution name; deselect it to hide the solution name.
- **Show Variation Key** – select the check box to show the variation key; deselect it to hide the variation key.
- **Highlight Curve on Hover** – select the check box to highlight a curve when you hover the cursor over it; deselect it to leave the curve as-is when you hover the cursor.
- **Text Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Text Font** – click **Edit Text Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Background Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Border Line Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Border Line Width** – set the line width by editing the real value in the text field.
- **Grid Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.

You can set different **Text Color** and **Text Font** settings for the Header Row.

Report2D Options: Marker

Marker settings change how markers are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Marker**:

- **Marker Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Marker Font** – click **Edit Marker Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Arrow Direction** – use the drop-down menu to set the arrow direction to Up, Down, Left, or Right.

Report2D Options: Marker Table

Marker Table settings change how marker tables are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Marker Table**:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Text Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Text Font** – click **Edit Text Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Background Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Border Line Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Border Line Width** – set the line width by editing the real value in the text field.
- **Grid Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Grid Line Width** – set the line width by editing the real value in the text field.

You can set a different **Text Color** and **Text Font** for the header row.

Report2D Options: X-Y Markers

X-Y Marker settings change how X-Y markers are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **X-Y Markers**:

Background Colors

- **Marker [#] Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.

Properties

- **On-screen intersection** – select the check box to enable on-screen intersection; deselect to disable.
- **Marker Font** – click **Edit Marker Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Text Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Line Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Line Style** – use the drop-down menu to select from: Solid, Dot, ShortDash, DotShortDash, Dash, DotDash, DotDot, DotDotDash, and Long Dash.
- **Line Width** – edit the text field to specify a line width.
- **Show Name** – select the check box to show X-Y marker names; deselect to hide.
- **Snap to Vertex** – select the check box to snap markers to the vertex; deselect to disable snapping.

Inter Marker Deltas

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Show Delta** – select the check box to show inter-marker deltas; deselect to hide.
- **Delta Font** – click **Edit Delta Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Delta Text Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Line Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Line Style** – use the drop-down menu to select from: Solid, Dot, ShortDash, DotShortDash, Dash, DotDash, DotDot, DotDotDash, and Long Dash.
- **Line Width** – edit the text field to specify a line width.

X Value Manual Format

The default number format is Auto. When number format is auto, X-value number format corresponds to X-axis number format with extra 2 precision.

- X Value Number format. Can be Auto, Decimal, Scientific, Integer.
- X Value Field Width. Default 3.
- X Value Field Precision, Default 2.

Y Value Manual Format

The default number format is Auto. When number format is auto, Y-value number format corresponds to Y-axis number format with extra 2 precision.

- Y Value Number format. Can be Auto, Decimal, Scientific, Integer.
- Y Value Field Width. Default 3.
- Y Value Field Precision, Default 2.

Report2D Options: Stacked

Stacked settings change how stacked plots are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Stacked**:

- **Auto Fit Mode** – select the check box to enable auto fit; deselect to turn off autofit.
- **Stack Height in Pixels** – set the default stack height.
- **Curve Grouping Strategy** – can be **Single**, **By Trace**, or **By Units**. **Single** means that a new stacked plot shows a single curve per stack. **By Trace** means that all curves are grouped by their trace. **By Units** means that all curves are grouped by their unit type.

If you change the **Curve Grouping Strategy** set by default, existing stacked plots remain unaffected. The new default will apply only to new stacked plots.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Report2D Options: Digital

Digital settings change how digital stack heights are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Digital**:

- **Digital Literal Foreground** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Expand Arrays/Records** – select the check box to automatically expand arrays and records; deselect it to disable.
- **Digital Stack Height in Pixels** – enter the stack height for Analog, Digital, Enum, Event, and Literal stacks.

Report2D Options: General

These settings contain general report options.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **General**:

- **Background Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Plot Area Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Highlight Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Accumulate Depth** – enter a value; the default is 4.
- **Enable Y Axis Stripes** – select the check box to add stripes to the Y axis; deselect to remove stripes.
- **Auto Scale Fonts** -on by default and when enabled scales text in plots and colorkey (contour plot, field plots in 3D modeler) for high resolution screens.

Curve Tooltip

- **Show Trace Name** – select the check box to display trace names in the tooltip when hovering the cursor over a curve.
- **Show Variation Key** – select the check box to display the variation key in the tooltip when hovering the cursor over a curve.
- **Show Solution Name** – select the check box to display the solution name in the tooltip when hovering the cursor over a curve.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clipboard Option

- **Capture Aspect Size Ratio** – select either **As Shown** or **Full Screen**.
- **Capture Background Color** – select either **As Shown** or **White**.

Format

- **Field Width** – enter a field width value.
- **Precision** – enter a precision value.
- **Use Scientific Notation** – select the check box to use scientific notation; deselect to disable scientific notation.

Report2D Options: Table

Table settings change how tables are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Table**:

- **Rows Per Page** – enter the number of rows you would like to display per page; the default is 2500.
- **Text Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Text Font** – click **Edit Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Border Width** – enter a value for the border width.
- **Border Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Grid Width** – enter a value for the grid width.
- **Grid Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Background Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Page Link Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Arrow Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.

Header Row

- **Text Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Text Font** – click **Edit Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Background Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.

Format

- **Field Width** – enter a value for the field width.
- **Precision** – enter a value for the precision.
- **Use Scientific Notation** – select the check box to enable scientific notation; deselect it to disable scientific notation.

Copy to Clipboard

- **With Header** – select to include header when copying the table to a clipboard; deselect to remove it.
- **With Tab Separator** – select to include tab separator when copying the table to a clipboard; deselect to remove it.

Setting HPC and Analysis Options

All analysis parameters are accessed via a single window. The machine list and options settings have been integrated into analysis configurations. The default configuration is for solving on a single, local machine. You can create many analysis configurations for remote and distributed solutions, and switch between them depending on the job being solved. Multiprocessing has been integrated into the machine lists.

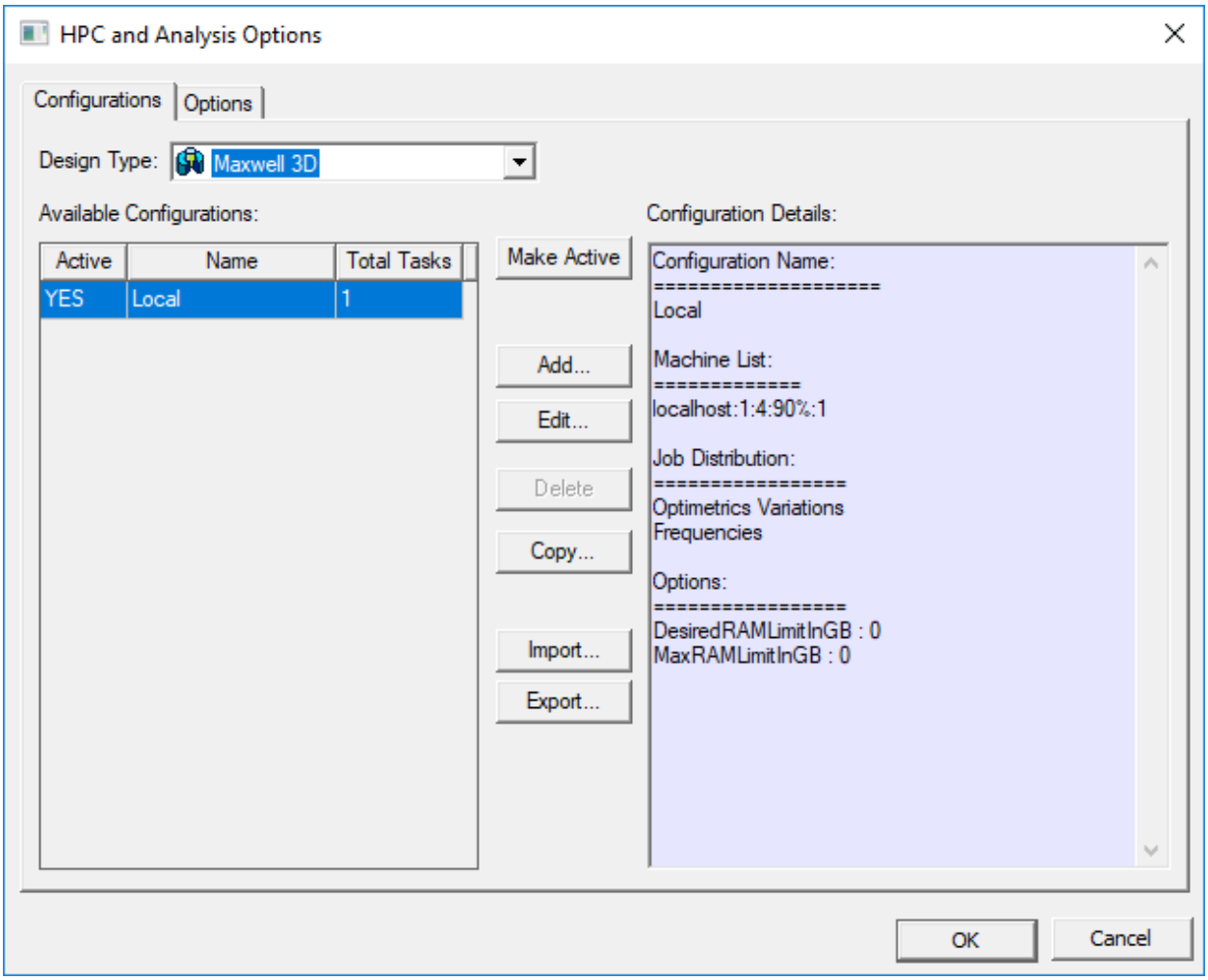
To set HPC and Analysis Options:

1. Click **Tools** > **Options** > **HPC and Analysis Options**.

   You can also access HPC and Analysis Options using the **HPC Options** icon on the **Simulation** ribbon.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **HPC and Analysis Options** window appears, displaying the **Configurations** tab.



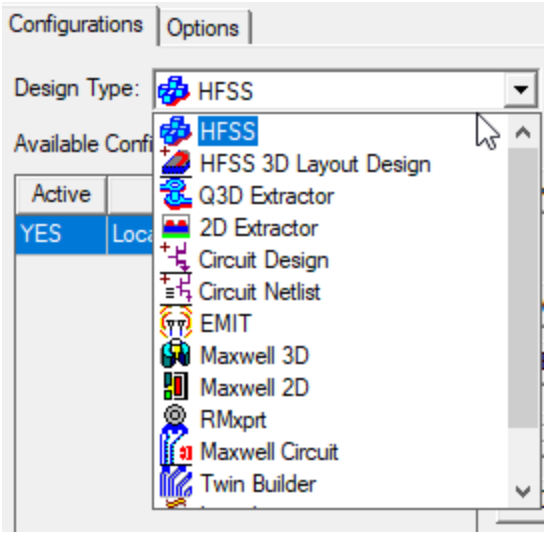
Configurations Tab

Available Configurations are described by **Name**, whether or not they are **Active**, and the **Total Tasks** the configuration can execute.Selecting a configuration from the list displays the details of that configuration in the **Configuration Details** panel.

From the **Configurations** tab, select the **Design Type** to display a list of available configurations for that type.

Configurations must be defined for all design types separately. To use similar analysis parameters for different design types, create separate analysis configurations for each design type. The active configuration is used when solving an analysis for that design type.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Selecting an Available Configuration

To activate a configuration, select it from the **Available Configurations** list and click **Make Active**. The active configuration will be indicated by a **YES** in the **Active** column.

Additional options include:

- **Add** – launches a dialog box to create a new analysis configuration.
- **Edit** – launches a dialog box to edit the currently selected analysis configuration.
- **Delete** – deletes the currently selected analysis configuration(s).

> **Note:**
>
> You cannot delete the Local configuration.

- **Copy** – creates a new analysis configuration, and launches a dialog box to edit it. If the dialog box is canceled, the new analysis configuration is not created.
- **Import** – allows you to import an *.acf file to create an analysis configuration.

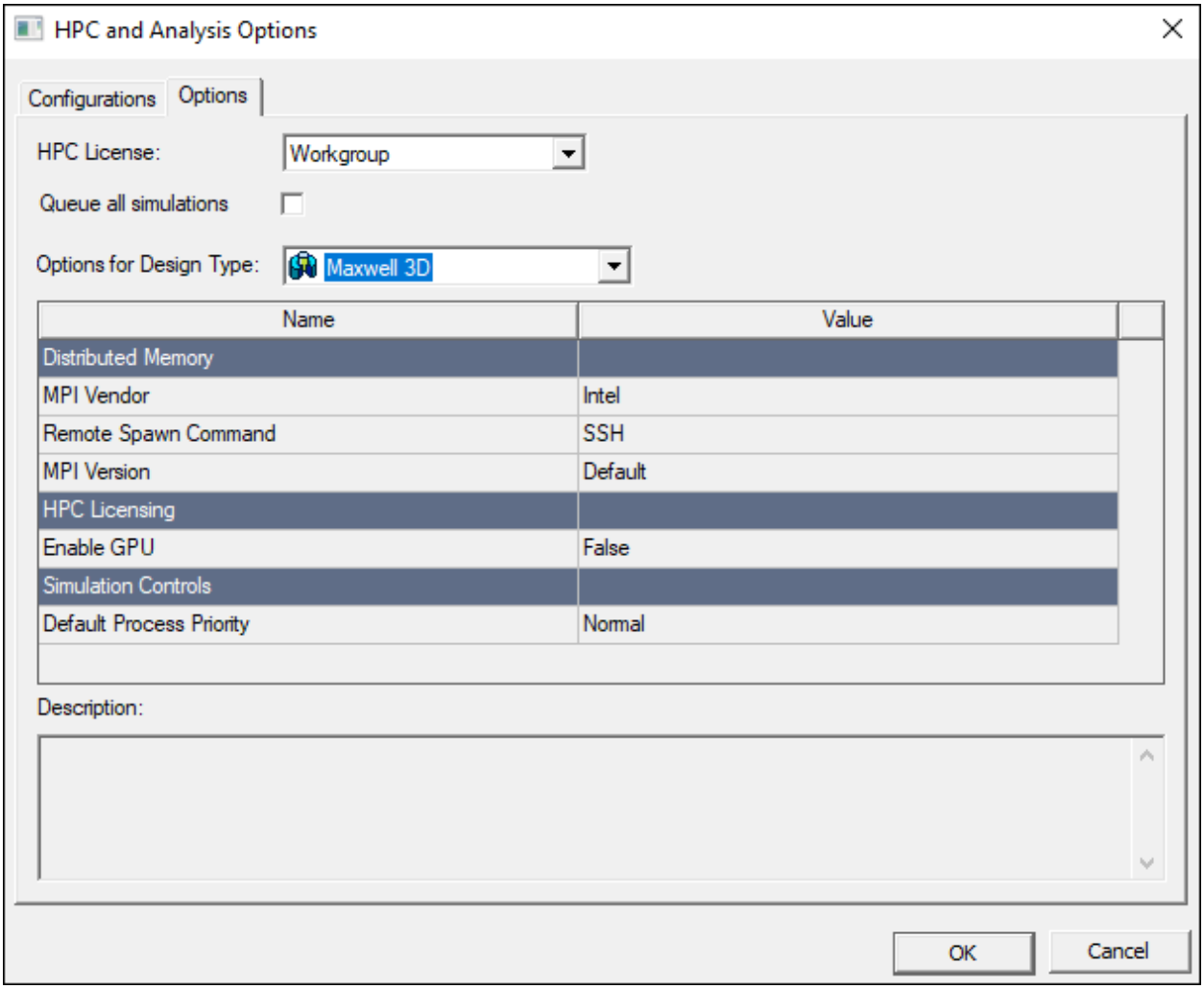
> **Note:**
>
> Importing analysis configurations always adds the imported analysis configurations to the current design type. If there is a name conflict between an imported analysis configuration and an existing analysis configuration, the imported configuration is renamed and a notification appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Export** – allows you to export the selected analysis configurations to an *.acf file. You can then import the configurations into a different design type, or import them on a different machine.

Options Tab

The **Options** tab in the **HPC and Analysis Options** dialog box contains general and product-specific settings.



Important:

Available options vary by design type. Certain items discussed on this page may not be visible, depending on the currently selected design type.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These options are not specified for, or saved as part of, the current analysis configuration. Instead, they are global and are always in effect for the given design type when both of the following conditions are true:

- A design of the matching design type is being solved.
- You have not specified corresponding overriding batch options on the command line.

From the **Options** tab, you can:

- Choose the HPC license type
- Enable queuing
- Specify Distributed Memory settings (for example, MPI for certain solvers)
- Set licensing options
- Enable GPU (for Transient, Matrix and SBR+ solves)
- Set the Default Process Priority

> **Note:**
>
> Solving with Intel MPI on a single Windows machine does not require MPI installation, but registration of the user's password with Intel MPI on the host is required if using password authentication. Solving with Microsoft MPI on a single Windows machine requires Microsoft MPI to be installed and the MS-MPI Launch Service to be started. These steps are not required for Microsoft MPI analysis on a Windows HPC cluster. Solving with Microsoft MPI on multiple Windows hosts is only supported on a Windows HPC cluster.
>
> Users running on Linux do not need to install MPI manually.

For **HPC License**, select **Auto**, **Workgroup** or **Pack**.

HPC licensing enables the use of cores and GPUs to accelerate simulations. In general, each core requires one unit of HPC, while each GPU requires eight units. The selected HPC license type determines which license is used, and how units of HPC are converted to license counts.

- **Workgroup** (formerly "pool") – One HPC workgroup license enables one unit of HPC.
- **Pack** – One HPC pack license enables eight units of HPC. Additional packs multiply by four, enabling 32, 128, 512,... , in the context of a single simulation.
- **Auto** – delegate the choice of Workgroup or Pack licensing to the Ansys Licensing Settings tool which is a separate application installed along with Electronics Desktop. When set to Auto, the default is to use HPC Pack first, then Workgroup.

Electronics Desktop products include four units of HPC for each licensed simulation. This means that up to four units can be used without requiring HPC licenses; license counting begins with the fifth unit. For example, a simulation that uses 36 cores requires 32 HPC units after

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

subtracting the four included cores. This simulation will check out 32 HPC workgroup licenses, or two HPC pack licenses.

HPC licenses enable all parallel and distributed simulations, including distributed variations. Distributed variations require a single set of solver licenses, plus HPC to enable the variations.

For HPC Workgroup, distributing *N* variations requires 8*(*N*–1) workgroup licenses and, together with the solver licenses, enables up to four HPC units per variation. Each additional set of *N* workgroup licenses will enable one additional HPC unit per variation. For HPC Pack, distributing *N* variations requires *N*–1 pack licenses and, together with the solver licenses, enables up to four HPC units per variation. Each additional set of *N* pack licenses will enable 8, 32, 128,... additional HPC units per variation.

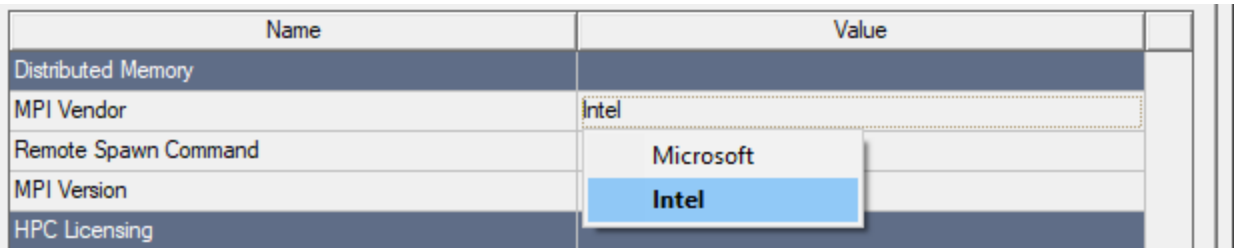
Ansys licensing supports distributed simulations when Electronics Desktop is called from other Ansys tools, such as optiSLang and Workbench. In such cases, distributed design points (variations) generally use HPC counts as described above.

> **Note:**
>
> Licensing for some calling products may include some distributed design points, in which case the total required HPC will be reduced.

If the **Queue all simulations** check box is selected, the Desktop queues any active simulations for design types that have **Save before solving** turned off in the **General Options** and then processes them in order. You can view and change the queue by using Show Queued Simulations.

To configure options:

1. Under **Distributed Memory**, for solvers that use MPI (HFSS, HFSS 3D Layout, Icepack, Maxwell and Q3D) use the drop-down menu to select the MPI Vendor for the selected design type.



   The solvers use the industry standard Message Passing Interface (MPI) and can perform solutions that distribute memory use across machines in a cluster or network. Memory used by the MPI-enabled solver is therefore limited by the set of machines that is available rather than the shared memory available on any single machine. This allows you to

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

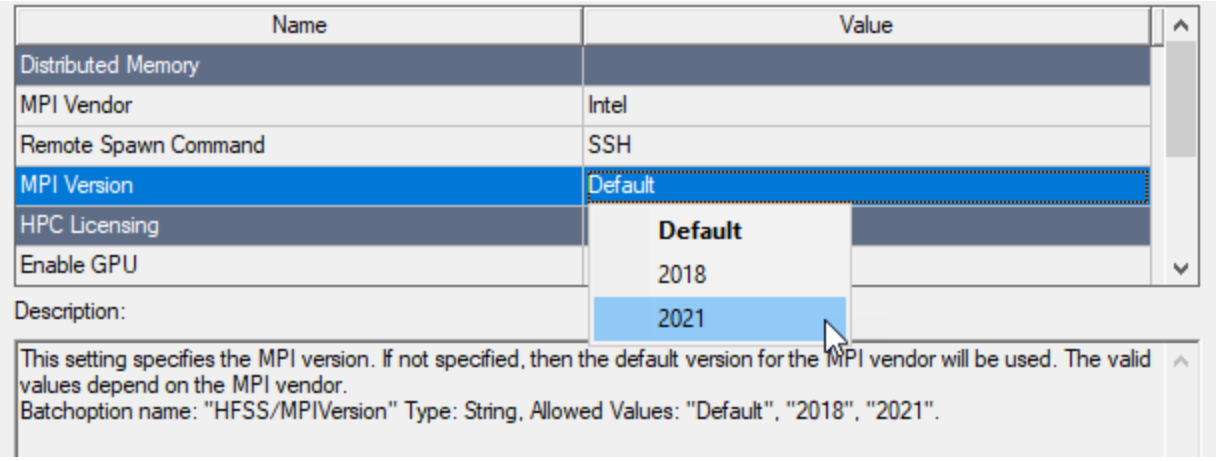
simulate larger structures and to optimally reconfigure the cluster of machines for the problem at hand. For solving on a single machine, MPI is not required, nor does it provide an advantage.

To use the distributed memory solution you will need to install MPI software from a supported third party vendor on all the machines you intend to use.

Depending on the MPI vendor, you may need to set passwords for authentication on the machines. Settings within each design type turn on distributed memory solutions and define the list of machines you intend to use.

You can specify the MPI version. If not specified, the default version for the MPI vendor will be used. This setting is ignored if there is only one supported version for the selected MPI Vendor. Multiple versions are only supported for Intel MPI, as follows:

- The value "Default" indicates that the default Intel MPI version should be used. This is Intel MPI 2021 in most cases.
- The value "2018" indicates that Intel MPI 2018 should be used.
- The value "2021" indicates that Intel MPI 2021 should be used.



Also see Setting Intel MPI Interconnect for more details.

InfiniBand Support for Windows

By default, MPI vendors use the fastest interconnect by default (typically InfiniBand is faster than Ethernet). If you want to override the default behavior and force the use of Ethernet, you can set the ANSOFT_MPI_INTERCONNECT environment variable to "eth" for the job. Also see Setting Intel MPI Interconnect for more details.

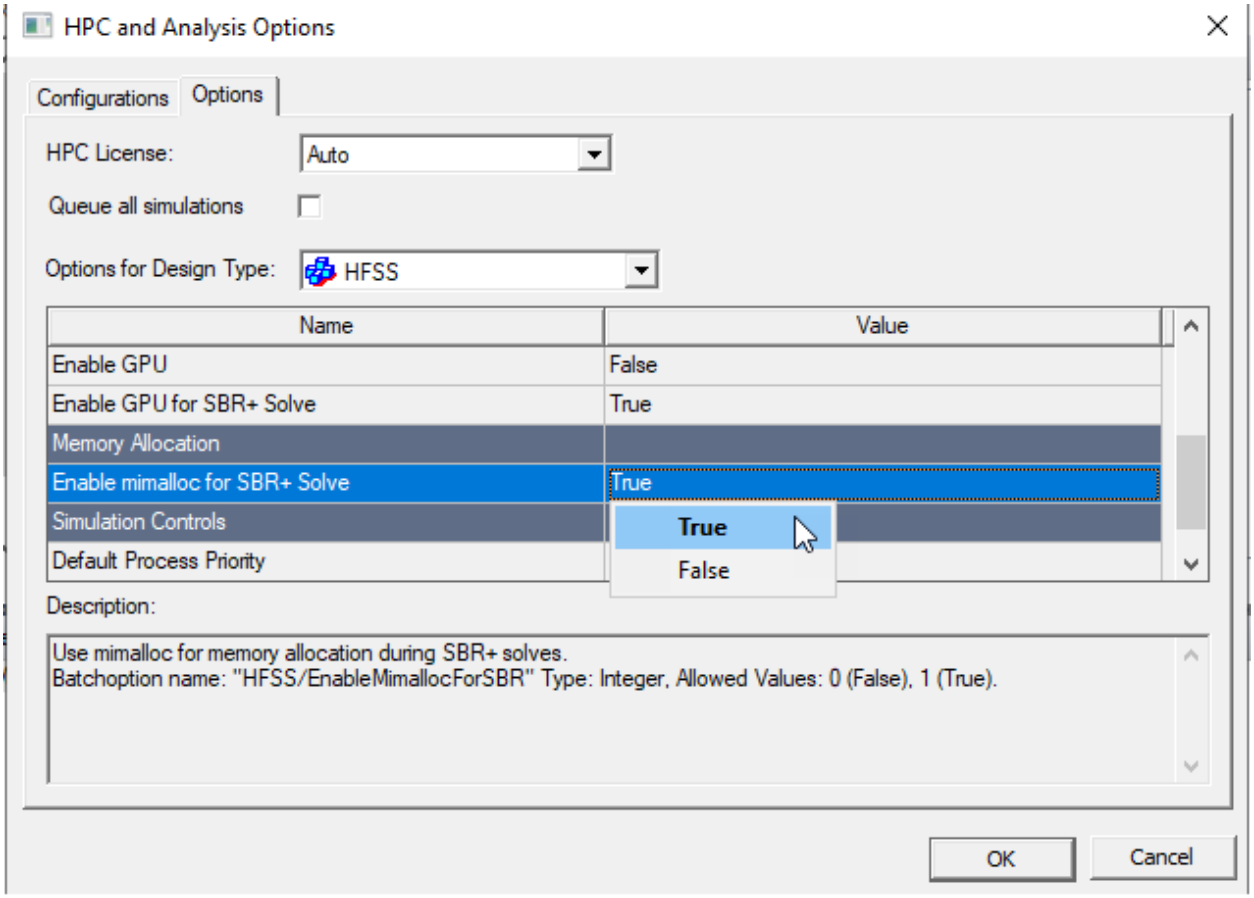
2. For Linux authentication, specify the Remote Spawn command as RSH or SSH (the default).
3. Electronics Desktop supports GPU acceleration for transient, frequency domain, SBR+, and Maxwell 3D AC magnetic matrix solutions.

   See: Enabling GPU.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For details on the requirements for GPU use, see Transient GPU Acceleration

4. **Memory Allocation**. The HFSS SBR+ solver leverages mimalloc by default on Windows to deliver better performance, especially in multi-threaded environments. On certain systems, this might cause the HFSS SBR+ not to execute. In this case, users can use the default Windows memory allocation by setting the "Enable mimalloc for SBR+ solve" option to "False"."



5. **Simulation Controls**: Optionally, you can select one of the following from the **Default Process Priority** drop-down menu:
   - **Critical** (highest) (Not recommended)
   - **Above Normal** (Not recommended)
   - **Normal** (Default)
   - **Below Normal**
   - **Idle** (lowest)

You can also set these values using Python scripts.

For information on editing configurations, see Editing Distributed Machine Configurations.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specifying the Remote Spawn Command (Linux)

An important step in using a high performance cluster is setting up authentication across machines so that the machines can be accessed without a password. By default, Maxwell uses SSH authentication on Linux to spawn commands on the remote machines but also supports RSH. The selection of which to use is made on the **Options** tab of the **Tools > Options > HPC and Analysis**.

SSH

You will need to set up passwordless access to use Maxwell on a Linux cluster with SSH or RSH. In general, for SSH, this is accomplished by:

1. Verifying that you have working SSH servers and clients on your machines.
2. Verifying that the server will accept passwordless logins. You may need to edit the /etc/ssh/ssh_d file to allow RSAAuthentication and PubkeyAuthentication.
3. Generating keys on the client system using the ssh-keygen program. Do not use a passphrase so that you can access the machine without a password.
4. Copying the public key generated in step 1 from the ~/.ssh directory to the server. The easiest way to transfer the keys is to use the ssh-copy-id program. Alternately, you can use any file transfer utility. If the server already has a list of existing keys for other clients add the new public key to the list.
5. Testing the connection. Log in to the client machine using the username that you used to create the identity keys. Open a new shell terminal and attempt to open an SSH login session. For example, type `ssh 192.168.0.4` (where the IP address is the address of the machine you are attempting to connect to). The server should allow you to log in without requesting a password.

Consult the documentation for your machines and network for detailed instructions.

RSH

If you choose to use RSH, you will need to make sure RSH is installed on all the machines and set up the machines so that you are not prompted for a password. There are different ways to set up passwordless RSH, so be sure to consult the documentation for your machines and network for detailed instructions.

Machine access using RSH without a password is often set up by editing the /etc/hosts.equiv file and adding entries for the hosts you would like to use without a password. This file lists hosts and users that are granted "trusted" access to the system.

If you look at the /etc/hosts.equiv file you should have something similar to the following:

Contents of the /etc/hosts.equiv file:

```
job1.n1.com
job2.n1.com
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
job3.n1.com
```

The machines job1, job2 and job3 can connect without a password. You may also need to verify that the files /etc/hosts.allow and /etc/hosts.deny are empty. Consult your local documentation for detailed instructions and troubleshooting suggestions.

Scheduler

The Remote Spawn Command setting is only meaningful when running on the Linux Operating System. The value 'Scheduler' is valid if the job is a scheduler job running under an LSF, SGE or SLURM scheduler, and only if the MPI Vendor is "Intel".

When submitting a job using the AnsysEM job submission GUI, the Remote Spawn Command for an analysis may be specified using the batchoption with pathname 'DesignType/RemoteSpawnCommand', where DesignType is the Design Type to analyze. The Remote Spawn Command setting is only meaningful when running on the Linux Operating System. The value 'Scheduler' is valid if the job is a scheduler job running under an LSF, SGE or SLURM scheduler, and only if the MPI Vendor is "Intel". To specify the value 'Scheduler' for this option for a scheduler job, the Remote Spawn Command must be specified using the 'DesignType/RemoteSpawnCommand' batchoption in the product command line when the product is launched. In addition, the 'DesignType/MPIVendor' batchoption must be specified with value "Intel" in the product command line when the product is launched. For interactive scheduler jobs, the Remote Spawn Command and the MPI Vendor may be specified with batchoptions or as design type options in the **HPC and Analysis Options** dialog box.

Setting MPI Version

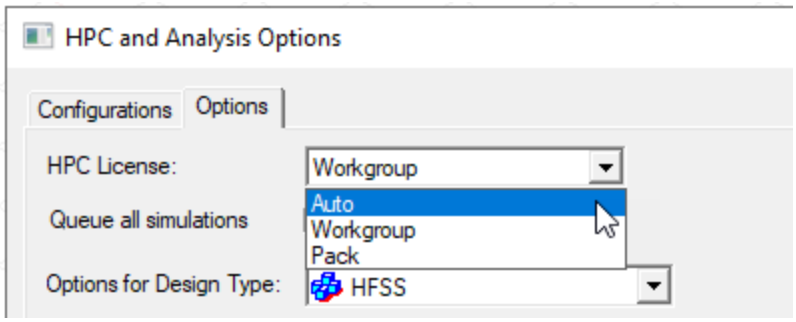
When submitting a job using the Ansys Electromagnetics job submission GUI, the MPI Version may be specified using the batchoption with pathname DesignType/MPIVersion, where DesignType is the type of design to analyze (e.g. HFSS). It allows selection of which Intel MPI version to use, for both Windows and Linux. Valid values are "Default", "2018", and "2021".

For interactive solves, the MPI version may be specified with the batchoption in the command line used to launch the product, or as a design type option in the HPC and Analysis Options dialog box.

Licensing Settings Tool

HPC licensing includes a choice called “Auto”, along with Workgroup and Pack. The “Auto” choice is available in the HPC License combobox found on the **Options** tab of the **HPC and Analysis Options** dialog box. If you make changes in the License Settings Tool, we recommend that you restart Electronics Desktop.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



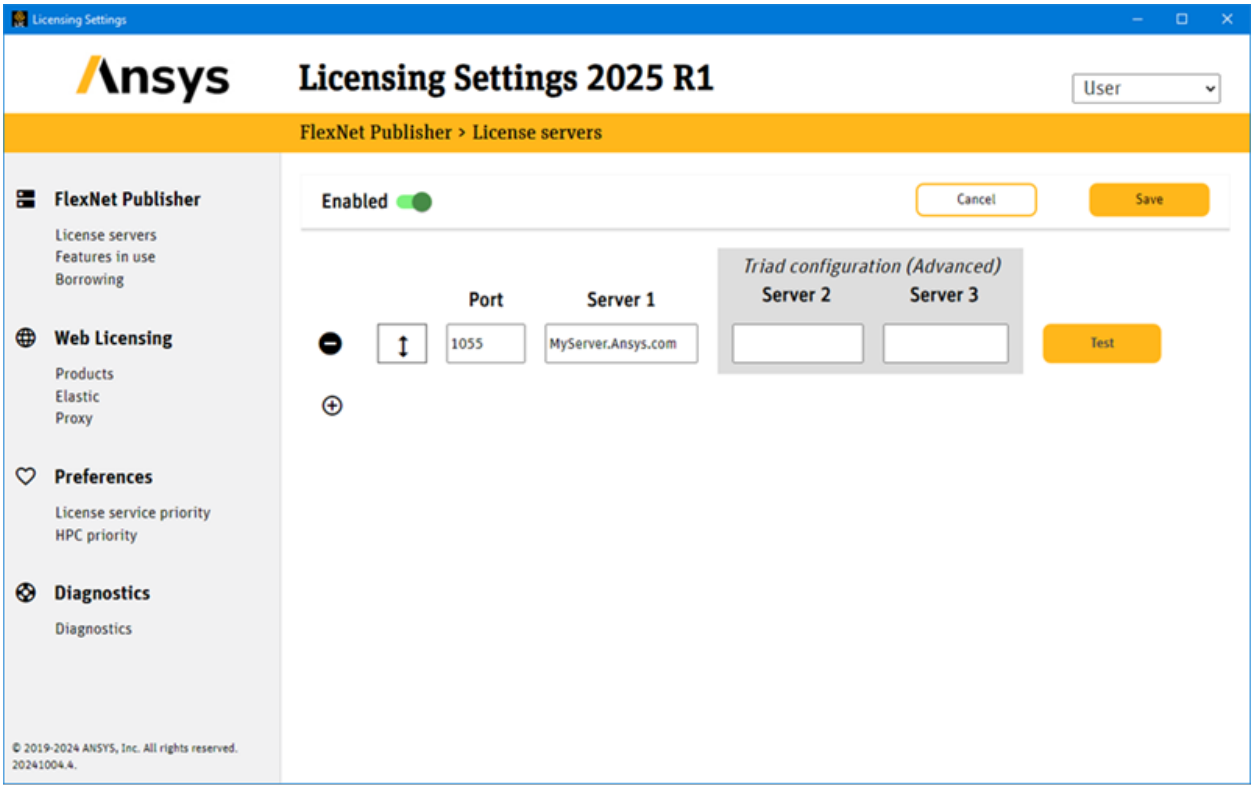
"Auto" means delegate the choice of Workgroup or Pack licensing to the Ansys Licensing Settings tool which is a separate application installed along with Electronics Desktop, accessible from the **Tools** menu. The HPC settings specified by the Licensing Settings tool are shared by other Ansys products. This provides you a single place where you can set the HPC preferences.

> **Note:** Important: HPC licensing preferences in the Licensing Settings tool are used only when Auto is selected in Electronics Desktop.

Launching the Licensing Settings Tool from the Main Menu

Click **Tools** > **Licensing Settings** to launch the Licensing Settings tool.

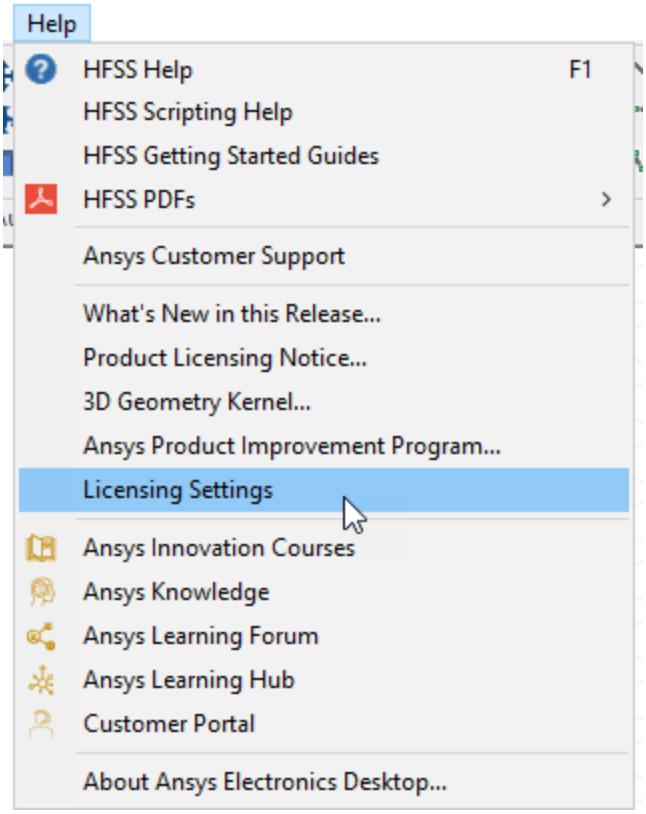
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Licensing Settings Help

You can obtain help with the Licensing Settings tool from the **Licensing Settings** menu item in the **Help** menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



License Fallbacks

When using Auto HPC licensing, in some cases if there aren't enough available licenses of the highest priority HPC license type, the lower priority HPC license type will be used. For instance, assume the user has set the priority order to "Ansys HPC, Ansys HPC Pack" in the Licensing Settings tool. If the user requests a nominal solve using 14 cores (10 Ansys HPC), but 10 Ansys HPC licenses aren't available, 2 Ansys HPC Pack licenses will be used if available.

Using GPU Acceleration for Maxwell 3D Solvers

Maxwell 3D solvers can be accelerated by using an NVIDIA® GPU accelerator. Numerical factorization of symmetric matrices is supported for GPU acceleration.

You can also enable distributed GPUs for AC Magnetic Frequency Sweeps.

Hardware Requirements

GPU acceleration for the Maxwell 3D AC Magnetic solver has been developed for NVIDIA cards and is officially supported with the Tesla® series. NVIDIA Tesla cards are recommended for the best performance when using several cards on one machine to solve using HPC. The following cards are supported:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- NVIDIA Tesla K20c (both workstation and server)
- NVIDIA Tesla K40c (both workstation and server)
- NVIDIA Tesla K80 (server with GPU cooling solution only, 2 GPUs consume 2 Ansys HPC licenses)
- NVIDIA Quadro K5000, K5200, M6000
- NVIDIA Quadro K6000 (both workstation and server, will be slow in graphic processing when used for GPU acceleration)

> **Note:**
>
> NVIDIA Tesla M2090 (Not supported in Release 17), a previous generation (code Fermi) GPU card, doesn't work for Workstation since it has no fan for active cooling, but rather needs server with GPU cooling solution (passive cooling) similar to the NVIDIA Tesla K80.

To obtain the best performance, the GPU used for running simulation jobs should not be attached to any display. Only GPU cards with CUDA® Compute Compatibility 2.0 and above should be used. To improve the speedup of visualization, you should install the GPU card on a system with PCI-E 3.0 slots. A mixture of interface cards with lower PCI-E versions may result in the data not being transferred from GPU to CPU at the highest speed.

Setup for Windows

1. After you install GPU cards and Nvidia graphics drivers, you should be able to find the cards in Windows Display Manager**.**
2. Run `nvidia-smi.exe` at C:\Program Files\NVIDIA Corporation\NVSMI to check if GPU cards are installed successfully. (The executable `nvidia-smi.exe` should be available after the display driver is installed.)
3. To further set up the configuration of GPU cards, open a command window as an administrator.
4. To improve the performance of GPU acceleration, it is recommended that you turn off the Error Correction Code (ECC) support using the `-e 0` option of nvidia-smi.

   ```
   nvidia-smi.exe -e 0
   ```

   New ECC settings become effective only after system reboot.
5. (Optional) For remote execution of GPU accelerated jobs (e.g., through Windows Remote Desktop Connection or RSM options), it is necessary to turn on the Tesla Compute Cluster (TCC) mode using the `-dm 1` option of nvidia-smi.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
nvidia-smi-exe -dm 1
```

New TCC settings become effective only after system reboot.

> **Note:**
>
> - The above step is unnecessary if you run Maxwell 3D AC Magnetic from a local machine.
> - Only Tesla cards support TCC.
> - You cannot run GPU accelerated jobs on remote GeForce and Quadro cards.

Setup for Linux

1. After you install an NVIDIA GPU card and graphics drivers, you should be able to find the card using the command:

   ```
   /sbin/lspci | grep -i nvidia
   ```

   You can also use the following command to check if a GPU card can be recognized by the system:

   ```
   /usr/bin/nvidia-smi
   ```

2. (Optional) The setting of GPUs to disable ECC (for performance), enable TCC (for remote execution), and enable Exclusive_Process (for GPU-distributed) are similar to Windows. You need the administrative right to make such changes.

   ```
   sudo nvidia-smi -e 0
   ```

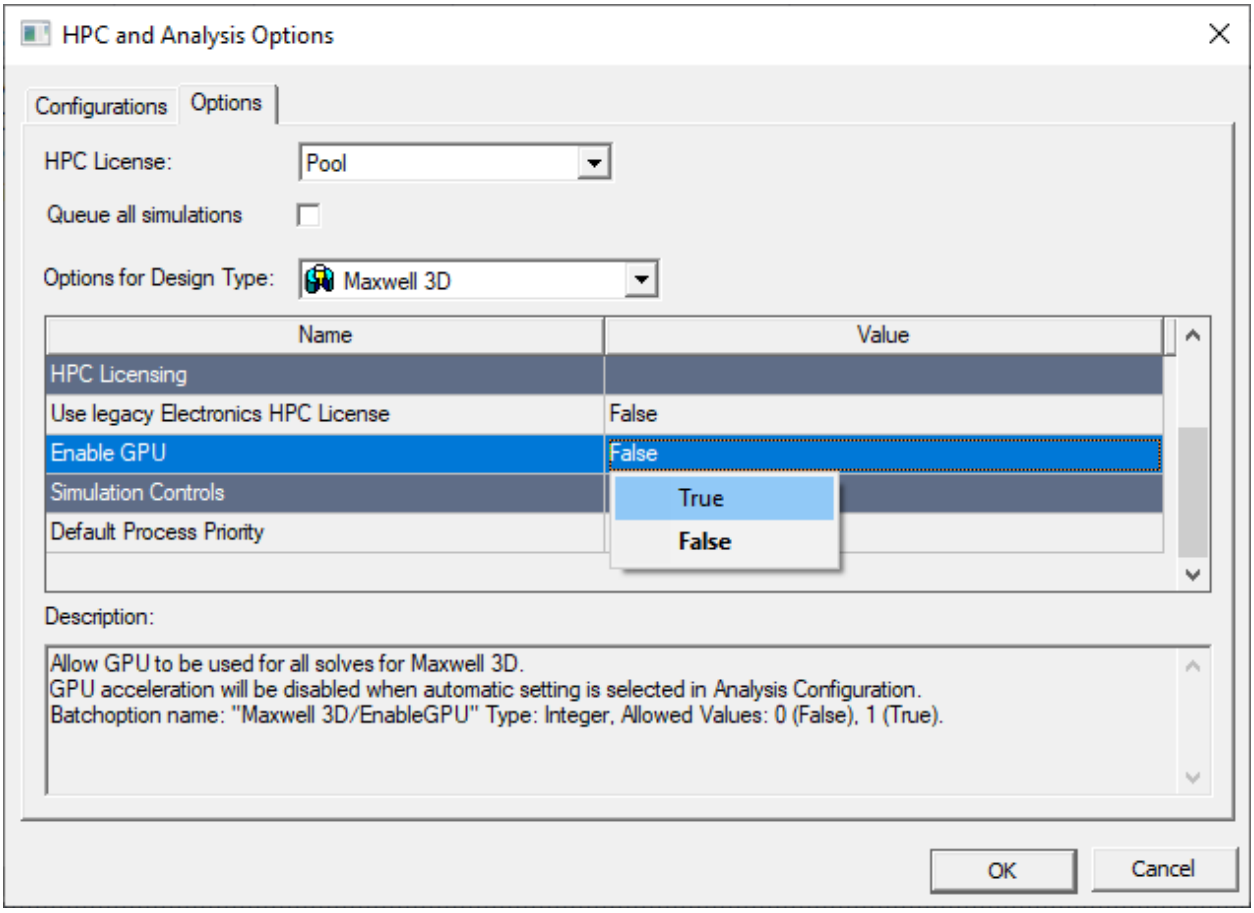
   ```
   sudo nvidia-smi -dm 1
   ```

   ```
   sudo nvidia-smi -c 3
   ```

Enabling GPUs for Maxwell 3D Solvers

1. Go to **Tools** > **Options** > **HPC and Analysis Options**.
2. In the Options tab, set the **Enable GPU** option to **True**.

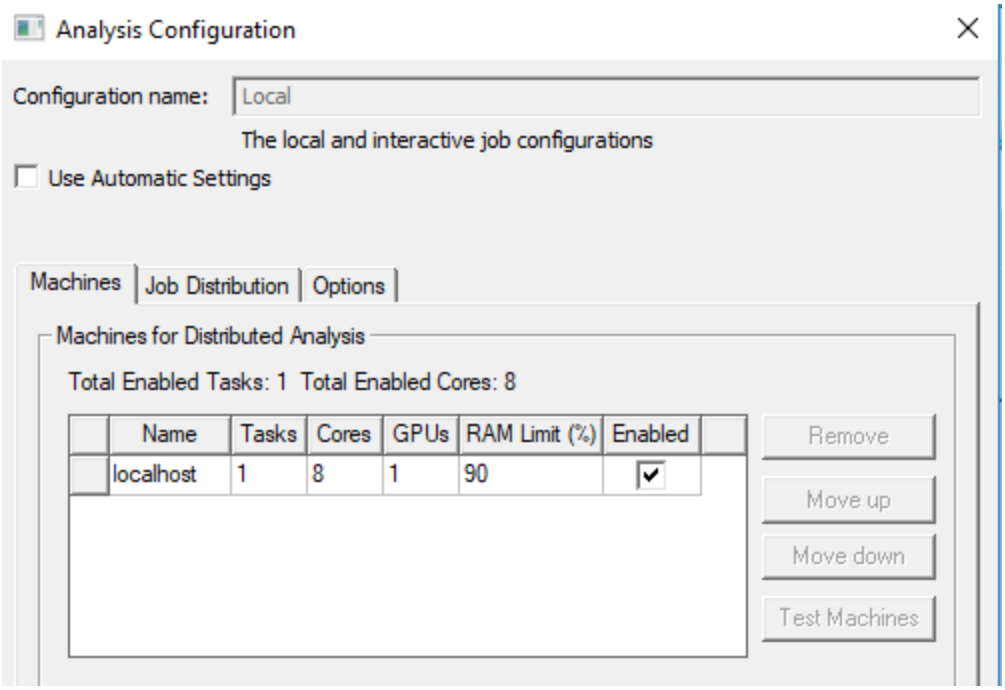
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Setting Up Non-distributed GPU for Maxwell 3D Solvers

In the **Analysis Configuration** dialog box **Machines** tab, in the **Machines for Distributed Analysis** section, set **GPUs** to 1.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

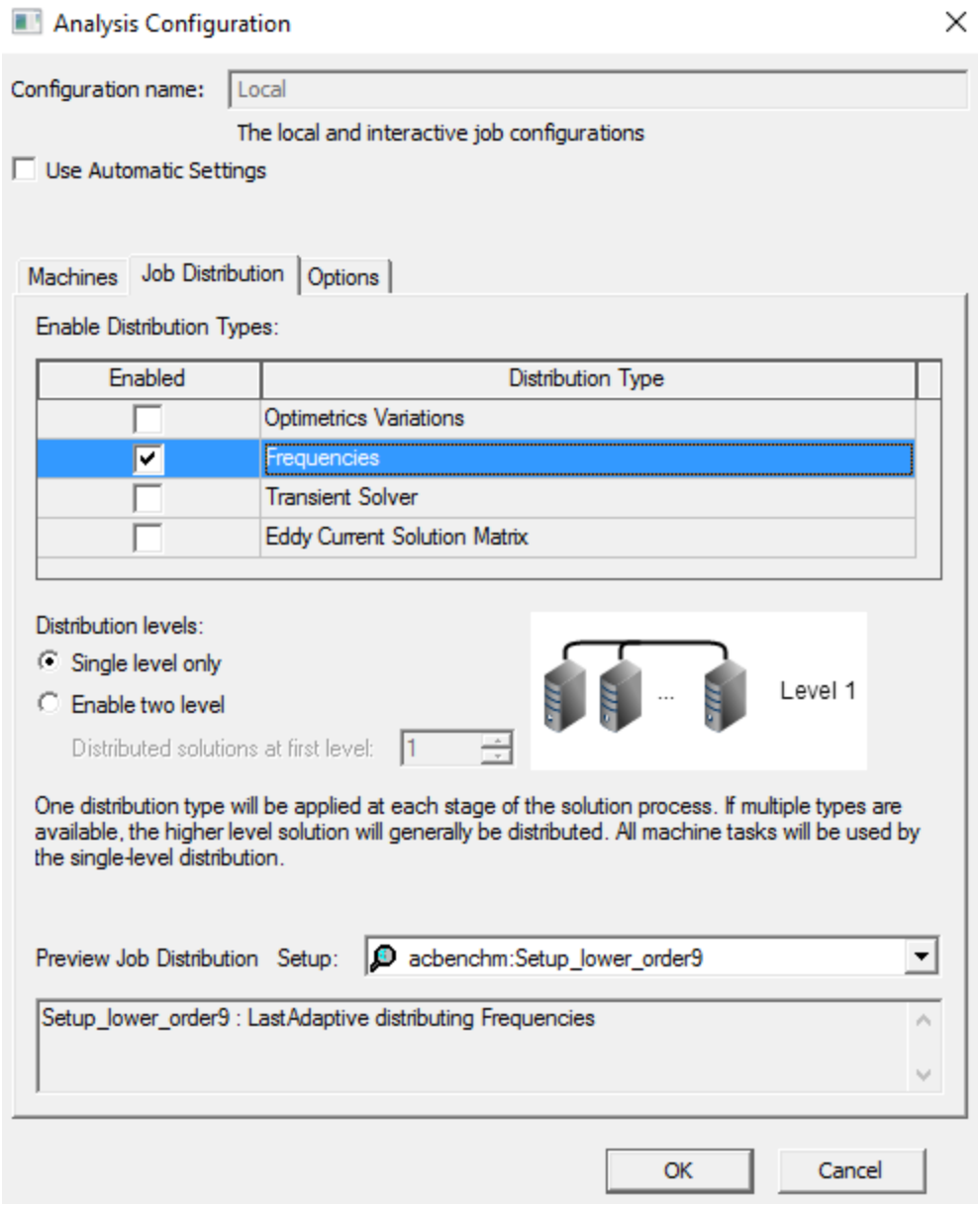


Enabling Distributed GPUs for AC Magnetic Frequency Sweep

To set up distributed GPUs for AC magnetic frequency sweeps:

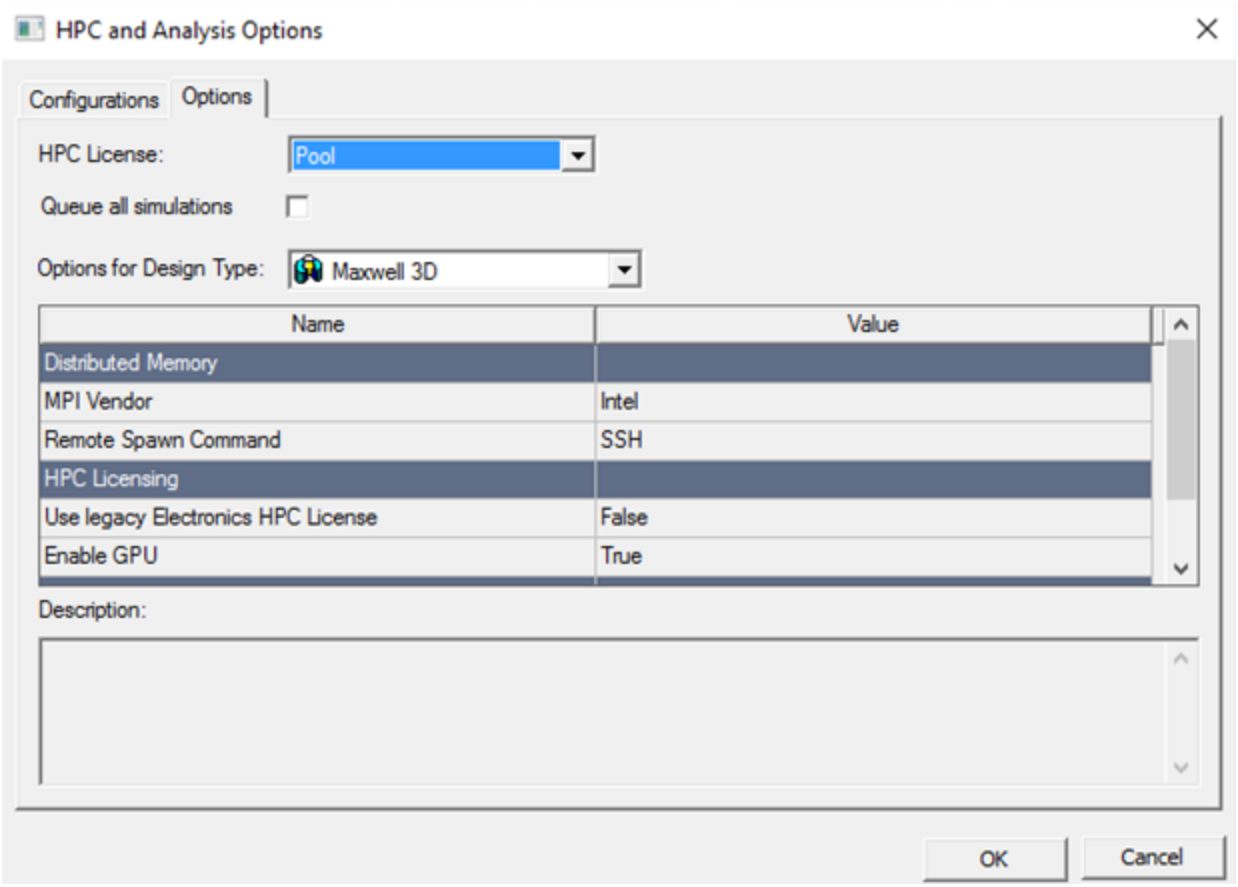
1. Go to **Tools** > **Options** > **HPC and Analysis Options** and click **Edit** in the dialog box, or select the **Simulation** tab of the ribbon, and click **Analysis Config** to edit Analysis Configurations.
2. Uncheck **Use Automatic Settings**, select the **Job Distribution** tab, then select **Frequencies**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Go to **Tools** > **Options** > **HPC and Analysis Options**, select the **Options** tab, and set **Enable GPU** to **True**.

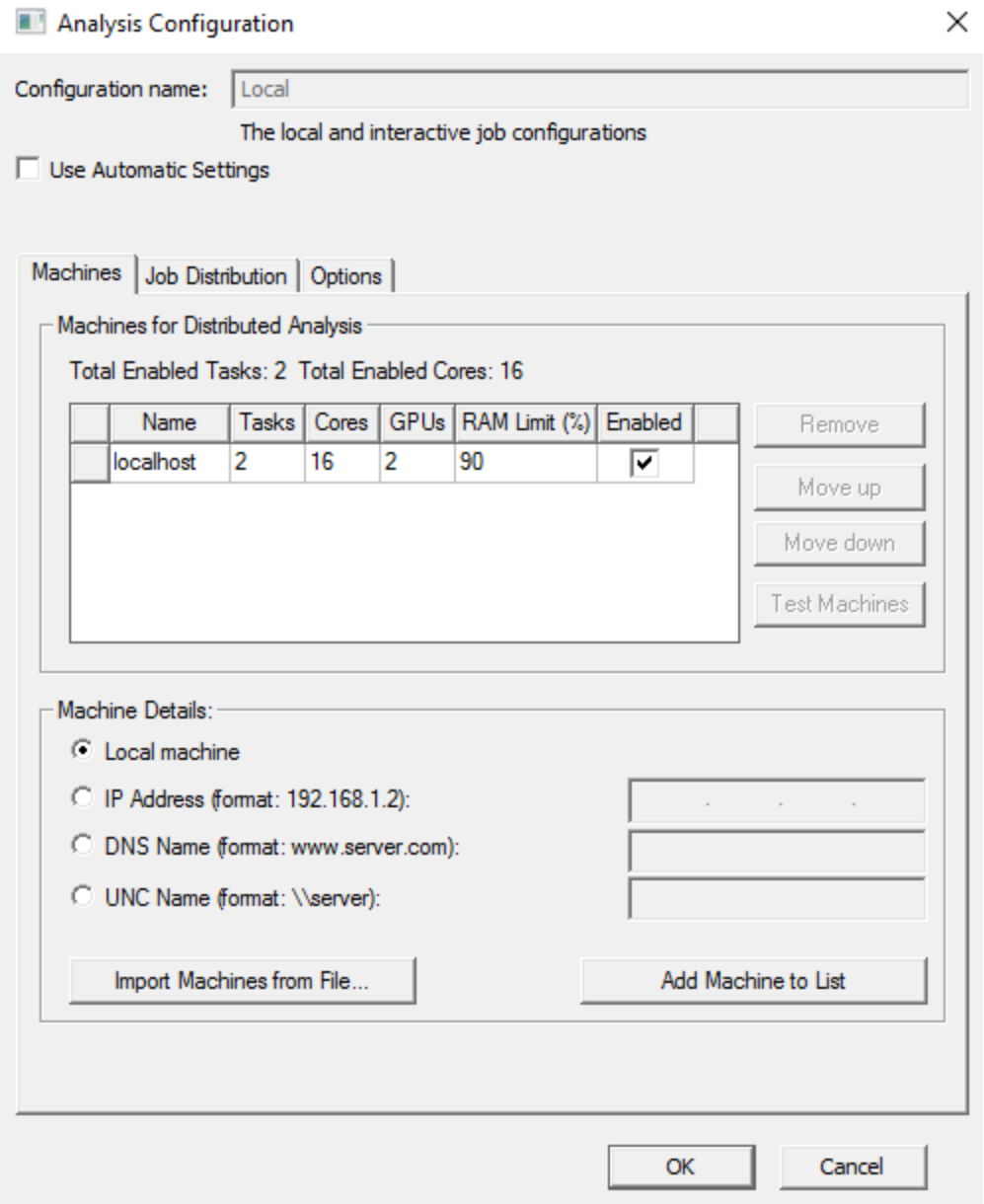
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Set up the Machines for distributed GPU usage.

   For example, the following figure shows two simulation jobs set to run on the local host; and up to two GPUs can be used for acceleration depending on the availability of licenses and GPU cards.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Setting Options via Configuration Files

In addition to setting options from the Electronics Desktop user interface (UI), you can also set options in several configuration files. Option settings in configuration files may apply to all users or only to a specific user, and they may apply to all hosts or only to specific hosts. There are four levels of settings, listed below from most specific (highest precedence) to most general (lowest precedence):

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Host-dependent user options – apply to the specified user on the specified host only
- Host-independent user options – apply to the specified user on all hosts
- Host-dependent default options – apply to all users on the specified host
- Installation default – default for all users on all hosts

In the list above, settings at any level override settings at lower levels . If there is no setting in any file, the application default value is used. See UpdateRegistry for instructions on selecting these levels.

> **Important:**
>
> Options set from the Desktop UI override and update user settings in configuration files. Otherwise, the existing configuration file settings are used.

Behavior Examples

Consider running an application as user **jsmith** on host **host123**. If there is no host-dependent user setting for the *Expand Project Tree on Insert* option in the host-dependent user options config file for user **jsmith** on host **host123**, but there is a setting for the "Expand Project Tree on Insert" option in the host-independent user options config file for user **jsmith**, then the latter setting will be used if it is not overridden using the Desktop UI. Any settings in the host-dependent default options config file or the installation default config file are ignored.

As another example, consider running an application as user **jdoe** on host **host123**. If there is no setting for the *Expand Project Tree on Insert* option in the host-dependent user options config file for **jdoe** on **host123** or in the host-independent user options config file for user **jdoe** or in the host-dependent default options config file for host **host123**, then the value from the installation default config file are used, if present.

Rules for Modifying Option Settings

Option settings displayed in the Desktop UI follow the above rules. That is, if there is a setting in any of the option config files, then the setting from the highest priority config file is displayed in the Desktop UI. If there is no setting in any of the option config files, then the global default value is used. You can modify settings using the Options dialog boxes in the Desktop UI. If the dialog box is closed with the **Cancel** button, changes made on any of the tabs are discarded. If the dialog box is closed with the **OK** button, any settings that have been changed by the user are written to the host-dependent user options config file. The changed values written to this file are then used the next time that the application is run by the same user on the same host. The Desktop UI option settings are not written to any of the other option config files.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Configuration File Locations

The configuration files for host-dependent and installation **default** options reside at: "<*installation-directory*>\ANSYS Inc\v252\AnsysEM\config". The configuration files for host-dependent and host-independent **user** options reside in a subfolder of the user's Documents folder (for Windows) or the user's HOME folder (for Linux). See the tables below for specific Windows and Linux file names and paths.

Products with Multiple Desktop Versions

For products that have multiple Desktop versions, each will have a separate user-specific config directory, with a different value for the <*ApplicationName&Version*> directory name.

Table of Directories and Files

The following table shows the directories and files, where the *Level Name* is the name used to describe an options config file when using the UpdateRegistry tool.

Config File	**Level Name**	**File Name**	**Windows Directory Path**	**Linux Directory Path**
host-dependent user options	user_machine	<*hostname*>_user.XML	%UserProfile%\Documents\Ansoft\<*ApplicationName&Version*>\config	$HOME/Ansoft/<*ApplicationName&Version*>/config
host-independent user options	user	user.XML		
host-dependent default options	install_machine	<*hostname*>.XML	*"<Installation_Directory>*\ANSYS Inc*"<Version>*\AnsysEM\config"	*"<Installation_Directory>*/ansys_inc/*"<Version>*/AnsysEM/config"
installation default	install	default.XML		

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- ***<hostname>*** is the name of the computer on which the Electronics Desktop software is installed
- **$HOME** is the user's home directory on Linux
- ***<ApplicationName&Version>*** is the product name (without spaces) followed by the four-digit year of the version, a decimal point, and the minor release number (such as ElectronicsDesktop2025.2)
- **%UserProfile%** is a Windows variable that represents the currently active user's profile (for example, C:\Users\JohnDoe)
- ***<Installation_Directory>*** is the root folder where the Ansys software is installed (typically, C:\Program Files, on Windows, or /opt, on Linux)
- ***<Version>*** is the lower-case letter "v" followed by last two digits of the product version's year and the minor release number, without a decimal point (such as v252)

The following table shows an example of specific file names and directory names for a typical Ansys Electronics Desktop installation on Microsoft Windows and on Linux. These are the files that apply to the current software version, user "**jsmith**," and hostname "**host123**":

Config File	Level Name	File Name	Windows Directory Path	Linux Directory Path
host dependent user options	user_machine	host123_user.XML	C:\Users\jsmith\Documents\Ansoft\ ElectronicsDesktop2025.2\config	/home/jsmith/Ansoft/ ElectronicsDesktop2025.2/config
host independent user options	user	user.XML		
host dependent default options	install_machine	host123.XML	"C:\Program Files\ANSYS Inc\v252\ AnsysEM\config"	/opt/ansys_inc/v252/AnsysEM/config
installation default	install	default.XML		

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

As with the temporary file location configuration files, the settings in these options files have precedence in the following sequence:

user_machine (highest precedence), user, install_machine, install (lowest precedence).

The first time you start and exit the application, the file at the "user_machine" level is created (<*hostname*>_user.XML). The other files are only created if you do one of the following:

- Use the Choose Registry Type command to switch to a machine-independent registry (user.XML), or
- Use the UpdateRegistry tool to specify an option at the "user," "install_machine," or "install" level.

If the temporary directory is set to an empty string in a configuration file, then that setting is ignored.

Setting or Removing Option Values in Configuration Files: UpdateRegistry Command

UpdateRegistry is a command line tool used to modify option settings in the options config files. You can use this command to add, change or remove settings from any of the option config files. This tool is included in the installation directory of each product. This feature makes it easier for different users to use Ansys Electromagnetics tools installed on shared directories or network drives.

The UpdateRegistry command has multiple command line formats, as shown below.

The following command line options are *mutually exclusive*:

- `-Set`
- `-Get`
- `-GetKeys`
- `-Delete`
- `-FromFile`

UpdateRegistry -Set Command

This command is used to add or modify an option setting in an option config file. If the option config file does not exist, it will be created. If the setting does not exist in the specified config file,

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

it will be added. If the setting already exists in the specified config file, then the value will be changed to the specified value.

Example:

```
UpdateRegistry -Set -ProductName <name> -
RegistryKey <keyPath> -RegistryValue <value> [
-RegistryLevel <level> ]
```

Required:

`<name>`	The application or product name and version. For example, ElectronicsDesktop2025.2. If the name contains spaces, it must be quoted. The name can be found in the ProductList.txt file in the install directory: "...\ANSYS Inc\v252\AnsysEM\config\"
`<keyPath>`	The pathname of the option setting. This includes the same analysis-related registry keys and values that are displayed by the -batchoptions help. For example, Desktop/Settings/ProjectOptions/AnimationMemory
`<value>`	The new value of the option, typically a string or a number. If the value contains spaces, it must be quoted.

Optional:

`<level>`	When specifying `-RegistryLevel`, this is a string denoting which config file to modify. One of: install, install_machine, user, and user_machine. If the level is not specified, the user_machine (host-dependent user options) file is modified.

UpdateRegistry -Get Command

This command is used to view an option value in an option config file. If the setting exists in the specified config file or files, then the value, the value type and the config file where the value was found will be reported. If no value is found, then that will also be reported.

Example:

```
UpdateRegistry -Get -ProductName <name> -
RegistryKey <keyPath> [ -RegistryLevel <level>
]
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Required:

`<name>`	The application or product name and version. For example, ElectronicsDesktop2025.2. If the name contains spaces, it must be quoted. The name can be found in the ProductList.txt file in the install directory: "...\ANSYS Inc\v252\AnsysEM\config\"
`<keyPath>`	The pathname of the option setting. This includes the same analysis-related registry keys and values that are displayed by the -batchoptions help. For example, Desktop/Settings/ProjectOptions/AnimationMemory

Optional:

`<level>`	When specifying `-RegistryLevel`, this is a string denoting which config file to modify. One of: install, install_machine, user, and user_machine. If the level is not specified, then all config files are searched in order of precedence.

UpdateRegistry -GetKeys Command

This command is used to view the allowed key names for all of the option settings, or to view a subset of the key names that match a string. For each key displayed, the current value, if any, is also reported. If a key has a value in multiple config files, then only the highest precedence value is reported.

Example:

```
UpdateRegistry -GetKeys [ <pattern> ] -
ProductName <name> [ -Case ]
```

Required:

`<name>`	The application or product name and version. For example, ElectronicsDesktop2025.2. If the name contains spaces, it must be quoted. The name can be found in the ProductList.txt file in the install directory: "...\ANSYS Inc\v252\AnsysEM\config\"

Optional:

`<pattern>`	If no pattern is specified, then all allowed key names are reported. If a pattern is specified, then only keys that match the pattern are shown. For example, Settings/Project. If the name contains spaces, it must be quoted. By default, the pattern match is case insensitive.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

-Case	If this command line option is specified, then the pattern match is case sensitive.

UpdateRegistry -Delete Command

This command is used to remove an option setting from an option config file. If the setting does not exist in the specified config file, the file will not be changed. If the setting exists in the specified config file, then it will be removed. A setting may need to be removed from an option config file, to allow the setting from a lower priority file to be used by the application.

Example:

```
UpdateRegistry -Delete -ProductName <name> -
RegistryKey <keyPath> [ -RegistryLevel <level>
]
```

Required:

<name>	The application or product name and version. For example, ElectronicsDesktop2025.2. If the name contains spaces, it must be quoted. The name can be found in the ProductList.txt file in the install directory: "...\ANSYS Inc\v252\AnsysEM\config\"
<keyPath>	The pathname of the option setting. This includes the same analysis-related registry keys and values that are displayed by the -batchoptions help. For example, Desktop/Settings/ProjectOptions/AnimationMemory

Optional:

<level>	When specifying -RegistryLevel, this is a string denoting which config file to modify. One of: install, install_machine, user, and user_machine. If the level is not specified, the user_machine (host-dependent user options) file is modified.

UpdateRegistry -FromFile Command

You can use this form of the UpdateRegistry command to set multiple key-value pairs from a file with a single UpdateRegistry command. You specify the -FromFile command line option. This option must be followed by a filename. The file may contain multiple entries, where each entry contains a registry key and a registry value. The key-value pairs are added to the registry level specified by the -RegistryLevel command line option; if no -RegistryLevel is specified, then the default registry level (user_machine) is used.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UpdateRegistry File Format

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

The file format is similar to the -batchoptions file format. An example UpdateRegistry file is shown below:

```
$begin 'AddEntries'
  'TempDirectory'='C:/temp/AnsysEM'
  'Desktop/Settings/ProjectOptions/HPCLicenseType'='Pool'
$end 'AddEntries'
```

Additional notes on the file format:

- The file may contain an arbitrary number of entries, one per line.
- Leading whitespace on each line is ignored. Spaces or tabs may be used to make the file more readable.

Registry key pathname:

- The registry key pathname appears before the equal sign "=" on each line.
- Each registry key pathname must be enclosed in single quotes.
- This includes the same analysis-related registry keys and values that are displayed by the -batchoptions help.

Registry value:

- The registry value appears after the equal sign on each line.
- Integral registry values must not be enclosed in quotes.
- All other registry values are treated as strings, and must be enclosed in single quotes.
- The forward slash "/" may be used as a directory separator on Windows and Linux. The back slash "\" may be used as a directory separator on Windows only.
- The back slash "\" is used as an escape character in the value string. That it, this character removes the special meaning of the following character.
- The single quote character normally ends the value string. The back slash may be used to remove this special meaning, and include a single quote in the string.
- To use a back slash as a directory separator on Windows, it must be escaped. That is, a double back slash "\\" is used to denote a single directory separator.

Alternative UpdateRegistry File Format:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Analysis Configuration File format, which is exported from the HPC and Analysis Options dialog box.

> **Note:**
>
> If a current registry does not exist,Ansys Electronics Desktop can detect earlier minor versions of same application on the same machine. If such a registry exists (and does not involve -help, -batchoptionhelp, IsBatchMode(), -regserver, -unregserver, running a script, or non graphical mode), a prompt displays allowing you to port the registry from an earlier version.

Example Uses for Export Options Features

The **Tools** > **Options** > **Export Options** feature is intended to make it easier for different users to use Ansys Electromagnetics tools installed on shared directories or network drives. This section outlines some use cases enabled by this feature.

> **Note:**
>
> Functionality featured in the examples in this topic apply to multiple design types.

Options that Apply to All Users

In many cases, an Ansys Electromagnetics tool installation is administered and maintained by a single user or group and used by a number of other users or groups. The permissions of the Ansys Electromagnetics tool installation may be set so that the administrator may add, delete or modify files, but other users may only read or execute these files. The administrator may set the recommended option settings in the installation default config file and/or the host dependent default options config file. These config files reside within the installation directory hierarchy, and should generally have the same permissions as other Ansys Electromagnetics tool installation files. This allows that administrator to control these settings, but does not allow other users to add, remove, or change these settings.

Each user can override any of these settings, if needed. This may be done using the Desktop UI, which affects the host-dependent user options config file. It may also be done using the host-independent user options config file. If user has overridden an option setting in either of the user files, the user may revert back to the option settings provided by the administrator by removing the setting of the same option in the host-dependent user option config file and/or the host-independent user option config file.

For global defaults, the administrator may set a value in the installation default config file. These settings will to apply to all users on all hosts.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In some cases, there are significant differences between the capabilities of different hosts. The host-dependent default config file may be used to specify different default values on some hosts. Any setting in a host dependent default config file would affect all users running on the specified host. The installation default value is used if there is no value specified for the setting in the host-dependent default config file for the current host. Note that the host-dependent default config file is named <*hostname*>.xml, where *hostname* is the name of the host computer.

Example: Searching for a Registry Key Pathname

Both administrators and ordinary users may occasionally use the UpdateRegistry command line tool to add, change or delete settings. To use this tool, the registry key pathname must be known by the user. The -GetKeys option may be used to quickly search for a key pathname if some information is known about it. For example, if the administrator knows that there is a setting related to issuing warning messages when available disk space is low, but does not know the exact key name, the following command may list some of the keys related to disk space:

```
UpdateRegistry -GetKeys disk -ProductName ElectronicsDesktop2025.2
```

This will display a list of all keys that match the string "disk" case insensitively.

Typical output may look like the following:

```
Registry keys matching pattern <disk> case insensitively:
Desktop/Settings/ProjectOptions/DiskLimitForAbort: value is <0> at
level <user_machine>
```

Example: Setting an Installation Default Value

The normal default for the Options/General/Desktop Performance/Warn when available disk space is less than setting is 0 MB. If the administrator is concerned that running out of disk space might be a common problem, the administrator could set the installation default for the warn setting setting to 1000 MB, for example. This limit would then apply to all users running on all hosts. The administrator could use the following command to change this setting for Ansys Electronics Desktop:

```
UpdateRegistry -Set -ProductName ANSYSElectronicsDesktop2025.2
 -RegistryKey Desktop/Settings/ProjectOptions/DiskLimitForAbort
 -RegistryValue 1000
 -RegistryLevel install
```

Example: Setting a Host-Dependent Default Value

For this example, assume that all hosts have two cores, except for three hosts: bighost1, bighost2, and bighost3, which have eight cores each. The administrator has set the Desktop/Settings/ProjectOptions/NumberOfProcessors option value to 2 in the installation

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

default config file. The administrator may set the Desktop/Settings/ProjectOptions/NumberOfProcessors option value to 8 in the host-dependent default config files for the three hosts having 8 cores: bighost1, bighost2 and bighost3. The administrator may log in to host bighost1 and run the following command to change this setting for the host-dependent default options config file for host bighost1:

```
UpdateRegistry -set -ProductName ElectronicsDesktop2025.2
 -RegistryKey Desktop/Settings/ProjectOptions/NumberOfProcessors
 -RegistryValue 8
 -RegistryLevel install_machine
```

To make this change for the other two hosts, the administrator would log in to bighost2 and bighost3, and run the same command on each of those hosts.

Example: Reverting from a User-Defined Option Value to the Administrator Default

Consider the case in which Electronics Desktop was installed and the administrator initially did not set a value for the Desktop/Settings/ProjectOptions/DiskLimitForAbort setting in the default installation config file. User jsmith (who always uses host jshost) wanted to be warned before disk space dropped to zero, so he set the Desktop/Settings/ProjectOptions/DiskLimitForAbort to 100 MB using the UI. This setting is recorded in the host dependent user options config file for host jshost and user jsmith. Now the administrator learns that many users are running into disk space issues, so that administrator sets the installation default value for the setting Desktop/Settings/ProjectOptions/DiskLimitForAbort to 1000 MB, as in the above example.

When user jsmith runs Electronics Desktop on host jshost, the disk limit is 100 MB, not 1000 MB, because the host-dependent user options config file overrides all of the other config files. User jsmith may revert to the administrator provided default by removing this setting from the host dependent user options config file for host jshost and user jsmith. The following command may be run by user jsmith on host jshost to remove this setting:

```
UpdateRegistry -Delete -ProductName ElectronicsDesktop2025.2
 -RegistryKey Desktop/Settings/ProjectOptions/DiskLimitForAbort
 -RegistryLevel user_machine
```

If user jsmith had added a value for this setting to the host-independent user options config file, then user jsmith would also run the following command to remove this setting from the host-independent user options config file:

```
UpdateRegistry -Delete -ProductName ElectronicsDesktop2025.2
 -RegistryKey Desktop/Settings/ProjectOptions/DiskLimitForAbort
 -RegistryLevel user
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

User Options and the UpdateRegistry Tool

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

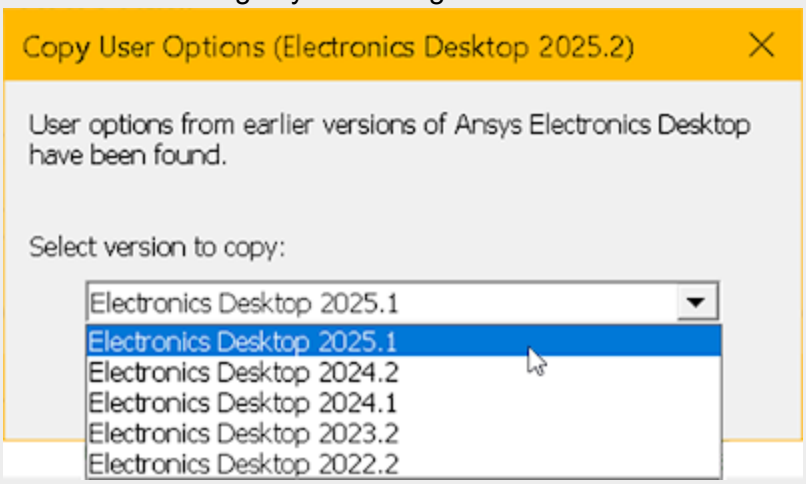
When you change an option's value using the Ansys Electronics Desktop UI, the new value is stored in the currently active user options registry. You can use the UpdateRegistry tool to add, modify, or remove settings. However, you cannot use the Desktop UI to *remove* settings from the registry. *You must use the UpdateRegistry tool to do that.*

If you have not already created a user options configuration file (either host-specific or host-independent), before launching the Ansys Electronics Desktop application for the first time, all settings will come from the *host-specific default* options configuration file or the *installation default options* configuration file. In a host-specific user options configuration file, any settings applicable to a different host from the current one will not be carried over to the new host. This behavior may be inconvenient if the user has preferred option settings that differ from the default settings, which apply to all users, especially if the user runs the application on a number of different hosts. In this case, the user may choose to use a host-independent registry and set the option values in this type of options file. See Understanding Registry Tools for more information. Host-independent option values can be used on all new hosts, overriding any values set by the administrator to apply to all users. Changes made in the UI can affect the user's host-specific or host-independent registry, depending on the currently specified registry type.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

If a current registry does not exist, the program detects earlier major and minor versions of same application on the same host. If such a registry exists (and does not involve -help, -batchoptionhelp, IsBatchMode(), -regserver, -unregserver, running a script, or non graphical mode), a prompt displays the earlier versions you can select, from which the registry will be migrated.



Copy over registry entries (both Windows and Ansys .xml files).

Example: Removing a Host-Dependent User Option Setting

For this example, user jsmith always uses host jshost to run Ansys Electronics Desktop. At some point, jsmith set the Autosave interval in **General Options** > **Project Options** tab to 1000 edits, and this value was written to the jsmith's host-dependent user options config file for host jshost. Now, jsmith wants to remove this setting and return to the default value of 10. User jsmith may run the following command on host jshost to remove the Desktop/Settings/ProjectOptions/AutoSaveInterval option value from this config file:

```
UpdateRegistry -Delete -ProductName ElectronicsDesktop2025.2
-RegistryKey Desktop/Settings/ProjectOptions/AutoSaveInterval
-RegistryLevel user_machine
```

Example: Getting a Value from a Specific Configuration File

In the previous example, the user jsmith may decide to check the Desktop/Settings/ProjectOptions/DiskLimitForAbort setting in the host-independent user configuration file before making any changes to this setting. The following command may be used to quickly view this setting before making the change:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
UpdateRegistry -Get -ProductName ElectronicsDesktop2025.2 -
RegistryKey Desktop/Settings/ProjectOptions/DiskLimitForAbort -
RegistryLevel user
```

Example: Getting a Value Using Precedence Rules

In may cases, the user is more interested in the value of a setting that will be applicable when running the product than in the setting in a single configuration file. If the -Get option is used with no -RegistryLevel specified, then the value reported is the value found in the highest precedence configuration file. If the user jsmith is interested in the highest precedence value for the Desktop/Settings/ProjectOptions/DiskLimitForAbort setting, then the following command may be used to report this information:

```
UpdateRegistry -Get -ProductName ElectronicsDesktop2025.2 -
RegistryKey Desktop/Settings/ProjectOptions/DiskLimitForAbort
```

Example: Adding a Host-Independent User Option Setting

Consider the case in which there is no value set for the Desktop/Settings/ProjectOptions/DiskLimitForAbort setting for all users for Ansys Electronics Desktop. The default is then 0 MB. User jsmith uses a variety of hosts and wants to be warned whenever disk space drops to 250 MB on any host. User jsmith may use the following command to set the Desktop/Settings/ProjectOptions/DiskLimitForAbort option value to 250 MB for all hosts:

```
UpdateRegistry -set -ProductName ElectronicsDesktop2025.2
-RegistryKey Desktop/Settings/ProjectOptions/DiskLimitForAbort
-RegistryValue 250 -RegistryLevel user
```

Example: Setting a RegistryKey not defined in ElectronicsDesktopRegistrySyntax.xml

```
"C:\Program Files\ANSYS Inc\v252\AnsysEM">UpdateRegistry -Set -
ProductName ElectronicsDesktop2025.2 -RegistryKey "3D
Editors/Preferences/Geometry3D/q" -RegistryValue "m"

registry key is not defined in this product

Register value <3D Editors/Preferences/Geometry3D/q> is created with
value <m>
```

Example: Change a Newly Set Registry Key

```
"C:\Program Files\ANSYS Inc\v252\AnsysEM\">UpdateRegistry -Set -
ProductName ElectronicsDesktop2025.2 -RegistryKey "3D
Editors/Preferences/Geometry3D/q" -RegistryValue "n"
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Register value <3D Editors/Preferences/Geometry3D/q> is set to <n>
```

If you attempt to set a registry key to the wrong data type, a message like the following appears.

```
Cannot set the registry value to different data type
The work around is delete it, then create it again
error on to set registry value at <3D
Editors/Preferences/Geometry3D/q>
```

Setting the Temporary Directory

The temporary directory may be viewed or set using the Electronics Desktop user interface (UI), or from the command line.

To set the directory via the UI:

1. Navigate to **Tools** > **Options** > **General Options**.
2. In the tree at the left side of the dialog box, expand the **General** branch and select **Directories**.
3. Use the **Temp** field and **Override** check box to enter a desired directory path. Values set in this manner are written to the user_machine level configuration file for the temporary directory. If the **Override** check box is cleared, clicking **OK** changes the user_machine level setting for the temporary directory to an empty string. This enables settings from the next highest precedence configuration file. The file that provides the currently active temporary directory setting is shown under the **Temp** edit box.

To set the temporary directory from the command line, using the -batchoptions command line option. See: Running Ansys Electronics Desktop from a Command Line and -Batchoptions Command Line Examples.

The temporary directory may be configured with an installation default value, as well as a host-dependent default value, a host-independent user-specified value and a host-dependent user-specified value. Temporary directory settings are stored in different files from the other option settings. These files are located in the same directories as the configuration files for the other option settings. The following table shows the directories and files used to store temporary directory settings.

Config File	**Level Name**	**File Name**	**Windows Directory Path**	**Linux Directory Path**
Host-dependent, user-specific temporary directory	user_machine	<*hostname*>.cfg	%UserProfile%\Documents\Ansoft\<*ApplicationName&Version*>\config	$HOME/Ansoft/<*ApplicationName&Version*>/config

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Config File	Level Name	File Name	Windows Directory Path	Linux Directory Path
Host-independent, user-specific temporary directory	user	default.cfg	"<*InstallationDirectory*>\ANSYS Inc\v252\AnsysEM\config"	<*InstallationDirectory*>/ansys_inc/v252/AnsysEM/config
Host-dependent, default temporary directory	install_machine	<*hostname*>.cfg		
Installation default temporary directory	install	default.cfg		

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- ***<hostname>*** is the name of the computer on which the Electronics Desktop software is installed
- **$HOME** is the user's home directory on Linux
- ***<ApplicationName&Version>*** is the product name (without spaces) followed by the four-digit year of the version, a decimal point, and the minor release number (such as ElectronicsDesktop2025.2)
- **%UserProfile%** is a Windows variable that represents the currently active user's profile (for example, C:\Users\JohnDoe)
- ***<InstallationDirectory>*** is the root folder where the Ansys software is installed (typically, C:\Program Files, on Windows, or /opt, on Linux)
- ***<Version>*** is the lower-case letter "v" followed by last two digits of the product version's year and the minor release number, without a decimal point (such as v252)

As with other options, the settings in these files have precedence in the following sequence: user_machine (highest precedence), user, install_machine, install (lowest precedence). The installer creates the file at the "install" level (default.cfg). The first time you start and then exit the application, the file at the "user_machine" level is created (<*hostname*>.cfg). The other files are only created if you use the UpdateRegistry tool to specify an option at the "user" or "install_machine" level. If the temporary directory is set to an empty string in a configuration file, then that setting is ignored.

Temporary Directory Configuration File Format

This section describes the format of temporary directory configuration files. The format is the same for files at all four levels: user_machine, user, install_machine, and install. These files are text files, so any text editor may be used to modify or create them.

An example temporary directory configuration file is shown below:

```
$begin 'Config'
tempdirectory='C:/TEMP/AnsysEM'
$end 'Config'
```

The temporary directory specified by this configuration file is C:/TEMP/AnsysEM.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important:

- The string containing the pathname of the temporary directory must be enclosed in single quotes.
- The forward slash (/) may be used as a directory separator on Windows and Linux. The backslash (\) may be used as a directory separator on Windows only.
- The backslash (\) is used as an escape character in the tempdirectory string. That it, this character removes the special meaning of the following character.
- The single quote character normally ends the tempdirectory string. The backslash may be used to remove this special meaning, and include a single quote in the string.
- To use a backslash as a directory separator on Windows, it must be escaped. That is, a double backslash "\\" is used to denote a single directory separator.
- On Windows, a UNC path normally begins with two backslash characters. In a tempdirectory string, each of these backslash characters must be doubled, so four consecutive backslashes "\\\\" are used in the config file.

Example: Config File with UNC

```
$begin 'Config'
tempdirectory='\\\\hostxyz\\TEMP\\abc'
$end 'Config'
```

Here, hostxyz is a host with a sharename TEMP having subdirectory abc used as the temporarydirectory. This shows that four backslashes are required for UNC names and that backslashes used as directory separators must be doubled.

Example: Config File with Single Quote

```
$begin 'Config'
tempdirectory='C:/TEMP/ab\'cd'
$end 'Config'
```

Temporary directory is C:/TEMP/ab'cd. This shows how to include a single quote in a tempdirectory pathname. It also shows that forward slashes may be used as directory separators on Windows.

UpdateRegistry: Setting or Removing Temporary Directory Values in Configuration Files

The UpdateRegistry command line tool, described above, may be used to view, add, change or remove the temporary directory setting from any of the temporary directory config files. The registry key for viewing or modifying the temporary directory is TempDirectory.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The -Get, -Set, and -Delete options are valid for viewing, adding, changing, or deleting a temporary directory setting.

The -GetKeys option does not list the temporary directory key.

Batchoptions Command Line Examples

The `-batchoptions` entries command line argument may be used to specify one or more batchoptions settings from the command line. To specify multiple entries using a single `-batchoptions` argument, the entries should be enclosed in double quotes. Alternatively, the batchoptions may be specified in a file using the `-batchoptions <filename>` command line argument format. In this case, the filename is an absolute or relative pathname of the file containing the batchoptions, as described above.

> **Important:**
>
> The two approaches may not be combined: either all batchoptions must be in a file or all batchoptions must be specified explicitly on the command line.

Batchoptions File Format

> **Note:**
>
> Functionality featured in the example in this section applies to multiple design types.

An example batchoptions file is shown below:

```
$begin 'Config'
   'Desktop/ProjectDirectory'='C:/test/projects'
   'Desktop/Settings/ProjectOptions/NumberofProcessors'=2
$end 'Config'
```

Additional notes on the file format:

- The file may contain an arbitrary number of batchoption entries, one per line.
- Leading whitespace on each line is ignored. Spaces or tabs may be used to make the file more readable.
- The **Registry Key** appears before the equals sign (=) on each line and must be enclosed in single quotes ('). The registry key includes the key path and the name of the registry value.
- The registry value (or option value) appears after the equals sign on each line.
- Registry keys are case-insensitive.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- There are two supported types of registry values—string and integer:
  - Each **string** value must be enclosed in single quotes (').
  - Do **not** enclose **interger** values in quotes.
- For file paths within string values, the forward slash (/) may be used as a directory separator on both Windows and Linux systems.
- Alternatively, on Windows only, the customary backslash (\ ) may be used as a directory separator. However, in string values, the backslash is used as an escape code for indicating special characters that cannot be typed directly (such as \n for a new line). Therfore, if you use the backslash as a directory path separator, each instance must be doubled (\\ ). An example is: '\\\\host3\\temp\\Ansoft'. In this case, each double backslash is interpreted as a single backslash (\\host3\temp\Ansoft).
- The single quote character (') normally ends a string value. If you need to include a single quote (or apostrophe) within a string, use the backslash-apostrophe (**\'**) escape sequence. For example, the string '%UserProfile%/Documents/Ansoft/John\'s_Files', is interpreted as: %UserProfile%/Documents/Ansoft/John's_Files.

Example -BatchOptions with -Remote (Windows)

> **Note:**
>
> Functionality featured in the examples in this section applies to multiple design types.

In this example, we run a batch HFSS analysis of project file project1.aedt. We want all temporary files and directories to be created in C:\temp\HFSS instead of using the default temporary directory. We decide that the analysis will be done on a remote host, at IP address 12.34.56.78. Because of limited memory on the remote host, we decide to run the analysis using only a single COM engine. Because the remote host has four cores, we decide to use four four processors for of the analysis. We can use the **-Remote** option to specify that there will be a single remote COM engine.

Here is a sample command line for this analysis, where the project file \\somehost\projects\project1.aedt is located in a shared directory specified using a UNC path:

```
ansysedt -BatchSolve -Remote -Machinelist list=12.34.56.78
 -batchoptions "'TempDirectory'='C:/temp/HFSS'
'Desktop/Settings/ProjectOptions/NumberofProcessors'=4"
\\somehost\projects\project1.aedt
```

An alternative is to use the **-Distributed** command line option. Because the -Machinelist list contains only one host, there is a single remote COM engine in this case also.

```
ansysedt -BatchSolve -Distributed -Machinelist list=12.34.56.78
 -batchoptions "'TempDirectory'='C:\\temp\\HFSS'
 'Desktop/Settings/ProjectOptions/NumberofProcessors'=4"
 \\somehost\projects\project1.aedt
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The second line of the first example shows that you can use the forward slash "/" as a Windows directory separator within option value 'strings'. In this case, it is used in the TempDirectory path. You can also use the customary backslash "\" as a Windows directory separator, but it must be doubled to "\\" because the backslash is also an escape character within parameter strings. This usage is demonstrated in the second line of the second example, again in the TempDirectory path.

Example -Batchsolve with -Machinelist (Windows)

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

Suppose that we want to run a batch HFSS analysis of project file project1.aedt. Because all of our hosts have multiples of 2 cores, we specify that we will use two threads for multiprocessing for both the distributed (HFSS/NumCoresPerDistributedTask) and non-distributed (Desktop/Settings/ProjectOptions/NumberOfProcessors) parts of the job. The analysis contains a sweep that will be distributed across three hosts: ahmed, bill, and catherine. The hosts ahmed and bill have four cores each, so we run two distributed COM engines on each of these hosts, each using two threads. Host catherine has only two cores, so we specify only one distributed COM engine on this host. This COM engine will also use two threads. We specify a desired RAMLimitPercent of 75 for this analysis.

Here is a sample command line for this analysis, where the project file \\dennis\projects\project1.aedt is located in a shared directory specified using a UNC path:

```
ansysedt -BatchSolve -Distributed
-Machinelist list=ahmed,ahmed,bill,bill,catherine
-batchoptions "'Hfss/RAMLimitPercent'=75
'Desktop/Settings/ProjectOptions/NumberOfProcessors'
'Hfss/NumCoresPerDistributedTask'=2"
\\dennis\projects\project1.aedt
```

Example -Batchsolve with -Machinelist (Linux)

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

In this example, we run a batch HFSS analysis of project file project2.aedt. We have four identical host computers—host1, host2, host3, and host4 for analysis, and each host has 4 cores. We do not use multiprocessing for the distributed analysis, so NumCoresPerDistributedTask=1. As each host has four cores, we specify multiprocessing using

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4 threads for the non-distributed part of the analysis, so NumberOfProcessors=4. Because we do not use multiprocessing for the distributed analysis, we will run four distributed COM engines on each host, with a singe core available for each engine. We will specify a RAM limit of 75% for this analysis.

Here is a sample command line for this analysis, where the project file /home/jsmith/projects/project2.aedt is located in a shared directory:

```
hfss -BatchSolve -Distributed
-Machinelist file=/home/jsmith/hosts/list2
-batchoptions "HFSS/'RAMLimitPercent'=75
'Desktop/Setings/ProjectOptions/NumberOfProcessors'=4
'HFSS/NumCoresPerDistributedTask'=1"
/home/jsmith/projects/project2.aedt
```

For this example, the hostnames are in the text file /home/jsmith/hosts/list2. The file contents are as follows:

```
host1
host1
host1
host1
host2
host2
host2
host2
host3
host3
host3
host3
host4
host4
host4
host4
```

Example -Batchsolve for Local (Windows)

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this example, we run a batch analysis of project file testproject.adsn on the local host. We want all temporary files and directories created in directory C:\temp\ansysedt instead of using the installation default temporary directory. Because the local host has four cores, we decide to use four threads for multiprocessing, for both distributed and non-distributed parts of the analysis.

Here is a sample command line for this analysis, where the project file \\host123\projects\testproject.adsn is located in a shared directory specified using a UNC path:

```
ansysedt -BatchSolve -Local -batchoptions
"TempDirectory='C:/temp/ansysedt'
'HFSS/SolverOptions/NumProcessors'=4
'HFSS/SolverOptions/NumProcessorsDistrib'=4"
\\host123\projects\testproject.adsn
```

Note that the batchoptions pathnames 'HFSS/SolverOptions/NumProcessors' and 'HFSS/SolverOptions/NumProcessorsDistrib' must be in single quotes because they both contain embedded spaces.

Batch Options and Analysis Configurations in the Registry

Analysis configurations are used to specify machines and options for local, remote, and distributed analysis, including capabilities that are enabled by HPC licenses.

How Analysis Configurations are Stored in the Registry

A configuration contains information beyond the machine or machines to use for a solution. Examples are the number of processors to allocate to the analysis for each machine in the list, memory limits, directory locations for personal libraries and temporary files, and many other preferences.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- Options are arranged as keys and values (in a structure similar to the *Windows Registry*). However, these options are not a part of the *Windows Registry* but are separately stored and maintained by the Ansys Electronics Desktop software. For more information concerning the configuration files comprisisng the options registry, see the following topics:

  Setting Options Via Configuration Files
  Setting the Temporary Directory

- For access to options and functionality beyond what is directly accessible via the user interface or batch options, refer to the documentation of the **UpdateRegistry** tool. This tool is discussed in the following help topic and in the topics that follow it in the same branch of the product help:

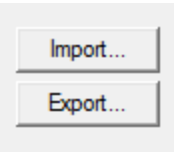
  Setting or Removing Option Values in Configuration Files: UpdateRegistry Command

General settings are associated with the Desktop application or 3D Editor. "HPCLicenseType" and "tempdirectory" values are at the root level of the registry. Other options are specific to a particular product or design type. For example, certain mesh, boundary, and memory limit settings as well as many other preferences are product-specific and are therefore associated only with the applicable design types. Such options appear with a consistent value name but in a different registry path for each applicable design type.

Copying a Configuration from one Design Type or Product to Another

To copy a configuration from one design type (or product) to another:

1. Click **Tools** > **Options** > **HPC and Analysis Options**. The *HPC and Analysis Options* dialog box appears.
2. On the **Configurations** tab of the **HPC and Analysis Options** dialog box, use the **Export…** button to export the configuration to a file.



3. Switch to the destination design type (or product) and use the **Import…** button to import the configuration data.

   Any data that is not applicable to the destination design type is ignored; any settings present in the destination design type that were not present in the source configuration will be assigned default values. The user may then edit the copy, as desired.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using HPC and Analysis for Configurations

Due to the complexity of the registry values for the configurations we do not recommend directly editing these values using the *UpdateRegistry* tool. Instead, use the **HPC and Analysis Options** dialog box to edit or create a configuration. (See: Setting HPC and Analysis Options.) Configurations created or edited using the GUI are stored in the **user_machine** level of the registry. You can create a configuration for one of the other registry levels using the following steps:

1. Create the configuration using the GUI, then export the configuration to a file.
2. Delete the configuration using the GUI so that it will not be present in the **user_machine** level. Then, exit the GUI.
3. Use the **UpdateRegistry** tool to import the data into the desired registry level using the **-FromFile** option to specify the file exported via the GUI, and using the **-RegistryLevel** option to specify the registry level where the configuration is to be stored. For example, an administrator may use this approach to create a configuration at the **install** level that may be used by any user on any machine.

Batch Options

There is a large number of both general and product-specific options supported by the software. These options have evolved over time. Therefore, older batchoptions files may no longer be valid, and the options listed in the user interface of the current software may differ from earlier versions you have used. Additionally, there are options beyond those listed within the user interface (that is, the UI provides a subset of the available options). You can generate a more complete list of options by running the UpdateRegistry tool with the -GetKeys switch and piping the output to a text file, as detailed in the following Windows procedure:

1. In a command window, navigate to the following folder:

   "<*installation_directory***>\ANSYS Inc\v252\AnsysEM"**

2. Type and enter the following command:

   UpdateRegistry -GetKeys -ProductName ElectronicsDesktop20*xx.y* **>** <*text_file_path*>**\Batchoptions.txt**

   Substitute the last two digits of the installed product version year and the minor release number for *xx.y* (such as **25.1**). Also, substitute the desired *text_file_path* (such as **%UserProfile%\Desktop**).

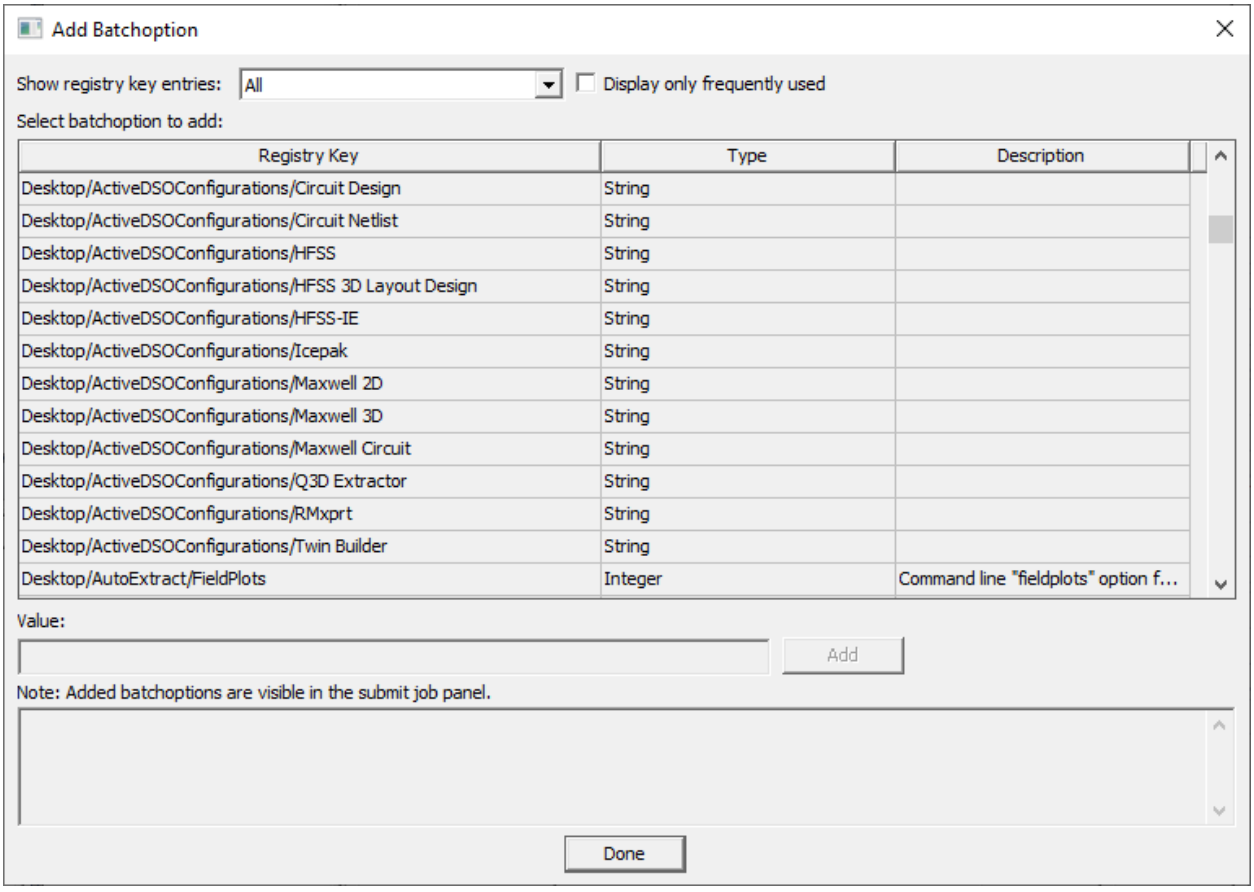
   The resulting **Batchoptions.txt** file will contain a nearly complete list of available options.

This procedure also works from the Linux <*installation_directory*>, which has the same downstream path (for example, under "opt/") as a Windows installation (typically under "C:\Program Files\").

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

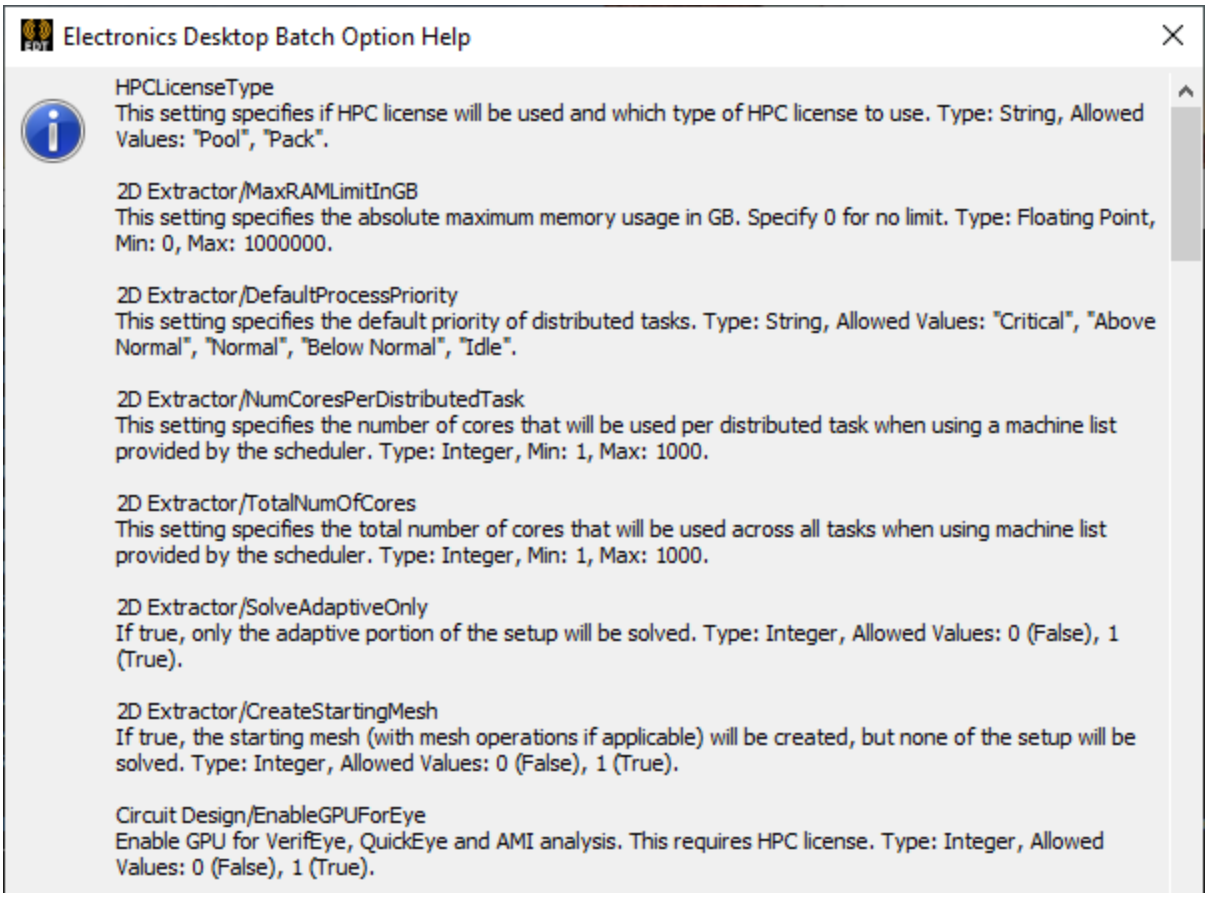
> **Note:**
>
> For special product-specific options that are neither available from the GUI nor listed by the UpdateRegistry tool, see Special Batch Options. Even though the -GetKeys switch does not list these special options, you can still use the UpdateRegistry tool to set them.

When you submit jobs to a remote computer or cluster, you can specify batch options using the job submission GUI. When using the GUI, you select the batch options from a list and therefore avoid typographical errors. For the most commonly used batch options, there is detailed information about the allowed values. Click **Submit** on the **Simulation** ribbon tab to access the *Submit Job To* dialog box. Then, click **Add** to access the **Add Batchoption** dialog box pictured below:



To assist users who need to specify batch options and are unable to use the job submission GUI, a new help option has been added. If the Electronics Desktop application is launched with the **-batchoptionhelp** command line argument, a message box is displayed which lists and describes the most common design type-specific batch options:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Managing Projects and Designs

An Ansys Electronics Desktop *project* is essentially a folder that includes one or more models, or *designs*.

Some basic tasks that can be performed on projects and designs include:

- Opening Projects
- Creating Projects
- Saving Projects
- Importing and Exporting Projects and Data
- Setting Project Options
- Validating Projects
- Copying and Pasting Projects and Designs
- Renaming Projects
- Setting Up a Maxwell Design
- Deleting Projects or Designs

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

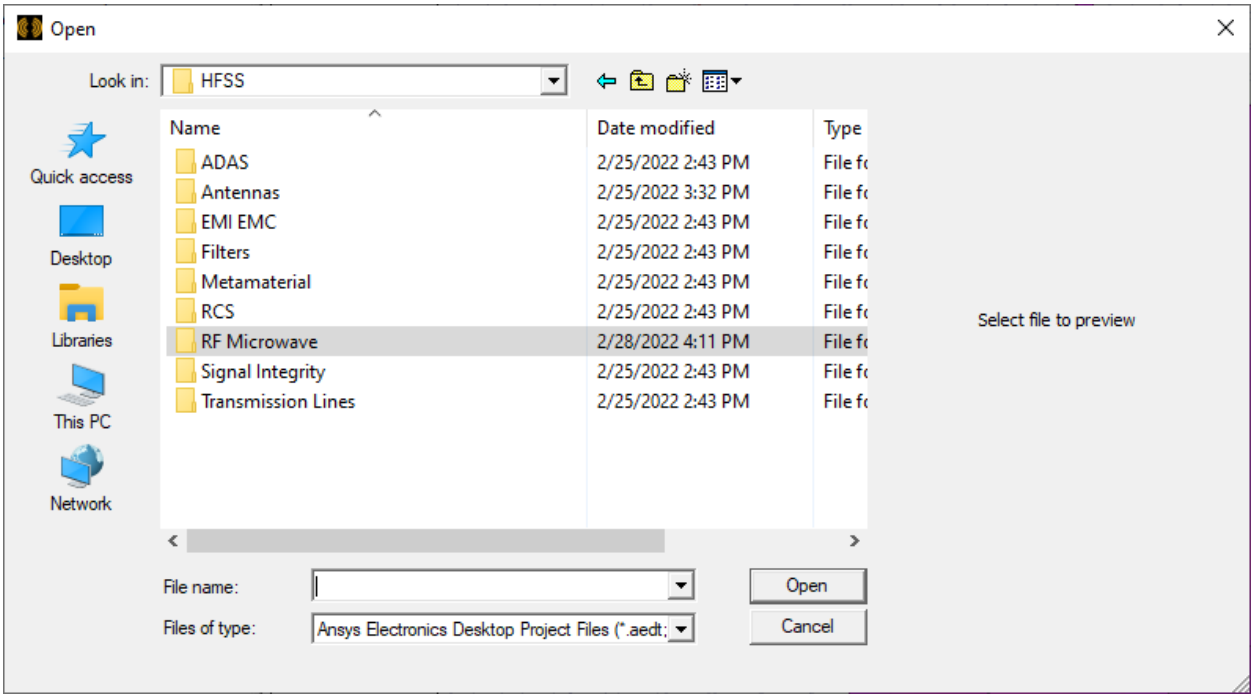
- Setting Read Only Designs
- Updating Design Components
- Undoing and Redoing Commands
- Inserting a Documentation File
- Saving Project Notes
- Printing

Opening Projects

Open a previously saved project using the **File** > **Open** command.

1. Click **File** > **Open** or click the **Open** icon on the **Desktop** tab of the ribbon.

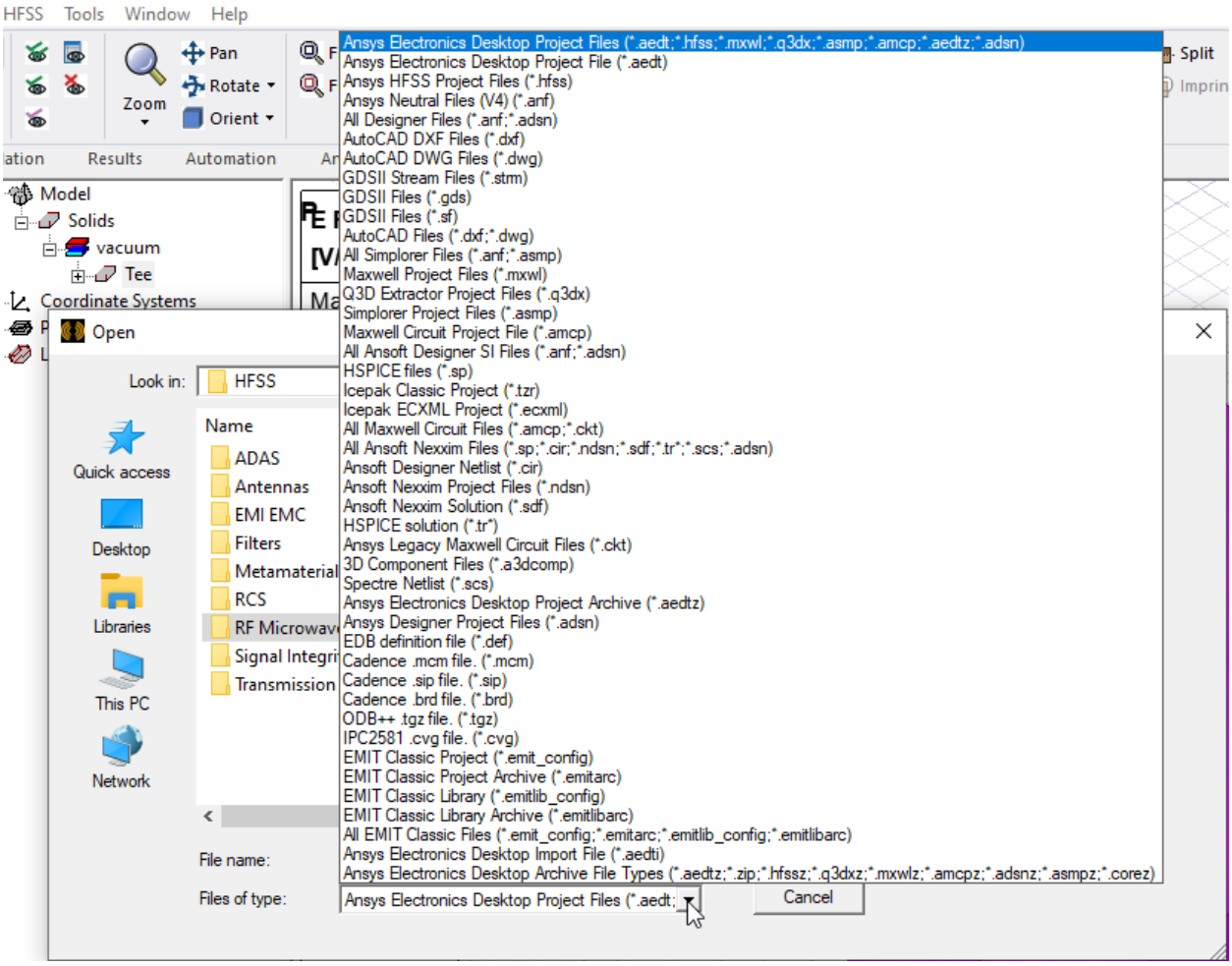
2. Use the file browser to find the project file.



By default, only files that can be opened or translated display. *.aedt is the main default, although, you can also view several types of legacy files. The dropdown for Files of type

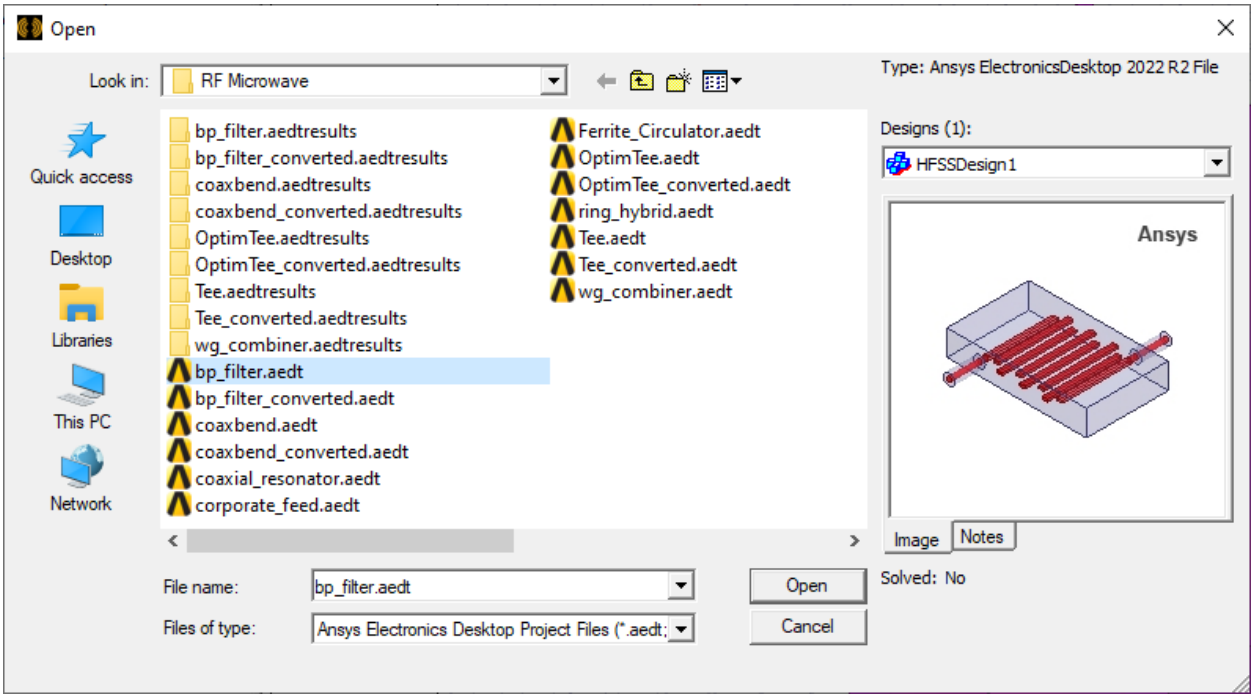
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

lists the kinds file and archive formats for which you can select filters.



3. Most of these will be familiar to those seeking them. The Ansys Electronics Desktop Import file (*.aedti) format is for importing exported projects from ANSYS Discovery, an ANSYS Workbench tool. This format allows import of side by side *.aedti and the corresponding geometry definition *.sat into the Ansys Electronics Desktop, and creates a ready-to-solve HFSS project.
4. Select the file you want to open.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Click **OK**.

   The project information appears in the Project tree.

If you open another project without editing the automatically created project, the automatically created project is removed.

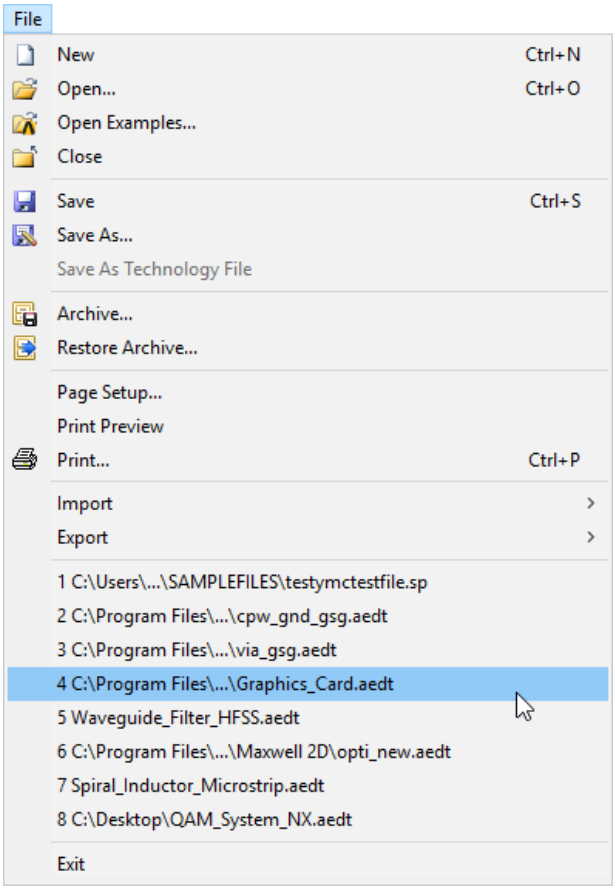
You can also open a saved project by:

- Dragging a project file icon to the Open icon
- Dragging a project file icon to the desktop
- Double-clicking a project file icon

Opening Recent Projects

Recently opened projects appear at the bottom of the **File** menu. To open one of these files, click its file name.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Opening Example Projects

You can directly access and open example projects included with product installation using **File** > **Open Examples**, or by clicking **Open Examples** on the **Desktop** tab of the ribbon. See Example Projects for additional description of these files.

Opening Legacy Maxwell Projects

When you open a legacy Maxwell project, virtually all of the project's preprocessing data is translated. Solution results and Optimetrics setup data are unavailable; however, the nominal model created for Optimetrics can be translated.

Objects that were considered non-model in the legacy project (**Model Object** check box was cleared in the Properties dialog) are translated but remain non-model objects in the translated project.

Following are additional notes about the translation of legacy project information:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Model Geometry	• The translated geometry's construction history is unavailable; therefore, the original object properties you defined cannot be modified in the **Properties** window. However, you can modify the geometry using Maxwell's modeling features. • For units unavailable in the current version of Maxwell, such as yards, the nearest available units are used; the model is scaled slightly to fit the new units.
Excitations and Boundaries	• Boundaries assigned to named interface selections or rectangle selections are not translated. • For a boundary assigned to the intersection of two faces, Maxwell 16 will create a new 2D sheet object from the intersecting area and assign the boundary to that object.
Materials	• Functions defined in legacy projects become project variables in version 12 and later; therefore, functional material properties and functional material assignment angles are *not* translated. • Perfect conductors become regular materials with conductivity values of 1E30. Maxwell 12 and later uses a conductivity threshold that treats all conductors over the threshold as perfect conductors. • Object coordinate systems are created for objects assigned anisotropic materials in legacy projects. The coordinate system is defined at the same origin as the global coordinate system, with the same orientation defined when the anisotropic material was assigned to the object in the legacy project. • Nonlinear materials from legacy projects that have magnetic saturation values greater than zero are treated as ferrite materials in version 16. Their properties are not modified. • When translating a legacy project, Maxwell does not import any variables that depend on intrinsic variables (this feature is unavailable in Maxwell 12 and later versions). For material properties depending on intrinsic variables, the variable reference is replaced with its expanded expression.
Mesh Operations	• Mesh refinement operations performed on arbitrary boxes in legacy projects are ignored. • Area-based and volume-based mesh operations are translated as length-based mesh operations in version 12 and later by taking their square roots and cube roots, respectively.
Optimetrics	• Setup information, including design variables, is not supported; however, the nominal model can be translated. • Parameterizing a translated model is limited because geometry construction history is unavailable.
Solution Setup	• The design's initial mesh is used for the solution. Meshes are not translated.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solutions	• Solution data is not translated; therefore, you must solve legacy Maxwell projects again.

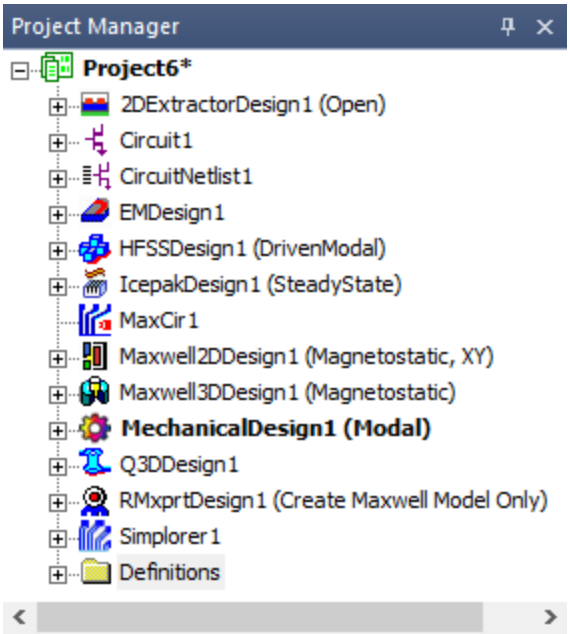
Contact Ansys Technical Support for assistance in converting legacy projects.

Creating Projects

To create a project, click **File** > **New**, or click the **New** icon on the **Desktop** tab of the ribbon.

A new project is listed in the project tree. It is named Project*n* by default, where *n* is the order in which the project was added to the current project folder.

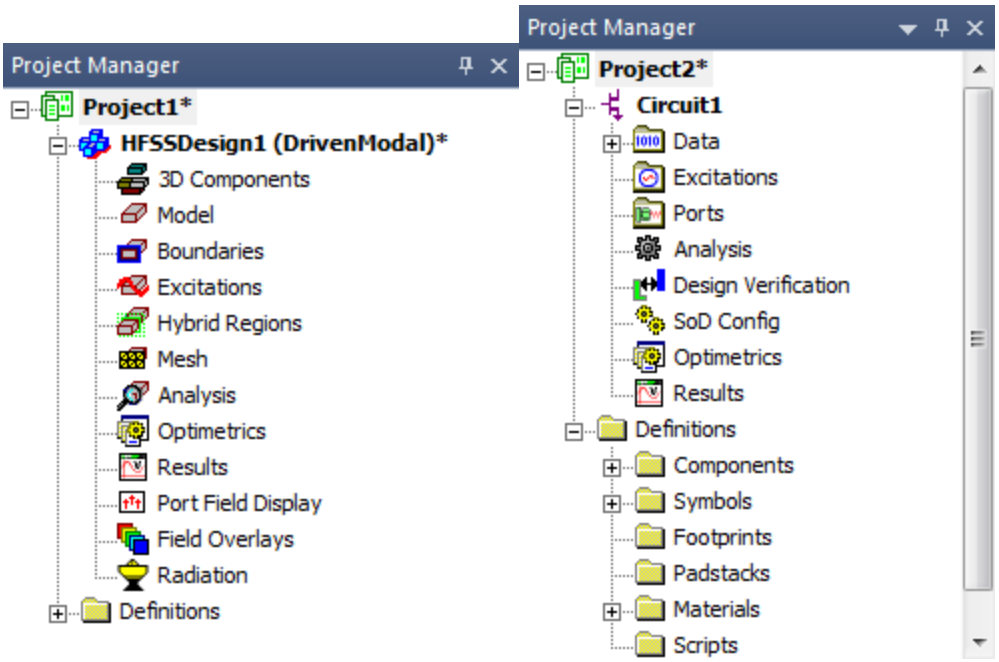
You can insert designs of any type into the project, where they are stored in the project tree.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The default name for each inserted project is <designType>*n*. You can also specify the name of the project when you save it using the **File** > **Save** or **File** > **Save As** commands.

You can view the contents of a project by clicking the plus sign (+) for each level of the hierarchy in the project or design.



Creating New 2D Designs from 3D Designs

Maxwell supports both 2D and 3D projects. Often, you may want to take a cross-section of a 3D model to perform a 2D analysis. The following cases represent the scenarios available for automatically converting models.

- 3D model to 2D XY model

  3D model to 2D RZ model

Keep the following points in mind before converting any project:

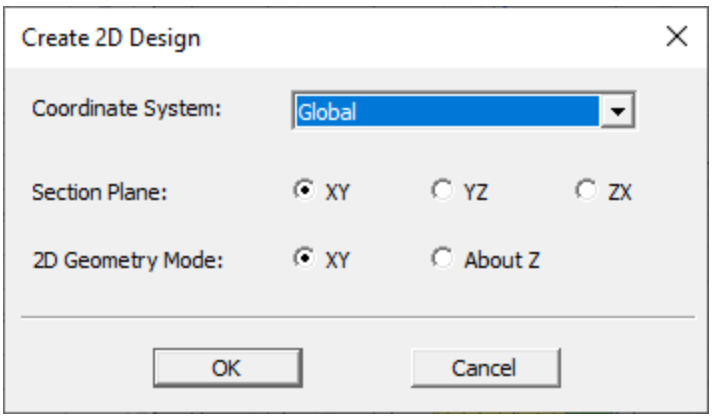
- A 2D design is created in the same project.
- All 3D geometry is used. You cannot select which geometry to include in the conversion.
- If there are any sheet objects in 3D, only those sheets that lie in the same plane as the section plane are converted.
- 1D objects are not converted from 3D to 2D.

To create a 2D design from an existing 3D design:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Maxwell 3D** > **Create 2D Design**.

   The **Create 2D Design** window appears.



2. Select the **Coordinate System** to be used for creating sections in the 3D model.
3. Choose the **Section Plane** within the chosen coordinate system. You can choose XY, YZ, or ZX.
4. Select the **2D Geometry Mode**, to use for the target 2D design. You can choose either XY or About Z.
5. Click **OK** to create the 2D design.

   The modeler creates a new design within the current project and generates the 2D model within the new design. All material properties and material assignments are copied to the new design.

Creating New 3D Designs from 2D Designs

Maxwell supports both 2D and 3D projects. Often, you may want to sweep a 2D model to create a full 3D design. The following cases represent the scenarios available for automatically converting models from 2D to 3D.

- 2D XY model to 3D model
- 2D RZ model to 3D model

Keep the following points in mind before you convert any project:

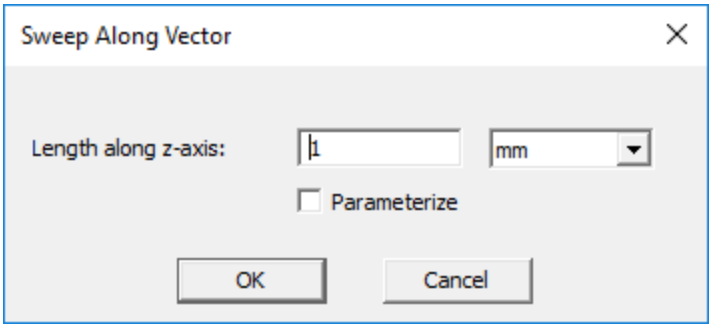
- A 3D design is created in the same project.
- All 2D geometry is used. You cannot select which geometry to include in the conversion.
- 1D objects are converted from 2D into 3D sheet objects.

To create a 3D design from an existing 2D XY design:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Maxwell 2D** > **Create 3D Design**.

   The **Sweep Along Vector** window appears.



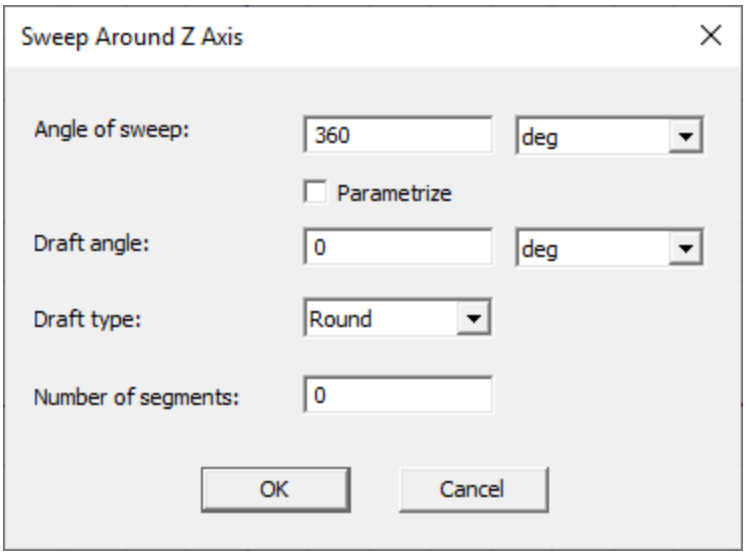
2. Enter a value in the **Length along z-axis** box to obtain a swept length of the model.
3. Choose the desired units of the sweep distance entered from the drop-down menu.
4. Click **OK** to create the 3D design.

   The modeler creates a new design within the current project and generates the 3D model within the new design. All material properties and material assignments are copied to the new design.

To create a 3D design from an existing 2D RZ design:

1. Click **Maxwell 2D** > **Create 3D Design**.

   The **Sweep Around Z Axis** window appears.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Enter the **Angle of sweep** and desired unit of measure to obtain a full 360° 3D model or a wedge model of the desired angular size.
3. Enter the desired **Draft angle** and **Draft type**. The draft angle is the angle to which you want a 2D object's profile, or shape, expanded (positive angle) or contracted (negative angle) as it is swept. The draft type instructs the modeler whether to maintain the original object shape and how to transition between adjacent faces. (Refer to "Draft Type Options" on page 7-266 for more details.)

   > **Note:**
   >
   > Entering a draft angle for a model where objects touch the z axis results in a error due to an invalid object being requested. Draft angles may only be used when no objects are touching the z axis.

4. Enter the **Number of segments** to use in creating the 3D model. If you do not change the default value (=0), the design will be created with true surfaces.

   Optionally check **Parametrize** for parametric analysis.

5. Click **OK** to create the 3D design.

   The modeler creates a new design within the current project and generates the 3D model within the new design. All material properties and material assignments are copied to the new design.

Creating a Target Design from 3D Designs

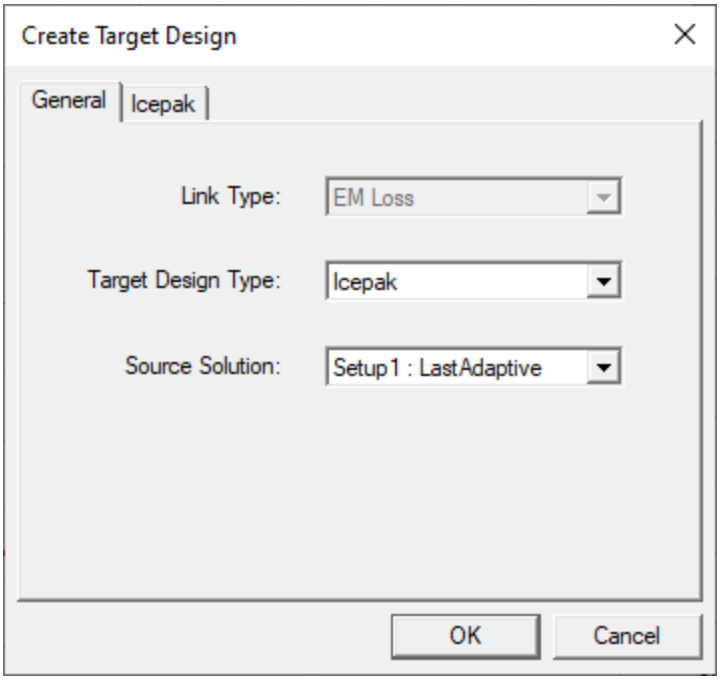
In an HFSS, Q3D, or Maxwell 3D design, **Create Target Design** automatically generates and inserts a thermal design (Icepak or Mechanical Thermal) into your project.

To create a target design from an existing **Maxwell 3D** design:

1. Click **Maxwell 3D** > **Create Target Design**.

   The **Create Target Design** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select the **Target Design Type** to be created.
3. Select the **Source Solution** to specify the solution data to be linked to the target design.
4. For Icepak designs, specify **Forced Convection** or **Natural Convection** on the **Icepak** tab.
    - For **Forced Convection**, specify **Flow Speed** and **Flow Direction**.
    - For **Natural Convection**, specify the **Gravity Vector Direction**.
5. Click **OK**.

Icepak Designs

The Icepak design is generated based on the source design and selections in the **Create Target Design** dialog box:

- Model geometry is created with corresponding material assignments. Vacuum objects are not created in target Icepak designs.
- Any assigned materials that do not contain thermal properties are assigned default thermal properties in Icepak.
- The Region (computational domain) is created with the appropriate dimensions.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- An EM loss excitation is assigned to the geometry, and variable mapping and intrinsic variables are linked to the source design.
- Design settings are specified.
    - Solution type: **Steady State**
    - Problem type: **Temperature and Flow**
- For forced convection simulations, the following are defined:
    - A free opening boundary condition with a velocity specified is assigned to the Region face to create forced flow.
    - A free opening boundary condition is assigned to the face of the Region opposite to the opening with velocity specified.
    - In the **Icepak Solve Setup** dialog box, forced convection **Solve Setup Defaults** are defined.
- For natural convection simulations, the following are defined:
    - **Gravity Vector**
    - Free opening boundary conditions are assigned to each face of the air Region.
    - In the **Icepak Solve Setup** dialog box, natural convection **Solve Setup Defaults** are defined.

Mechanical Designs

The Mechanical Thermal design is generated based on the source design:

- Model geometry is created with corresponding material assignments. For any vacuum objects, **Solve Inside** is disabled.
- Any assigned materials that do not contain thermal properties are assigned default thermal properties in Mechanical.
- A convection boundary is assigned to the faces of all 3D objects (not 2D sheet objects). **A Film Coefficient** is specified as **Uniform** at 10 w_per_m2kel. Objects with **Solve Inside** disabled and those with air assigned as the material are not assigned to the convection boundary.
- An EM loss excitation is assigned to the geometry, and variable mapping and intrinsic variables are linked to the source design.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Saving Projects

Use the **File** > **Save As** command or select the **Desktop** tab of the Ribbon and click the **Save As** icon to:

- Save a new project.
- Save the active project with a different name or in a different location.
- Save the active project in another file format for use in another program.

Use the **File** > **Save** command or select the **Desktop** tab of the Ribbon and click the **Save** icon to save the active project.

> **Warning:**
>
> Be sure to save models periodically. Saving frequently helps prevent the loss of work if a problem occurs.
>
> Although Ansys Electronics Desktop has an auto-save feature, it may not save frequently enough for your needs.

Each solver has a Save Before Solving setting located in the **Tools** > **Options** window. This setting is enabled by default.

A prompt appears when you attempt to save a previously versioned file. If you agree to the prompt, the file is upgraded to the Ansys Electronics Desktop version in which you are running the software. In this case the file may no longer be compatible with previous versions. If you do not agree to the prompt, the file is not saved, so the file retains its previous compatibility.

If you have a simulation running, you see a warning that if you continue, Ansys Electronics Desktop will abort the simulation. If you OK the warning, Ansys Electronics Desktop aborts the simulation and saves the project.

Path Name Length Issues for Windows

For most Windows programs, the current directory pathname length is limited to 259 characters on startup. Thus, the pathname length of the directory containing installed programs should be limited to no more than 259 characters because double-clicking on an application in Windows

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Explorer will set the working directory to the directory containing the application when the application is started.

Win32 Long Paths Not Enabled

If win32 long paths are not enabled on the analysis host, then essentially all files are limited to a maximum absolute pathname length of 259 characters. Directories may be limited to a maximum absolute pathname length of about 246 characters.

Win32 Long Paths Are Enabled

If win32 long paths are enabled on the analysis host, then many files are not subject to the maximum absolute pathname length limit of 259 characters. Below is a partial list of files that are still subject to this limit even if win32 Long Paths are enabled:

- Project files
- Project Archive files
- Ansys EDB (Electronics Database) files, typically stored in the `ProjectName.aedb` folder
- The temporary directory and most temporary files

  Because some temporary files and directories have automatically generated names that are long, it is best to use a short pathname for the temporary directory.

  > **Important:**
  >
  > Although most temporary files are created within the temporary directory, there may be some temporary files within the project directory. As a result, the project file directory pathname should be well below the 259 character limit.

To enable Long Path support on Windows, both of the following requirements must be met:

- Long paths must be enabled on the machine

  Use either of two ways to do this:

  - **Via Registry Setting:** `HKLM\SYSTEM\CurrentControlSet\Control\FileSystem : LongPathsEnabled=1`
  - **Via Group Policy Tool:** The “Enable Win32 long paths” setting is in the folder: `Computer Configuration\Administrative Templates\System\Filesystem`

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Changing Auto-Save Settings

Recent actions performed on the active project are stored in an auto-save file in case of a sudden workstation crash or other unexpected problem. The auto-save file is stored in the same directory as the project file and is named Project*n*.aedt.auto by default. Ansys Electronics Desktop automatically saves all data for the project to the auto-save file, except solution data.

By default, Ansys Electronics Desktop automatically saves project data after every 10 edits. An edit is any action performed that changes data in the project or the design, including actions associated with project management, model creation, and solution analysis.

After a problem occurs, you may be able to choose to re-open the original project file in an effort to recover the solution data, or to open the auto-save file. If the original file is not available, attempting to open the file provides a message that the auto-save is being used. If neither file is available, an error message displays.

To modify auto-save settings:

1. Click **Tools** > **Options** > **General Options**.

   The **Options** window appears.

2. Under **Desktop Configuration**, verify that **Do Autosave** is selected.

   This option is selected by default.

3. In the **Autosave interval** box, enter the number of *edits* that you want to occur between automatic saves. By default, this option is set to 10.

   > **Note:**
   >
   > Auto-save *always* increments forward; therefore, even when you undo a command, Ansys Electronics Desktop counts it as an edit.

4. Click **OK** to apply the specified auto-save settings.

   Once the specified number of edits is carried out, a "model-only" save will occur. This means that Ansys Electronics Desktop does not save solutions data or clear any undo/redo history. When Ansys Electronics Desktop auto-saves, an ".auto" extension is appended to the original project file name. For example, Project1.aedt will automatically be saved as Project1.aedt.auto.

> **Warning:**
>
> When you close or rename a project, Ansys Electronics Desktop deletes the auto-save file.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Save Before Solving Option

The **Save before solving** option forces a full save before running the solve and is enabled by default. For efficiency reasons, the project is saved only if it has been modified since its last save.

To change this setting, click **Tools** > **Options** to open the Options window. Select the **General** tab and use the **Save before solving** check box to toggle the option.

You can start a new solve while running another without having to abort the running solve. If you start a solve while another solve is running and the Save before solving option is set, Ansys Electronics Desktop asks if you want solve without saving first. This lets you perform multiple solves, and if you have not edited the project in between solves, crash recovery will work.

Recovering Project Data from an Auto-Save File

Following a sudden workstation crash or other unexpected problem, you can recover project data from its auto-save file.

> **Warning:** When you recover a project's auto-save file you *cannot* recover any solutions data; recovering an auto-save file means you will lose any solutions data that existed in the original project file.

To recover project data in an auto-save file, if Ansys Electronics Desktop containing a design has unexpectedly crashed:

1. Launch Ansys Electronics Desktop from your desktop.
2. Click **File** > **Open**.
3. Select the original Project*n*.aedt project file for which you want to recover its Project*n*.aedt.auto auto-save file.

   The **Crash Recovery** window appears, giving you the option to open the original project file or the auto-save file.

4. Select **Open project using autosave file** to recover project data in the auto-save file**, and then click OK.** Ansys Electronics Desktop replaces the original project file with the data in the auto-save file.

   Ansys Electronics Desktop immediately overwrites the original project file data with the auto-save file data, removing the results directory (solutions data) from the original project file as it overwrites to the auto-save file.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Warning:** If you choose to recover the auto-save file, you cannot recover the original project file that has been overwritten; recovering data in an auto-save file is *not* reversible.

Importing and Exporting Projects and Data

Ansys Electronics Desktop can import and export a variety of file types.

Importing Files

You can import the following types of files into a Maxwell design:

- 2D model (GDSII) files
- 3D model files
- DXF and DWG format files
- Data table files
- Plot data

Object, material, and parameter names with non-ASCII characters are not allowed, and therefore, not allowed for data transfer. Such transfers fail and produce an error message.

For certain file types, the import dialog contains **Validation and Healing Options**, which are enabled by default.

These types are:

- ACIS SAB files (*.sab)
- ACIS SAT files (*.sat)
- Ansys 3D Modeler files (*.sm3)
- Autodesk Inventor files (*.ipt, *.iam)
- CATIA V4/V5 files (*.exp, *.model, *.CATpart, *.CATproduct)
- Creo Parametric files (*.prt, *.asm)
- IGES files (*.iges, *.igs)
- JT files (*.jt)
- Nastran files (*,nas)
- NX files (*.prt)
- Parasolid files (*.x_t, and *.x_b)
- SOLIDWORKS Files (*.SLDPRT, *.SLDASM)
- STEP files (*.step, *.stp)

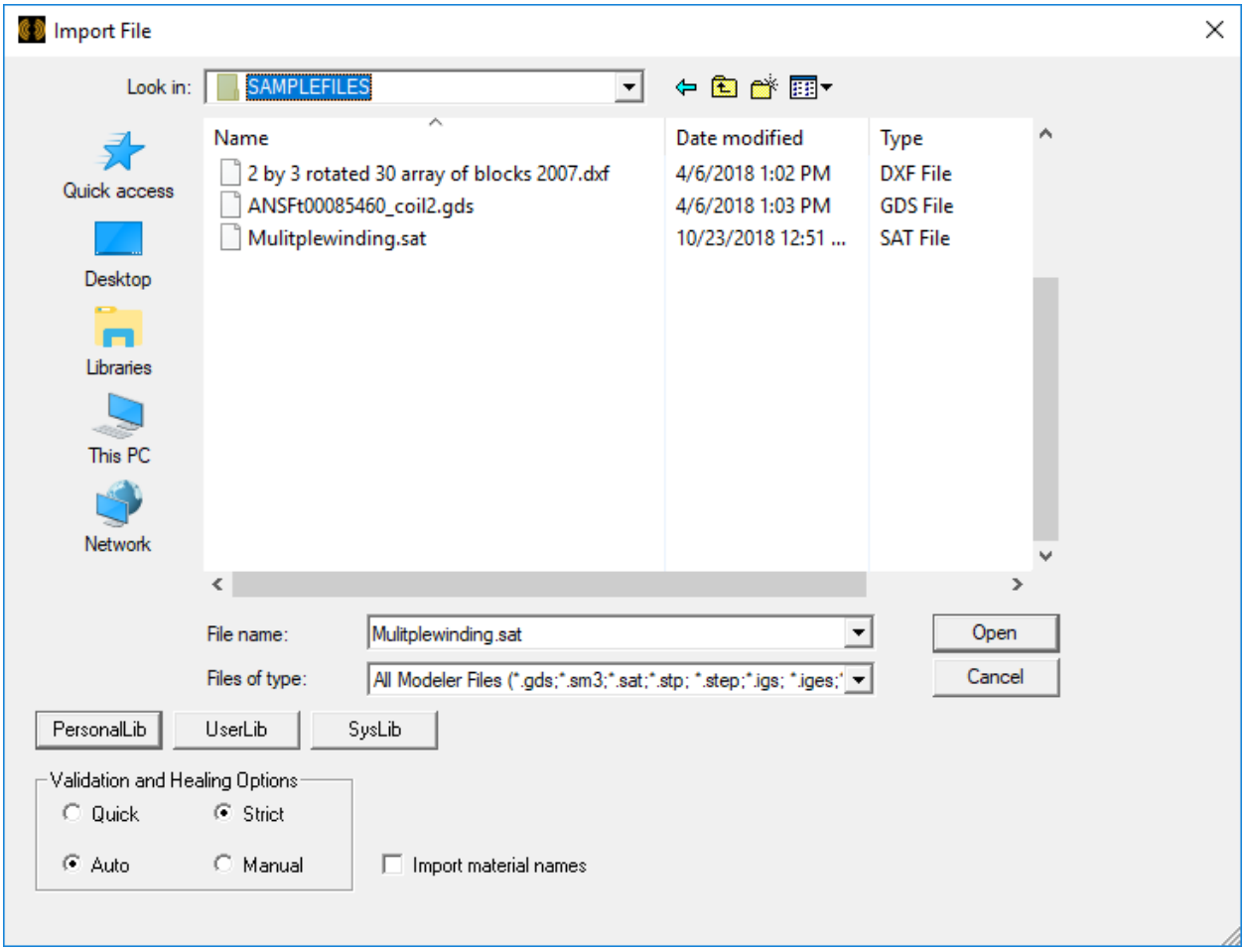
For more information, see Heal.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To import a file:

1. Click **Modeler > Import**.

   The **Import File** window appears.



   If applicable to the file type, **Validation and Healing Options** are enabled by default. You can change them as desired.

   You can also use the check box to choose whether to **Import material names**.

2. Click **Open**.
3. Depending on the file type, additional options may appear. Set these as appropriate.
4. Click **OK**.

   The file loads.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Importing 2D Model (GDSII) Files

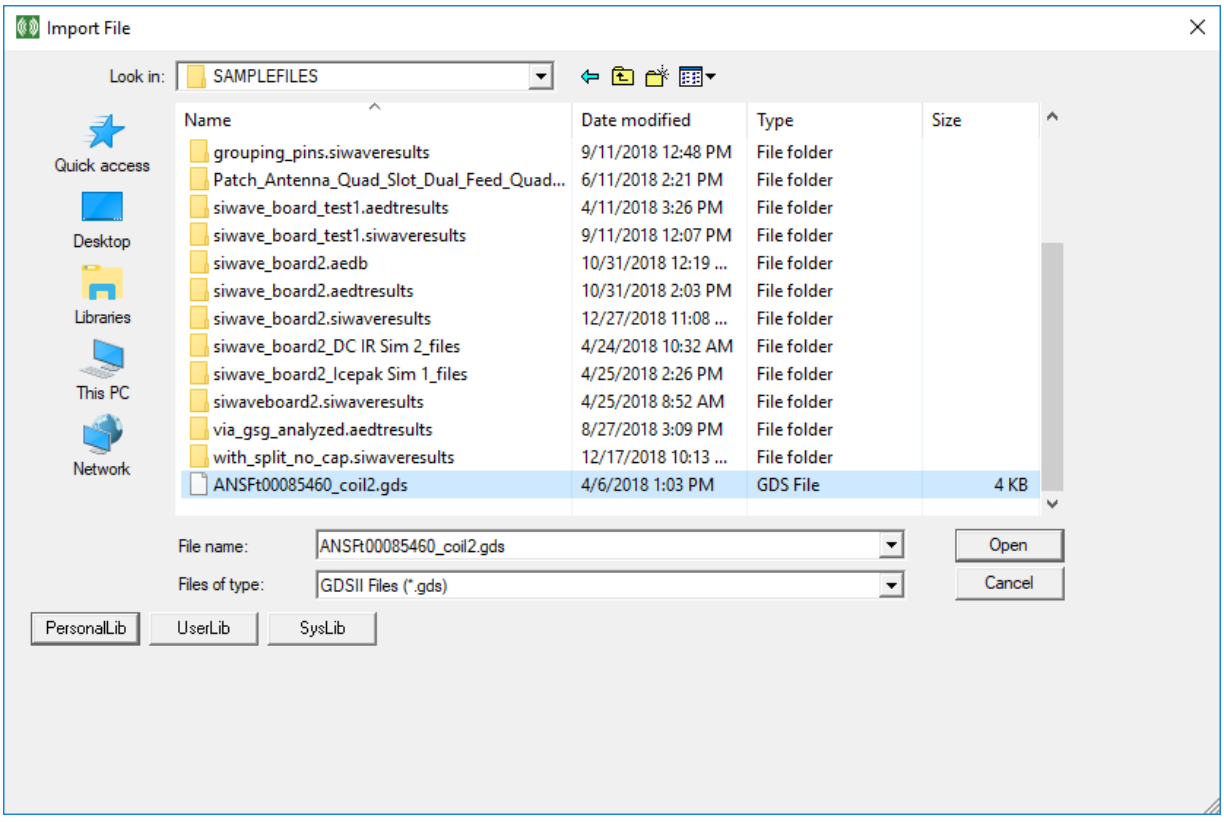
2D model files can be imported directly into the active **Modeler** window. GDSII is the industry standard file format for 2D graphical design layout data.

> **Note:**
>
> - Object, material, and parameter names with non-ASCII characters are not allowed for data transfer. Such transfers fail and produce an error message.
> - If you import a file into an active **Modeler** window that contains an existing model, the file is added to the existing model; it will not replace it.

To import a 2D model file:

1. Click **Modeler** > **Import**.

   The **Import File** dialog box appears.

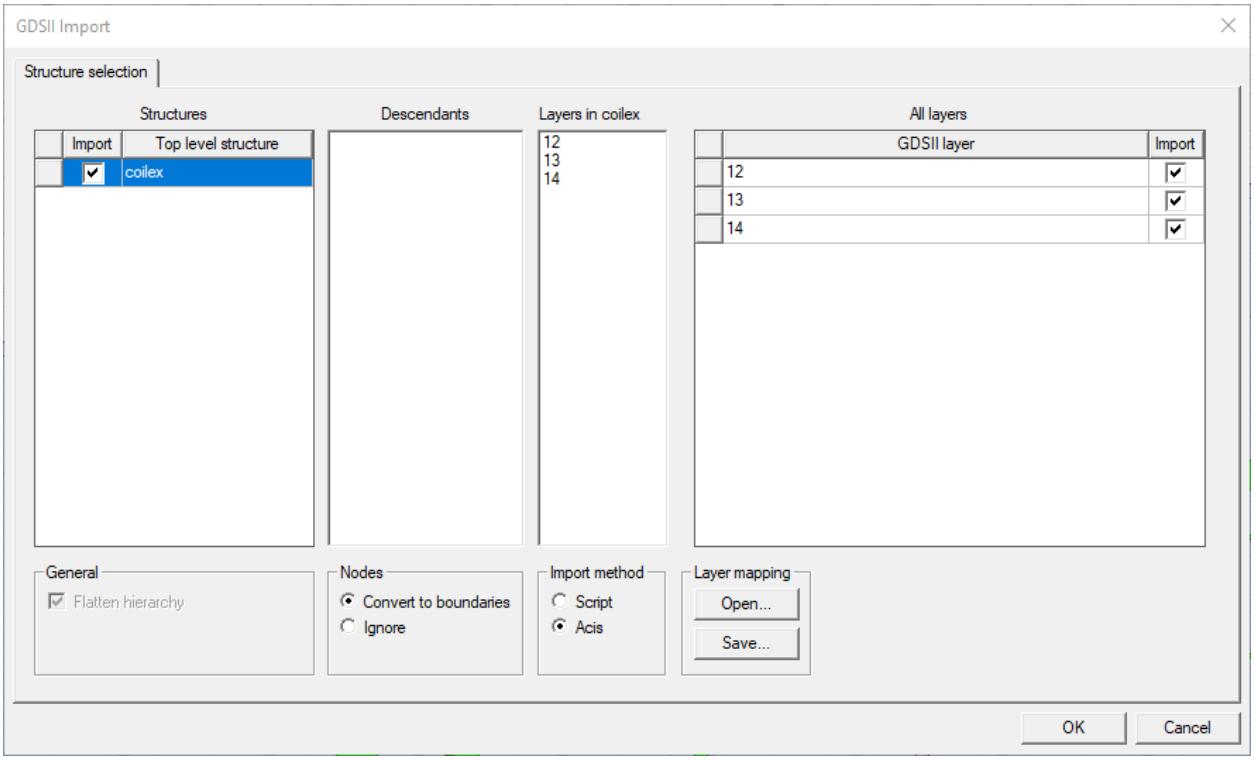


2. Select a file type from the **Files of type** drop-down menu. For 2D model files, this is **GDSII Files (*.gds)**.
3. Use the file browser to find and select the file you want to import.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **Open**.

   The **GDSII Import** window appears.



5. Use the panes to select your desired options:
   - **Structure Selection** – a GDSII file may contain several top-level structures. Use this pane to select structure(s) for import. When multiple structures are imported, Maxwell creates multiple designs under the current project, one for each of the GDSII structures.
   - **Descendants** – GDSII files are hierarchical and may contain many sub-layouts. This panel shows the sub-layouts in the selected top-level designs.
   - **Layers in [Structure Name]** – this panel's name changes based on the name of the selected structure, and displays that structure's layers, by layer number.
   - **All Layers** – lists all layers from all structures in the file. Use the **Import** check boxes to select or deselect layers for import.
   - **General** – the **Flatten hierarchy** check box is selected by default. Maxwell always flattens any hierarchical geometry in the GDSII.
   - **Nodes** – GDSII supports nodes and boundaries as separate data types. Normally, boundaries represent polygons. Maxwell can either convert objects that use the nodes data type to boundary types, or ignore them. Use the radio buttons to select either **Convert to boundaries** (default) or **Ignore**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Import Method** – select either Script or Acis.
- **Layer Mapping** – if desired, create a mapping of GDSII layer numbers to layer names in the design stackup.

6. Click **OK**.

   The file is imported into the active **Modeler** window.

Creating GDSII Layer Mapping Files

While importing a 2D model (GDSII) file, you can create a mapping of the GDSII layer numbers to layer names in the design stackup.

To create and use the mapping file:

1. Use a text editor to create a text file that maps GDSII layer numbers to layer names in the stackup. The layer mapping file must have a ***.tech** suffix.

   In *.tech format:

   - / is the comment character
   - Units may be specified with a line UNITS <string> before the lines of layer information. <string> is any of the allowed desktop length units. The default unit is nm.
   - Each layer is specified by a line that contains <*import layer*> <*product layer*> <*layer color*> <*layer elevation*> <*layer thickness*>

   where:

   - <*import layer*> is the name of the DXF layer
   - <*destination layer*> is the name to map the DXF layer to
   - <*layer color*> is a string from the choices listed here
   - <*layer elevation*> is specified in nms
   - <*layer height*> is specified in nms

   For example:

```
/ ------------------------------------------------------------
/ import#  destination     Color    Elevation  Thickness
/ ------------------------------------------------------------
S12         signal12         blue      1100       530
TR          trace            red       6620       530
S3          signal3          yellow    8150       2000
```

2. Save the *.tech file locally.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. During import, click **Open** in the **Layer mapping** panel to locate and open the existing layer mapping file.
4. Click **OK**.

   The file is imported into the active **Modeler** window.

Tech File Colors with RGB Values

Consult the table below for color string choices and their corresponding RGB values.

> **Note:**
>
> - Colors listed as two words can also be entered in Pascal case (e.g., GhostWhite is a valid substitute for ghost white).
> - Gray can be spelled using either the American or British spelling (Grey).

Color String(s)	RGB
snow1, snow	{ 255, 250, 250 }
snow2	{ 238, 233, 233 }
snow3	{ 205, 201, 201 }
snow4	{ 139, 137, 137 }
ghost white	{ 248, 248, 255 }
white smoke	{ 245, 245, 245 }
gainsboro	{ 220, 220, 220 }
floral white	{ 255, 250, 240 }
old lace	{ 253, 245, 230 }
linen	{ 250, 240, 230 }
antique white	{ 250, 235, 215 }
AntiqueWhite1	{ 255, 239, 219 }
AntiqueWhite2	{ 238, 223, 204 }
AntiqueWhite3	{ 205, 192, 176 }
AntiqueWhite4	{ 139, 131, 120 }
papaya whip	{ 255, 239, 213 }
blanched almond	{ 255, 235, 205 }
bisque1, bisque	{ 255, 228, 196 }
bisque2	{ 238, 213, 183 }
bisque3	{ 205, 183, 158 }
bisque4	{ 139, 125, 107 }
PeachPuff1, peach puff	{ 255, 218, 185 }
PeachPuff2	{ 238, 203, 173 }

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
PeachPuff3	{ 205, 175, 149 }
PeachPuff4	{ 139, 119, 101 }
NavajoWhite1, navajo white	{ 255, 222, 173 }
NavajoWhite2	{ 238, 207, 161 }
NavajoWhite3	{ 205, 179, 139 }
NavajoWhite4	{ 139, 121, 94 }
moccasin	{ 255, 228, 181 }
cornsilk1, cornsilk	{ 255, 248, 220 }
cornsilk2	{ 238, 232, 205 }
cornsilk3	{ 205, 200, 177 }
cornsilk4	{ 139, 136, 120 }
ivory1, ivory	{ 255, 255, 240 }
ivory2	{ 238, 238, 224 }
ivory3	{ 205, 205, 193 }
ivory4	{ 139, 139, 131 }
LemonChiffon1, lemon chiffon	{ 255, 250, 205 }
LemonChiffon2	{ 238, 233, 191 }
LemonChiffon3	{ 205, 201, 165 }
LemonChiffon4	{ 139, 137, 112 }
seashell1, seashell	{ 255, 245, 238 }
seashell2	{ 238, 229, 222 }
seashell3	{ 205, 197, 191 }
seashell4	{ 139, 134, 130 }
honeydew	{ 240, 255, 240 }
honeydew2	{ 224, 238, 224 }
honeydew3	{ 193, 205, 193 }
honeydew4	{ 131, 139, 131 }
mint cream	{ 245, 255, 250 }
azure1, azure	{ 240, 255, 255 }
azure2	{ 224, 238, 238 }
azure3	{ 193, 205, 205 }
azure4	{ 131, 139, 139 }
alice blue	{ 240, 248, 255 }
lavender	{ 230, 230, 250 }
LavenderBlush1, lavender blush	{ 255, 240, 245 }
LavenderBlush2	{ 238, 224, 229 }
LavenderBlush3	{ 205, 193, 197 }

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
LavenderBlush4	{ 139, 131, 134 }
MistyRose1, misty rose	{ 255, 228, 225 }
MistyRose2	{ 238, 213, 210 }
MistyRose3	{ 205, 183, 181 }
MistyRose4	{ 139, 125, 123 }
DarkSlateGray1	{ 151, 255, 255 }
DarkSlateGray2	{ 141, 238, 238 }
DarkSlateGray3	{ 121, 205, 205 }
DarkSlateGray4	{ 82, 139, 139 }
dark slate gray	{ 47, 79, 79 }
SlateGray1	{ 198, 226, 255 }
SlateGray2	{ 185, 211, 238 }
SlateGray3	{ 159, 182, 205 }
SlateGray4	{ 108, 123, 139 }
slate gray	{ 112, 128, 144 }
light slate gray	{ 119, 136, 153 }
midnight blue	{ 25, 25, 112 }
navy, navy blue	{ 0, 0, 128 }
cornflower blue	{ 100, 149, 237 }
slate blue	{ 106, 90, 205 }
medium slate blue	{ 123, 104, 238 }
light slate blue	{ 132, 112, 255 }
SlateBlue1	{ 131, 111, 255 }
SlateBlue2	{ 122, 103, 238 }
SlateBlue3	{ 105, 89, 205 }
SlateBlue4	{ 71, 60, 139 }
dark slate blue	{ 72, 61, 139 }
medium blue	{ 0, 0, 205 }
royal blue	{ 65, 105, 225 }
RoyalBlue1	{ 72, 118, 255 }
RoyalBlue2	{ 67, 110, 238 }
RoyalBlue3	{ 58, 95, 205 }
RoyalBlue4	{ 39, 64, 139 }
blue1, blue	{ 0, 0, 255 }
blue2	{ 0, 0, 238 }
blue3	{ 0, 0, 205 }
blue4, dark blue	{ 0, 0, 139 }

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
DodgerBlue1, dodger blue	{ 30, 144, 255 }
DodgerBlue2	{ 28, 134, 238 }
DodgerBlue3	{ 24, 116, 205 }
DodgerBlue4	{ 16, 78, 139 }
DeepSkyBlue1, deep sky blue	{ 0, 191, 255 }
DeepSkyBlue2	{ 0, 178, 238 }
DeepSkyBlue3	{ 0, 154, 205 }
DeepSkyBlue4	{ 0, 104, 139 }
sky blue	{ 135, 206, 235 }
SkyBlue1	{ 135, 206, 255 }
SkyBlue2	{ 126, 192, 238 }
SkyBlue3	{ 108, 166, 205 }
SkyBlue4	{ 74, 112, 139 }
light sky blue	{ 135, 206, 250 }
LightSkyBlue1	{ 176, 226, 255 }
LightSkyBlue2	{ 164, 211, 238 }
LightSkyBlue3	{ 141, 182, 205 }
LightSkyBlue4	{ 96, 123, 139 }
steel blue	{ 70, 130, 180 }
SteelBlue1	{ 99, 184, 255 }
SteelBlue2	{ 92, 172, 238 }
SteelBlue3	{ 79, 148, 205 }
SteelBlue4	{ 54, 100, 139 }
light steel blue	{ 176, 196, 222 }
LightSteelBlue1	{ 202, 225, 255 }
LightSteelBlue2	{ 188, 210, 238 }
LightSteelBlue3	{ 162, 181, 205 }
LightSteelBlue4	{ 110, 123, 139 }
light blue	{ 173, 216, 230 }
LightBlue1	{ 191, 239, 255 }
LightBlue2	{ 178, 223, 238 }
LightBlue3	{ 154, 192, 205 }
LightBlue4	{ 104, 131, 139 }
powder blue	{ 176, 224, 230 }
pale turquoise	{ 175, 238, 238 }
PaleTurquoise1	{ 187, 255, 255 }
PaleTurquoise2	{ 174, 238, 238 }

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
PaleTurquoise3	{ 150, 205, 205 }
PaleTurquoise4	{ 102, 139, 139 }
dark turquoise	{ 0, 206, 209 }
medium turquoise	{ 72, 209, 204 }
turquoise	{ 64, 224, 208 }
Turquoise1	{ 0, 245, 255 }
Turquoise2	{ 0, 229, 238 }
Turquoise3	{ 0, 197, 205 }
Turquoise4	{ 0, 134, 139 }
cyan1, cyan	{ 0, 255, 255 }
cyan2	{ 0, 238, 238 }
cyan3	{ 0, 205, 205 }
cyan4, dark cyan	{ 0, 139, 139 }
LightCyan1, light cyan	{ 224, 255, 255 }
LightCyan2	{ 209, 238, 238 }
LightCyan3	{ 180, 205, 205 }
LightCyan4	{ 122, 139, 139 }
cadet blue	{ 95, 158, 160 }
CadetBlue1	{ 152, 245, 255 }
CadetBlue2	{ 142, 229, 238 }
CadetBlue3	{ 122, 197, 205 }
CadetBlue4	{ 83, 134, 139 }
medium aquamarine	{ 102, 205, 170 }
aquamarine1, aquamarine	{ 127, 255, 212 }
aquamarine2	{ 118, 238, 198 }
aquamarine3	{ 102, 205, 170 }
aquamarine4	{ 69, 139, 116 }
dark green	{ 0, 100, 0 }
dark olive green	{ 85, 107, 47 }
DarkOliveGreen1	{ 202, 255, 112 }
DarkOliveGreen2	{ 188, 238, 104 }
DarkOliveGreen3	{ 162, 205, 90 }
DarkOliveGreen4	{ 110, 139, 61 }
dark sea green	{ 143, 188, 143 }
DarkSeaGreen1	{ 193, 255, 193 }
DarkSeaGreen2	{ 180, 238, 180 }
DarkSeaGreen3	{ 155, 205, 155 }

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
DarkSeaGreen4	{ 105, 139, 105 }
SeaGreen1	{ 84, 255, 159 }
SeaGreen2	{ 78, 238, 148 }
SeaGreen3	{ 67, 205, 128 }
SeaGreen4, sea green	{ 46, 139, 87 }
medium sea green	{ 60, 179, 113 }
light sea green	{ 32, 178, 170 }
pale green	{ 152, 251, 152 }
PaleGreen1	{ 154, 255, 154 }
PaleGreen2	{ 144, 238, 144 }
PaleGreen3	{ 124, 205, 124 }
PaleGreen4	{ 84, 139, 84 }
SpringGreen1, spring green	{ 0, 255, 127 }
SpringGreen2	{ 0, 238, 118 }
SpringGreen3	{ 0, 205, 102 }
SpringGreen4	{ 0, 139, 69 }
medium spring green	{ 0, 250, 154 }
lawn green	{ 124, 252, 0 }
light green	{ 144, 238, 144 }
green1, green	{ 0, 255, 0 }
green2	{ 0, 238, 0 }
green3	{ 0, 205, 0 }
green4	{ 0, 139, 0 }
chartreuse1, chartreuse	{ 127, 255, 0 }
chartreuse2	{ 118, 238, 0 }
chartreuse3	{ 102, 205, 0 }
chartreuse4	{ 69, 139, 0 }
green yellow	{ 173, 255, 47 }
lime green	{ 50, 205, 50 }
yellow green	{ 154, 205, 50 }
forest green	{ 34, 139, 34 }
olive drab	{ 107, 142, 35 }
OliveDrab1	{ 192, 255, 62 }
OliveDrab2	{ 179, 238, 58 }
OliveDrab3	{ 154, 205, 50 }
OliveDrab4	{ 105, 139, 34 }
dark khaki	{ 189, 183, 107 }

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
khaki	{ 240, 230, 140 }
khaki1	{ 255, 246, 143 }
khaki2	{ 238, 230, 133 }
khaki3	{ 205, 198, 115 }
khaki4	{ 139, 134, 78 }
LightYellow1, light yellow	{ 255, 255, 224 }
LightYellow2	{ 238, 238, 209 }
LightYellow3	{ 205, 205, 180 }
LightYellow4	{ 139, 139, 122 }
yellow1, yellow	{ 255, 255, 0 }
yellow2	{ 238, 238, 0 }
yellow3	{ 205, 205, 0 }
yellow4	{ 139, 139, 0 }
gold1, gold	{ 255, 215, 0 }
gold2	{ 238, 201, 0 }
gold3	{ 205, 173, 0 }
gold4	{ 139, 117, 0 }
pale goldenrod	{ 238, 232, 170 }
light goldenrod yellow	{ 250, 250, 210 }
light goldenrod	{ 238, 221, 130 }
LightGoldenrod1	{ 255, 236, 139 }
LightGoldenrod2	{ 238, 220, 130 }
LightGoldenrod3	{ 205, 190, 112 }
LightGoldenrod4	{ 139, 129, 76 }
goldenrod	{ 218, 165, 32 }
goldenrod1	{ 255, 193, 37 }
goldenrod2	{ 238, 180, 34 }
goldenrod3	{ 205, 155, 29 }
goldenrod4	{ 139, 105, 20 }
dark goldenrod	{ 184, 134, 11 }
DarkGoldenrod1	{ 255, 185, 15 }
DarkGoldenrod2	{ 238, 173, 14 }
DarkGoldenrod3	{ 205, 149, 12 }
DarkGoldenrod4	{ 139, 101, 8 }
rosy brown	{ 188, 143, 143 }
RosyBrown1	{ 255, 193, 193 }
RosyBrown2	{ 238, 180, 180 }

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
RosyBrown3	{ 205, 155, 155 }
RosyBrown4	{ 139, 105, 105 }
indian red	{ 205, 92, 92 }
IndianRed1	{ 255, 106, 106 }
IndianRed2	{ 238, 99, 99 }
IndianRed3	{ 205, 85, 85 }
IndianRed4	{ 139, 58, 58 }
saddle brown	{ 139, 69, 19 }
sienna	{ 160, 82, 45 }
sienna1	{ 255, 130, 71 }
sienna2	{ 238, 121, 66 }
sienna3	{ 205, 104, 57 }
sienna4	{ 139, 71, 38 }
peru	{ 205, 133, 63 }
burlywood	{ 222, 184, 135 }
burlywood1	{ 255, 211, 155 }
burlywood2	{ 238, 197, 145 }
burlywood3	{ 205, 170, 125 }
burlywood4	{ 139, 115, 85 }
beige	{ 245, 245, 220 }
wheat	{ 245, 222, 179 }
wheat1	{ 255, 231, 186 }
wheat2	{ 238, 216, 174 }
wheat3	{ 205, 186, 150 }
wheat4	{ 139, 126, 102 }
sandy brown	{ 244, 164, 96 }
tan	{ 210, 180, 140 }
tan1	{ 255, 165, 79 }
tan2	{ 238, 154, 73 }
tan3	{ 205, 133, 63 }
tan4	{ 139, 90, 43 }
chocolate	{ 210, 105, 30 }
chocolate1	{ 255, 127, 36 }
chocolate2	{ 238, 118, 33 }
chocolate3	{ 205, 102, 29 }
chocolate4	{ 139, 69, 19 }
firebrick	{ 178, 34, 34 }

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
firebrick1	{ 255, 48, 48 }
firebrick2	{ 238, 44, 44 }
firebrick3	{ 205, 38, 38 }
firebrick4	{ 139, 26, 26 }
brown	{ 165, 42, 42 }
brown1	{ 255, 64, 64 }
brown2	{ 238, 59, 59 }
brown3	{ 205, 51, 51 }
brown4	{ 139, 35, 35 }
dark salmon	{ 233, 150, 122 }
salmon	{ 250, 128, 114 }
salmon1	{ 255, 140, 105 }
salmon2	{ 238, 130, 98 }
salmon3	{ 205, 112, 84 }
salmon4	{ 139, 76, 57 }
LightSalmon1, light salmon	{ 255, 160, 122 }
LightSalmon2	{ 238, 149, 114 }
LightSalmon3	{ 205, 129, 98 }
LightSalmon4	{ 139, 87, 66 }
orange1, orange	{ 255, 165, 0 }
orange2	{ 238, 154, 0 }
orange3	{ 205, 133, 0 }
orange4	{ 139, 90, 0 }
dark orange	{ 255, 140, 0 }
DarkOrange1	{ 255, 127, 0 }
DarkOrange2	{ 238, 118, 0 }
DarkOrange3	{ 205, 102, 0 }
DarkOrange4	{ 139, 69, 0 }
coral	{ 255, 127, 80 }
coral1	{ 255, 114, 86 }
coral2	{ 238, 106, 80 }
coral3	{ 205, 91, 69 }
coral4	{ 139, 62, 47 }
light coral	{ 240, 128, 128 }
tomato1, tomato	{ 255, 99, 71 }
tomato2	{ 238, 92, 66 }
tomato3	{ 205, 79, 57 }

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
tomato4	{ 139, 54, 38 }
OrangeRed1, orange red	{ 255, 69, 0 }
OrangeRed2	{ 238, 64, 0 }
OrangeRed3	{ 205, 55, 0 }
OrangeRed4	{ 139, 37, 0 }
red1, red	{ 255, 0, 0 }
red2	{ 238, 0, 0 }
red3	{ 205, 0, 0 }
red4, dark red	{ 139, 0, 0 }
hot pink	{ 255, 105, 180 }
HotPink1	{ 255, 110, 180 }
HotPink2	{ 238, 106, 167 }
HotPink3	{ 205, 96, 144 }
HotPink4	{ 139, 58, 98 }
DeepPink1, deep pink	{ 255, 20, 147 }
DeepPink2	{ 238, 18, 137 }
DeepPink3	{ 205, 16, 118 }
DeepPink4	{ 139, 10, 80 }
pink	{ 255, 192, 203 }
pink1	{ 255, 181, 197 }
pink2	{ 238, 169, 184 }
pink3	{ 205, 145, 158 }
pink4	{ 139, 99, 108 }
light pink	{ 255, 182, 193 }
LightPink1	{ 255, 174, 185 }
LightPink2	{ 238, 162, 173 }
LightPink3	{ 205, 140, 149 }
LightPink4	{ 139, 95, 101 }
maroon	{ 176, 48, 96 }
maroon1	{ 255, 52, 179 }
maroon2	{ 238, 48, 167 }
maroon3	{ 205, 41, 144 }
maroon4	{ 139, 28, 98 }
pale violet red	{ 219, 112, 147 }
PaleVioletRed1	{ 255, 130, 171 }
PaleVioletRed2	{ 238, 121, 159 }
PaleVioletRed3	{ 205, 104, 137 }

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
PaleVioletRed4	{ 139, 71, 93 }
medium violet red	{ 199, 21, 133 }
violet red	{ 208, 32, 144 }
VioletRed1	{ 255, 62, 150 }
VioletRed2	{ 238, 58, 140 }
VioletRed3	{ 205, 50, 120 }
VioletRed4	{ 139, 34, 82 }
magenta1, magenta	{ 255, 0, 255 }
magenta2	{ 238, 0, 238 }
magenta3	{ 205, 0, 205 }
magenta4, dark magenta	{ 139, 0, 139 }
violet	{ 238, 130, 238 }
plum	{ 221, 160, 221 }
plum1	{ 255, 187, 255 }
plum2	{ 238, 174, 238 }
plum3	{ 205, 150, 205 }
plum4	{ 139, 102, 139 }
orchid	{ 218, 112, 214 }
orchid1	{ 255, 131, 250 }
orchid2	{ 238, 122, 233 }
orchid3	{ 205, 105, 201 }
orchid4	{ 139, 71, 137 }
medium orchid	{ 186, 85, 211 }
MediumOrchid1	{ 224, 102, 255 }
MediumOrchid2	{ 209, 95, 238 }
MediumOrchid3	{ 180, 82, 205 }
MediumOrchid4	{ 122, 55, 139 }
dark orchid	{ 153, 50, 204 }
DarkOrchid1	{ 191, 62, 255 }
DarkOrchid2	{ 178, 58, 238 }
DarkOrchid3	{ 154, 50, 205 }
DarkOrchid4	{ 104, 34, 139 }
dark violet	{ 148, 0, 211 }
blue violet	{ 138, 43, 226 }
purple	{ 160, 32, 240 }
purple1	{ 155, 48, 255 }
purple2	{ 145, 44, 238 }

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
purple3	{ 125, 38, 205 }
purple4	{ 85, 26, 139 }
medium purple	{ 147, 112, 219 }
MediumPurple1	{ 171, 130, 255 }
MediumPurple2	{ 159, 121, 238 }
MediumPurple3	{ 137, 104, 205 }
MediumPurple4	{ 93, 71, 139 }
thistle	{ 216, 191, 216 }
thistle1	{ 255, 225, 255 }
thistle2	{ 238, 210, 238 }
thistle3	{ 205, 181, 205 }
thistle4	{ 139, 123, 139 }
gray0, black	{ 0, 0, 0 }
gray1	{ 3, 3, 3 }
gray2	{ 5, 5, 5 }
gray3	{ 8, 8, 8 }
gray4	{ 10, 10, 10 }
gray5	{ 13, 13, 13 }
gray6	{ 15, 15, 15 }
gray7	{ 18, 18, 18 }
gray8	{ 20, 20, 20 }
gray9	{ 23, 23, 23 }
gray10	{ 26, 26, 26 }
gray11	{ 28, 28, 28 }
gray12	{ 31, 31, 31 }
gray13	{ 33, 33, 33 }
gray14	{ 36, 36, 36 }
gray15	{ 38, 38, 38 }
gray16	{ 41, 41, 41 }
gray17	{ 43, 43, 43 }
gray18	{ 46, 46, 46 }
gray19	{ 48, 48, 48 }
gray20	{ 51, 51, 51 }
gray21	{ 54, 54, 54 }
gray22	{ 56, 56, 56 }
gray23	{ 59, 59, 59 }
gray24	{ 61, 61, 61 }

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
gray25	{ 64, 64, 64 }
gray26	{ 66, 66, 66 }
gray27	{ 69, 69, 69 }
gray28	{ 71, 71, 71 }
gray29	{ 74, 74, 74 }
gray30	{ 77, 77, 77 }
gray31	{ 79, 79, 79 }
gray32	{ 82, 82, 82 }
gray33	{ 84, 84, 84 }
gray34	{ 87, 87, 87 }
gray35	{ 89, 89, 89 }
gray36	{ 92, 92, 92 }
gray37	{ 94, 94, 94 }
gray38	{ 97, 97, 97 }
gray39	{ 99, 99, 99 }
gray40	{ 102, 102, 102 }
gray41, dim gray	{ 105, 105, 105 }
gray42	{ 107, 107, 107 }
gray43	{ 110, 110, 110 }
gray44	{ 112, 112, 112 }
gray45	{ 115, 115, 115 }
gray46	{ 117, 117, 117 }
gray47	{ 120, 120, 120 }
gray48	{ 122, 122, 122 }
gray49	{ 125, 125, 125 }
gray50	{ 127, 127, 127 }
gray51	{ 130, 130, 130 }
gray52	{ 133, 133, 133 }
gray53	{ 135, 135, 135 }
gray54	{ 138, 138, 138 }
gray55	{ 140, 140, 140 }
gray56	{ 143, 143, 143 }
gray57	{ 145, 145, 145 }
gray58	{ 148, 148, 148 }
gray59	{ 150, 150, 150 }
gray60	{ 153, 153, 153 }
gray61	{ 156, 156, 156 }

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
gray62	{ 158, 158, 158 }
gray63	{ 161, 161, 161 }
gray64	{ 163, 163, 163 }
gray65	{ 166, 166, 166 }
gray66	{ 168, 168, 168 }
dark gray	{ 169, 169, 169 }
gray67	{ 171, 171, 171 }
gray68	{ 173, 173, 173 }
gray69	{ 176, 176, 176 }
gray70	{ 179, 179, 179 }
gray71	{ 181, 181, 181 }
gray72	{ 184, 184, 184 }
gray73	{ 186, 186, 186 }
gray74	{ 189, 189, 189 }
gray	{ 190, 190, 190 }
gray75	{ 191, 191, 191 }
gray76	{ 194, 194, 194 }
gray77	{ 196, 196, 196 }
gray78	{ 199, 199, 199 }
gray79	{ 201, 201, 201 }
gray80	{ 204, 204, 204 }
gray81	{ 207, 207, 207 }
gray82	{ 209, 209, 209 }
light gray	{ 211, 211, 211 }
gray83	{ 212, 212, 212 }
gray84	{ 214, 214, 214 }
gray85	{ 217, 217, 217 }
gray86	{ 219, 219, 219 }
gray87	{ 222, 222, 222 }
gray88	{ 224, 224, 224 }
gray89	{ 227, 227, 227 }
gray90	{ 229, 229, 229 }
gray91	{ 232, 232, 232 }
gray92	{ 235, 235, 235 }
gray93	{ 237, 237, 237 }
gray94	{ 240, 240, 240 }
gray95	{ 242, 242, 242 }

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
gray96	{ 245, 245, 245 }
gray97	{ 247, 247, 247 }
gray98	{ 250, 250, 250 }
gray99	{ 252, 252, 252 }
gray100, white	{ 255, 255, 255 }

Importing 3D Model Files

3D model files can be imported directly into the active **Modeler** window using the **Modeler** > **Import** menu. Supported 3D model file formats are listed in the table that follows.

Type	Extension(s)	Supported Version(s)
ACIS SAB	*.sab	Up to 2024
ACIS SAT	*.sat	Up to 2024
Ansoft Geometry	*.AnstGeom	All
Ansys 3D Modeler	*.sm3	Up to 2022R2 See UDMs and CAD integration with Workbench.
AutoCAD	*.dxf, *.dwg	**Linux**: Versions 2.5 (AC1002) through 2012 (AC1024) **Windows**: Versions 2.5 (AC1002) through 2017 (AC1027) See Importing DXF and DWG Format Files.
Autodesk Inventor	*.ipt, *.iam	Up to 2025
CATIA	*.exp, *.model, *.CATPart, *.CATproduct	CATIA V4 – Up to 4.2.5 CATIA V5 – Up to V5_6R2024
Creo Parametric	*.prt, *.asm	Pro/Engineer 19.0 to Creo 11.0
GL Transmission Format	*.gltf (ASCII), *.glb (binary)	Up to version 2.0
IGES	*.iges, *.igs	5.1, 5.2, 5.3
JT	*.jt	Up to v10.9
NASTRAN	*.nas	CROD, CBEAM, CTRIA3, and CQUAD4 element types
Siemens NX	*.prt	UG11 to UG18, UG NX, NX5 to NX12, NX1847 to NX2412

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type	Extension(s)	Supported Version(s)
Parasolid	*.x_t, *.x_b	9.0 through 36.1.227
SOLIDWORKS	*.SLDPRT, *.SLDASM	From 97 up to 2025
STEP	*.step, *.stp	AP 203 E1/E2, AP 214, AP 242 E1
STL	*.stl	2.0 (see Lightweight Geometry in SBR+)

Note:

- Object, material, and parameter names with non-ASCII characters are not allowed for data transfer. Such transfers fail and produce an error message.
- If you import a file into an active **Modeler** window that contains an existing model, the file is added to the existing model; it will not replace it.

To import a 3D model file:

1. Click **Modeler** > **Import**.

   The **Import File** dialog box appears.

2. Select the file type you want from the **Files of type** drop-down menu.
3. Set any import options available for the selected file type. Options may include:
   - **Validation and Healing Options** – see Healing an Imported Object.
   - **Import Material Names** – use the check box to import material names.
   - **Import Free Surfaces** – for Creo Parametric files, use the check box to import surfaces as well as parts.
   - **Stitch Tolerance and Units** – for CATIA, STEP and IGES files, specify stitch tolerance and units. The default value (**Auto**) comes from the Healing options.
   - **STL Import Options** – for STL files for Modal or Terminal solutions, select the modeling units to which the imported model is scaled and whether to merge faces that are on the same plane, and whether to Heal on import. Heal attempts to remove common issues while importing stl files, including closing solids when they are open due to minor cracks in triangulation and removing self-intersections.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The generic format specification for STL does not include units. When "**Auto** is chosen for the file units, the current Model units are normally used.

Checking **Reduce** enables fields for specifying Reduce Max Error and Reduce Percentage.

- **Create Groups for Sub Assembly** – for formats that allow importing MCAD Assemblies as groups, use the check box to retain the assembly structure of objects using groups. For every subassembly in the model, a group is created and it retains hierarchical information by creating group hierarchy. See: Group Commands for Modeler Objects.

4. Use the file browser to find and select a file for import.
5. Click **Open**.

   The file is imported into the active **Modeler** window

> **Note:**
>
> While objects created in Maxwell can always be classed in the history tree as either a solid, sheet, or wire, some imported objects may have mixture of these. Maxwell places such objects in an **Unclassified** folder in the history tree.

Importing DXF and DWG Format Files

You can import AutoCAD versions 2.5 (AC1002) through 2012 (AC1024) on Linux and versions 2.5 (AC1002) through 2017 (AC1027) on Windows. The entities are imported as 2D, not 3D. The types of entities imported are:

- 2D Polyline
- Polyline
- Line
- Arc
- Circle
- Ellipse
- Solid
- Block

To import a ***.dxf** or ***.dwg** model file (which may use an associated *.tech file):

1. Click **Modeler** > **Import**.

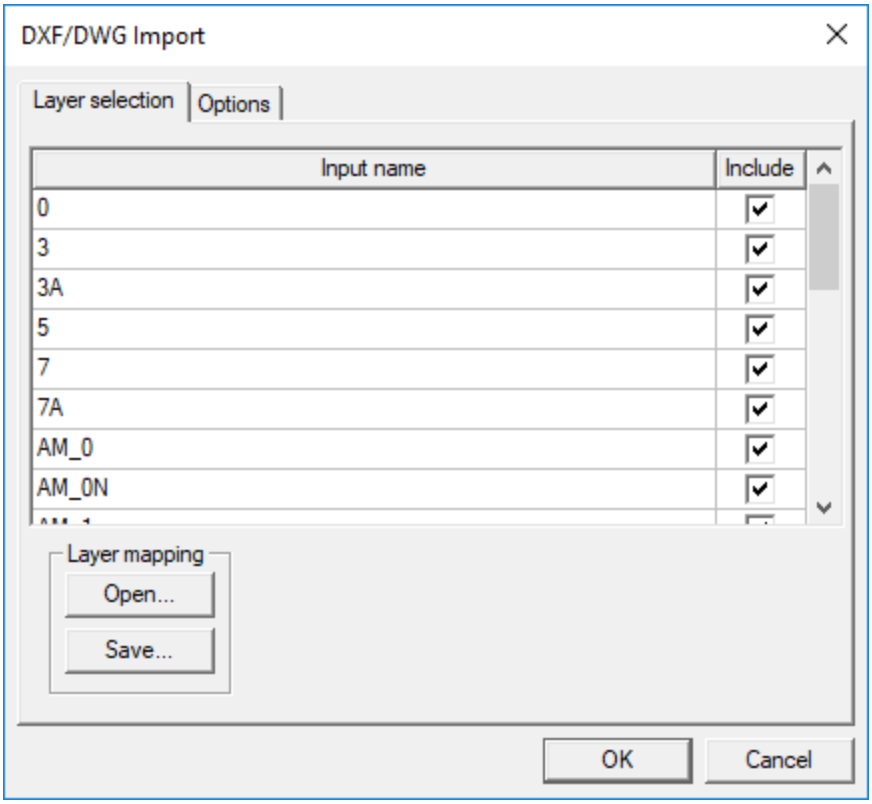
   The **Import File** dialog box appears.

2. Use the file browser to find and select an AutoCAD file for import.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **Open**.

   The **DXF/DWG Import** dialog box opens with the **Layer Selection** tab selected.



   The **Input Name** field shows the name of the layer in the DXF/DWG file. This field is not editable.

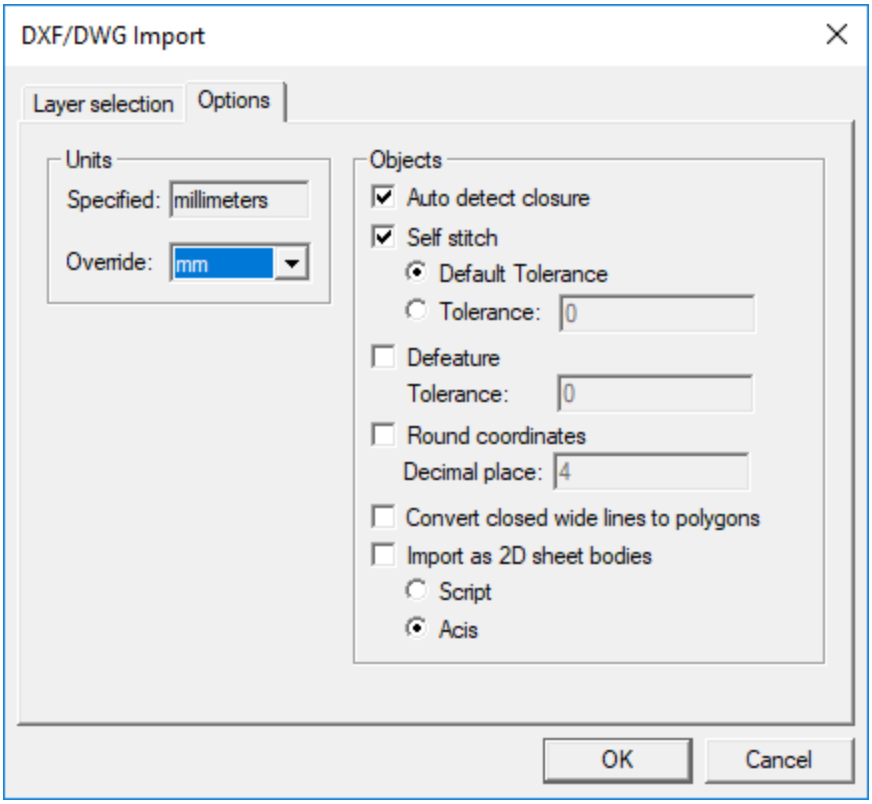
4. Use the **Include** check boxes to specify which layers to import from the selected file.
5. If there is an associated *.tech file, click **Open** to find and select it. A *.tech file is a plain text file that includes units, layer names, color, elevation, and thickness information. For example:

```
units um

//Layer_Name Color Elevation Thickness

BOTTOMLAYER purple 0 200

MIDLAYER green 500 200

TOPLAYER blue 1000 200
```

6. Click **Options**.

   The **Options** tab displays.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



7. In the **Units** area, the **Specified** field displays the layout units in the file for import. Use the **Override** drop-down menu to select a different unit of measure.
8. In the **Objects** area, use the check boxes to fine tune the import:
    - **Auto-detect closure** causes polylines to be checked to see whether or not they are closed. If a polyline is closed, the modeler creates a polygon in the design.
    - **Self-stitch** joins multiple straight line segments to form polylines. If the resulting polyline is closed, a polygon is created.
        - **Default Tolerance** uses default values to whether two coordinates should be considered for joining.
        - **Tolerance** allows you to specify a specific value for tolerance. This is useful if particular features in a model are outside of a normal tolerance allowance.
    - **De-feature Tolerance** removes certain small features in the imported geometry to reduce complexity. The features that are removed include: multiple points placed within the specified distance; thin or narrow regions ("thins" and "spikes"); and extraneous points along straight line segments.
    - **Round Coordinates Decimal Place** rounds all imported data to the specified number of decimal points.
    - **Convert closed wide lines to polygons** imports wide polylines as polygons. This affords more flexibility in changing the shape of such an object.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Import as 2D sheet bodies** causes imported objects to be organized in terms of 2D sheets.

9. Select either **Script** or **Acis** for the import method.
10. Click **OK**.

    The file is imported into the active **Layout** window.

Importing Data Tables

You can import data table files that contain data in the following formats:

- Tab-separated – Maxwell will recognize complex data if the values are separated by a tab (e.g., *real, imaginary*).
- Comma-separated – Maxwell will recognize complex data if the values are separated by a space (e.g., *real imaginary*).

1. Click **Maxwell** > **Results** > **Import Solutions**.
    - Alternatively, right-click **Results** in the project tree and then click **Import Solutions** on the shortcut menu.

    The **Imported Data** dialog box appears.

2. Click **Import Table**.

    The **Table Import** dialog box appears.

3. In the **File Name** text box, type the name of the data table file you want to import or click **Browse** and use the file browser to locate the file.
4. If the data in the table is complex, select the format — real/ imaginary, or magnitude/ phase — in which to import the data.

    If the data is simple, this option will be ignored.

5. Click **Load File**. Note that the file has not been imported yet.
6. Optionally, type a new name in the **Source Name** box that indicates the origin or the data table, or accept the default name.
7. Optionally, type a new name in the **Table Name** box that describes the data in the table, or accept the default name
8. In the **All Columns** list, the headings of each column in the data file are listed. Optionally, specify a new name for a column heading by doing the following:
    a. In the **All Columns** list, click the heading you want to change. The heading appears in the **Column Name** box.
    b. Type a new name in the **Column Name** box, and then click **Set Column Name**.

    The heading is changed to the new name in every place it appears in the **Imported Data** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. In the **Independent Data Columns** list, the first heading in the data table file is listed by default. In the **Dependent Data Columns** list, the second and subsequent headings in the data table file are listed by default. Optionally, click a heading name and then click an arrow button to move it from one column to another.
10. If the data in the **Dependent Data Columns** list contains matrix data, select **Matrix Data**. If it contains field data, select **Field Data**.
11. Click **Import**.

    You return to the **Imported Data** dialog box.

12. Click the data you want to import in the **Current Imports** list, and then click **OK**.

The solution data is now available for postprocessing.

Importing Tabular Arrays

Array data in tabular format can be imported from a file using the **Array Import** wizard that is invoked by selecting **Tools** > **Import Array from Table** and specifying a .csv file from the window that opens.

The tabular array .csv file should contain the following formatting:

```
arrIndex1 arrIndex2 arrIndex3
c1 "a0" d1
c2 "a2" d8
c3 "a3" d9
```

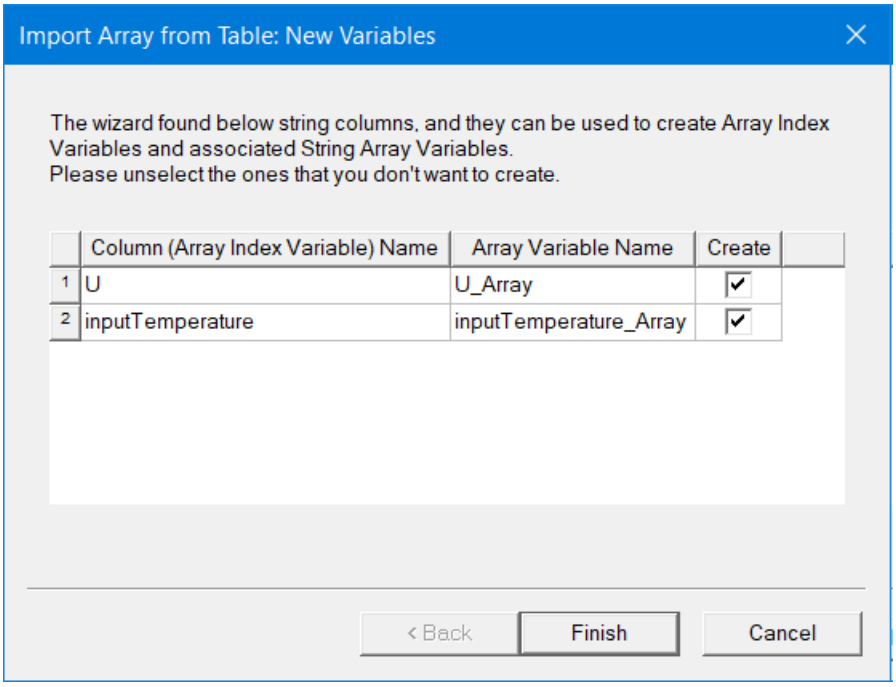
- First row is a list of Array Index Variable names separated by tabs or commas
- Remaining rows consist of a list of string values separated by tabs or commas
- Double quotation marks (“ “) for string values are optional.

After you select a .csv file to open:

- If no existing array variables are present in the design, the **Import Array From Table: New Variables** window opens.
- If one or more existing array variables are present in the design, the **Import Array From Table: Existing Variables** window opens.

The **Import Array From Table: New Variables** window is shown below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

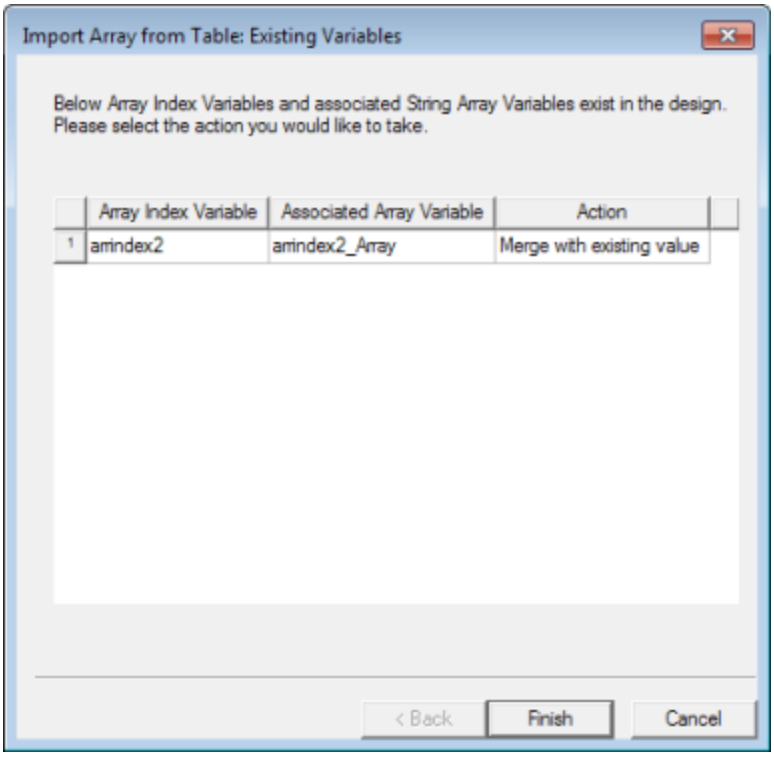


The following controls are available:

- **Column (Array Index Variable) Name** cannot be modified and is used to create the array index variable.
- The associated **Array Variable Name** has the default name "<*ColumnName*>_Array". You can modify the **Array Variable Name**, but variable uniqueness and validity is checked upon closing the wizard.
- Click the **Create** checkbox to specify that the array index variable should be created.
- Use the control buttons at bottom to go **Back** to the previous wizard window, proceed to **Finish** and close the wizard dialog box, or click **Cancel** to close the window without making changes.

If one or more existing array variables are present in the design, when you click **Next** in the above wizard dialog box, the **Import Array From Table: Existing Variables** dialog opens.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following controls are available:

- **Array Index Variable** name can *not* be modified.
- **Associated Array Variable** can be modified, but variable uniqueness and validity is checked upon closing the wizard.
- The following choices appear in the **Action** drop-down menu:
  - **Merge with existing value**
  - **Replace existing value**
  - **No Action**

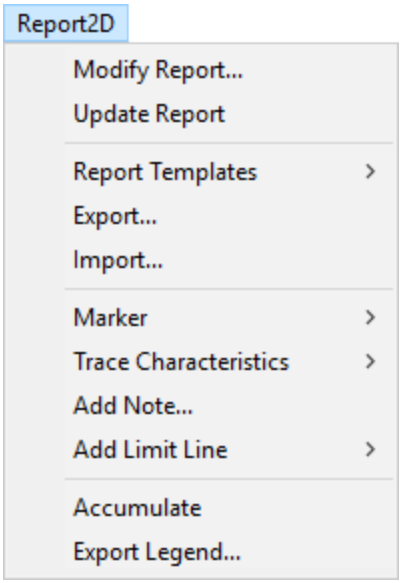
  If you select **Replace existing value**, you are then prompted for verification.
- If you import a file to a design where a string column already exists as an **ArrayIndexVariable**, the value of the variable is checked when imported. If the existing array value is a superset of new array values, the array variable will not appear in the **ExistingVariables** window.

You can also use the same .csv tabular array file to set up a parametric analysis. When the parametric analysis is set up, the **ArrayIndexVariable** is searched and its index is used to create the table. Whether the **ArrayIndexVariable** refers to a double array or a string array, you must use the array element from the imported file – rather than the index value itself – to set up the **Parametric from File** analysis.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Importing Plot Data

When a report is open in Ansys Electronics Desktop, the **Report2D** menu appears.



> **Note:**
>
> You *must* have a report open in order to import plot data.

The **Report2D** > **Import Data** command lets you import plot data from comma delimited files (*.csv), tab delimited files (*.tab), Ansoft PlotData files (*.dat), post-processor files (*.txt), or Ansoft ReportData files (*.rdat).

1. From the top menu bar, click **Report2D** > **Import**.

   The **Open** window appears.

2. Browse for and select the desired file.
3. Click **Open** to import the file into the current report.

   The imported traces appear in the Project tree under the current report.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> For a report trace where the primary sweep is NOT the same as the x-component, an export report then import report may not produce the same curve. In this circumstance, only *.rdat import the same curve/trace. For other formats, the import produces two separated traces.

Exporting Files

You can export the following types of files from Ansys Electronics Desktop projects:

- 2D model files
- 3D model files
- Graphics files
- Reports as data or graphics files

Exporting 2D Model Files

Maxwell exports 2D models in AutoCAD *.dxf format. In this format, the geometry located within the XY plane is exported.

> **Note:**
>
> If you want to export a plane that does not coincide with the global xy plane, you must create a relative coordinate system to redefine the location of the origin.

To export a file to ***.dxf** format:

1. Click **Modeler** > **Export**.

   The **Export File** window appears.
2. Browse to and select a location to save your file.
3. In the **File Name** field, name the file.
4. From the **Save as Type** drop-down menu, select **AutoCAD DXF Files (*.dxf)**.
5. Click **Save**.

   The file is exported to the specified location with the appropriate file format.

Exporting 2D AutoCAD Drawing Interchange Format Files

When you export a file in the AutoCAD Drawing Interchange Format (**.dxf**), the 2D geometry located within the xy plane on the working coordinate system is exported. You can choose to

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

export only the selected parts of the geometry or all of the parts.

> **Note:** If you want to export a plane that does not coincide with the global xy plane, you must create a relative coordinate system to redefine the location of the origin. See *Creating a Relative Coordinate System* for more information.

To export a file to a **.dxf** format:

1. Click **Modeler** > **Export**.

   The **Export File** dialog box appears.
2. Use the file browser to find the directory where you want to save the file.
3. Type the name of the file in the **File name** box.
4. Select **AutoCAD DXF Files(*.dxf)** from the **Save as type** drop-down menu.
5. Click **Save**. The file is exported to the specified location as a **.dxf** file.

Exporting 3D Model Files

You can export 3D models to 3D model file formats.

Supported 3D model file formats include the following:

Type	**Extension(s)**
ACIS SAB (exported with version 5.0)	*.sab (Exported with version 5.0)
ACIS SAT (exported with version 5.0)	*.sat (Exported with version 5.0)
AutoCAD	*.dxf
GDSII	*.gds
IGES	*.iges, *.igs (Exported as version 5.1)
OBJ Files (for importing models to Ensight.)	*.obj
Parasolid	*.x_t, *.x_b
STEP	*.step, *.stp (Exported as version AP242 Ed1)
STL	*.stl
GL Transmission Format	.gltf or .glb (exported as version 2.0)

To export a file to a 3D model format:

1. Click **Modeler** > **Export**.

   The **Export File** window appears.
2. Browse to and select a location to save your file.
3. In the **File Name** field, name the file.

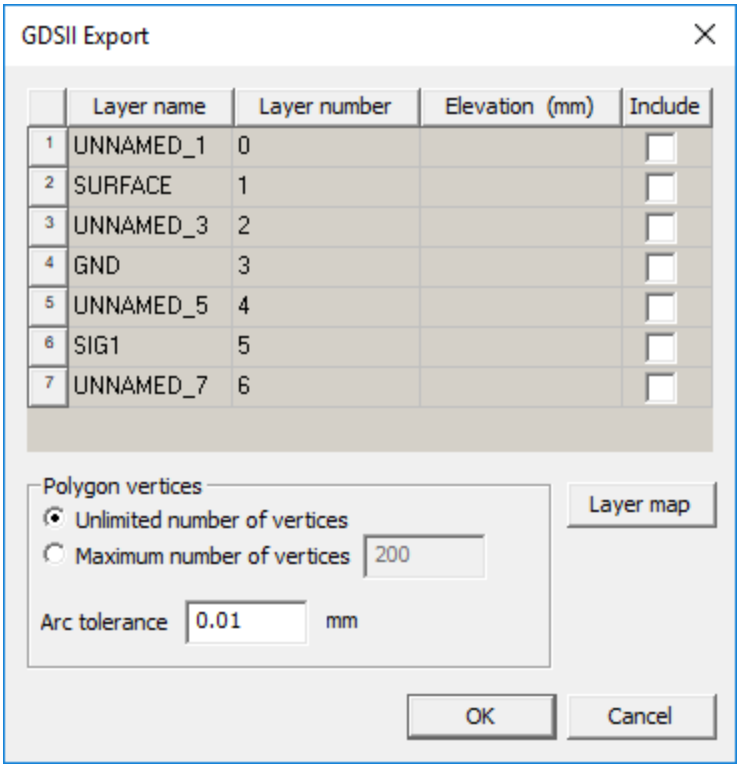
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. From the **Save as Type** drop-down menu, select the desired 3D file type.
5. Click **Save**.

   The file is exported to the specified location with the appropriate file format. Some file types launch an additional dialog box for specifying settings prior to export. These are described below.

GDSII Files

There are additional options for *.gds files:

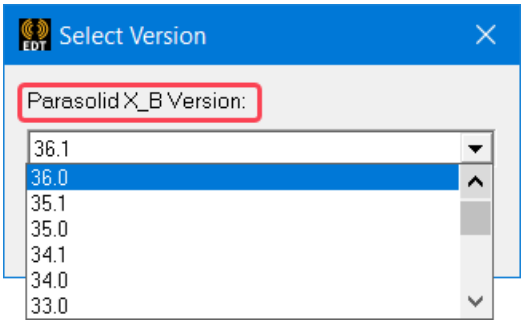


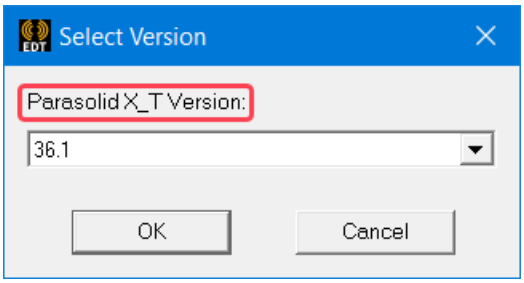
- Use the **Include** check boxes to select layers for export.
- Click **Layer map** to define a *.layermap file, if desired. This is a text file that maps GDSII layer numbers to layer names in the stackup. The *.layermap file can have the same format as the .tech file used in GDSII import, but it only needs the layer name and number. Any other information is ignored.
- In the **Polygon Vertices** area, select either **No Limit to the number of vertices** or **Limit the number of vertices** and specify a value.
- For **Arc Tolerance**, leave the default or specify a value.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parasolid Files

There is one additional option for Parasolid files:





Use the drop-down menu to select a Parasolid version.

Exporting Graphics Files

You can export to the following graphics formats:

Extension	Contents
.bmp	Bitmap files
.gif	Graphics Interchange Format files
.jpg	Joint Photographics Experts Group files
.png	Portable Network Graphics format files
.tiff	Tagged Image File Format files.
.wrl	Virtual Reality Modeling Language (VRML) files.
.gltf or **.glb**	Graphics Language Transmission Format (GLTF); only available on Windows

To export a Modeler image to a graphics format:

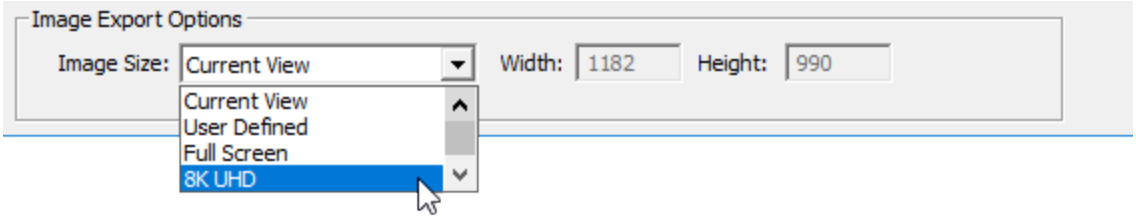
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Modeler** > **Export**.

   The **Export File** window appears.

2. Browse to and select a location to save your file.
3. In the **File Name** field, name the file.
4. From the **Save as Type** drop-down menu, select the desired image format.

   When you select an image format, **Image Export Options** appear. Note that this window does not appear for .gltf, .glb, or .wrl files, which are not image formats; if you are saving to one of these formats, skip to Step 6.



5. Choose your **Image Size**.

   Options include: Current View, User Defined (which allows you to specify **Width** and **Height**), Full Screen, 8K UHD, 4K UHD, 1080p HD, 720p HD, and 480p SD.

6. Click **Save**.

   The file is exported to the specified location with the appropriate file format.

You can also export an image file to a specified resolution using scripting commands. Fonts and line thickness are not scaled, only the image. You will have to iteratively increase font sizes until you find a suitable output.

Example Script:

```
oEditor.ExportModelImageToFile("C:/Users/Documents/highresexample_
image.jpg", 7680, 4320,

[

"NAME:SaveImageParams",

"ShowAxis:="        , "True",

"ShowGrid:="        , "True",

"ShowRuler:="       , "True",

"ShowRegion:="      , "Default",

"Selections:="      , ""
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
])
```

Exporting Options Files

Ansys Electronics Desktop can export user and host options files in XML format.

Exporting Options Files Using the Desktop UI

Export options files by selecting **Tools** > **Options** > **Export Options Files**. This brings up a browser dialog box to select the destination directory.

Click **Open** to copy all config files for the current user and current host to the specified directory. Config files for the install, install_machine, user, and user_machine levels will be copied, if they exist. One additional file, admin.XML, will also be copied to the destination directory; this file does not contain user configurable options.

Exporting Options Files Using a Script

A script command has been added that exports options config files:

ExportOptionsFiles

Syntax	`ExportOptionsFiles <DestinationDirectory>`
Return Value	None.
Parameters	BSTR <DestinationDirectory>
Python Example	`oDesktop.ExportOptionsFiles ('D:/test/export')`

Validating Projects

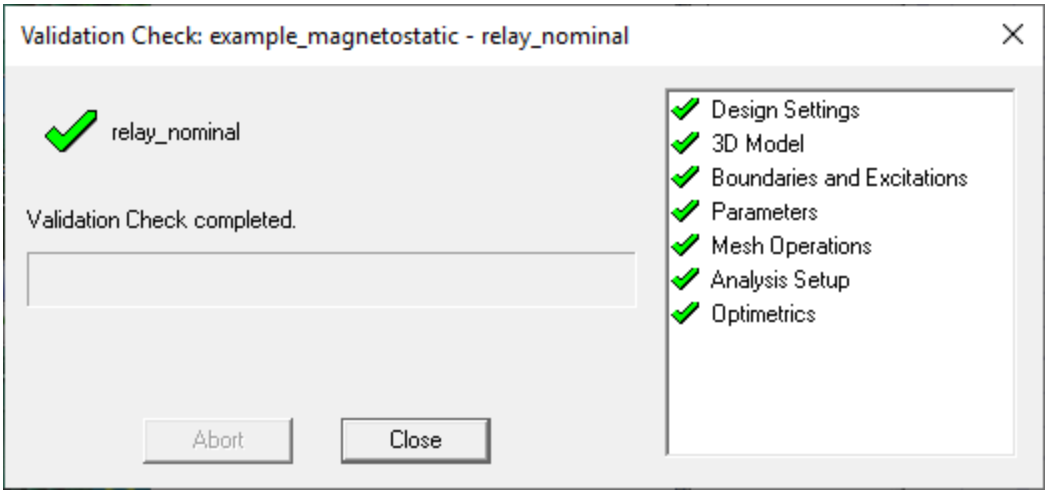
Before running an analysis on a model, it is important to first perform a validation check on the project. During a validation check, the solver examines the setup details of the active project to verify that all the necessary steps have been completed and that their parameters are reasonable.

To perform a validation check on the active project:

1. Click **Maxwell** > **Validation Check**. You can also click the **Validate** icon on the **Simulation** ribbon tab.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Electronics Desktop checks the project setup, and the **Validation Check** window appears.



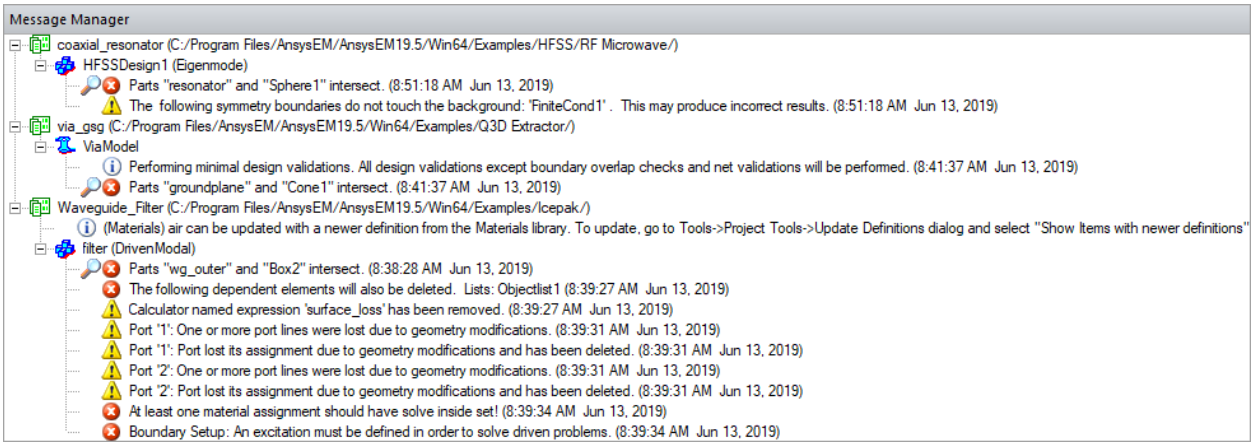
2. View the results of the validation check in the **Validation Check** window.

   The following icons can appear next to an item:

	Indicates the step is complete.
	Indicates the step is incomplete.
	Indicates the step may require your attention.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. View any messages in the **Message Manager** window.



4. If the validation check indicates that a step in your project is incomplete or incorrect, carefully review the setup details for that particular step and revise them as necessary.
5. After you have revised any setup details, click **Maxwell** > **Validation Check** to run another validation check. Repeat until the Validation Check completes with no issues.
6. Click **Close**.

Validations Tab

Model Settings

You can adjust the degree to which the software checks a model for faults that could jeopardize mesh accuracy. There are four levels of model validation that a user can specify for a given design: **Warning Only**, **Basic**, **Strict**, and **None**. Note that this setting affects only the "3D Model" stage of a design validation. The default Entity Check Level is **Strict**.

- The **Warning Only** entity check setting allows all models to pass 3D Model validation regardless of any faults that are found. These faults are posted in the message window as warnings.
- The **Basic** entity check setting allows most models to pass 3D Model validation. Some faults are flagged as model errors, thereby prohibiting a design from proceeding to the meshing stage of an analysis. You must correct such errors before attempting to analyze the design under the Basic setting, or you must change the entity check level to Warning Only.
- The **Strict** entity check setting enforces a tighter tolerance for model faults than the Warning Only and Basic settings. All model faults that are found during 3D Model validation are posted to the message window. These errors must be corrected before attempting to analyze the design under the Strict setting, or you must change the entity check level to Basic or Warning Only.
- The **None** setting disables model validation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In addition, check boxes allow you to adjust model validation to

- **Ignore Unclassified Objects**
- **Skip Intersection Checks**

These settings are unchecked by default.

Maxwell 2D and Maxwell 3D Design Settings

For both 2D and 3D designs, you can choose either:

- **Perform full validations** (the default setting) – All design validations are performed.
- **Perform minimal validations** – All design validations except boundary overlap validation are performed.

For 3D designs only, you can select **Skip mesh checks** to prevent the solver from failing validation when the mesh has a few elements of poor quality.

Adjusting Settings

To adjust settings on the **Validations** tab:

1. Select **Maxwell** > **Design Settings** to open the **Design Settings** dialog box; then select the **Validations** tab.
2. Choose the desired level of Model validation from the **Entity Check Level** drop-down menu.
3. Optionally, adjust Model validation to **Ignore Unclassified Objects**, or to **Skip Intersection Checks**.
4. Optionally, select **Perform minimal validations** if you want to omit boundary overlap validation checking.
5. Optionally, **Skip mesh checks** to prevent the solver from failing validation when the mesh has a few elements of poor quality.
6. Optionally, click the **Save as Default** button to make the current selections the default settings.
7. Click **OK** to accept the selections and close the dialog.

> **Note:** You can also view and set design settings by selecting the current design, and then in the **Properties** window, select the tab for the solver.

Copying and Pasting a Project or Design

To **copy** a project or design:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the Project Tree, select a project or design to enable the menu command **Edit** > **Copy**.
2. Click **Edit** > **Copy**.

   The project or design is copied for pasting.

To **paste** a project or design:

1. In the Project Tree, select a project or design to enable the menu command **Edit** > **Paste**.
2. Click **Edit** > **Paste**.

   The project or design is pasted under the selected project, and an icon is added to the project tree.

> **Note:**
>
> You can also use keyboard shortcuts.

Renaming Projects

In general, use the **File** menu commands to manage projects. If you move or change the names of files without using these commands, the software may not be able to find information necessary to solve the model.

To rename an existing, active project:

1. Select the project in the Project Tree.
2. Right-click and select **Rename**.

   This activates the text field for the project name.

3. Type the new project name and press **Enter**.

   The new project name appears in the directory and the project remains in the original location.

Archiving Projects

Use the **File** > **Archive** command to place a project and any other files related to the project that you want to include in a *.<product>z file or *.zip format archive. You can make notes about the contents of the archive and specify whether to include results and solutions files. The **Archive** command attempts to automatically detect the necessary files for linked projects and automatically include them in the archive. You can also add additional files to the archive, including results files, external files and projects. For example, if a project linked to the main project also has linked or associated files, you can add them.

Archive File Types

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

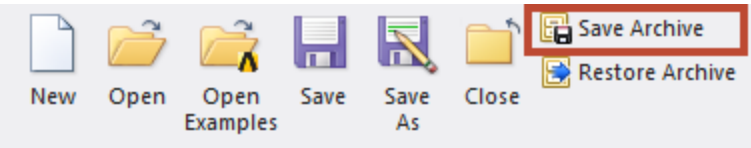
Internally, project archive files are .zip files, and are compatible with any program that can read .zip files. Project archive files have an extension that is unique for each product, and is generated by adding a 'z' to the project file extension (for example, *.aedtz, *.adsnz). This extension displays as the default when saving and restoring archive files.

Archive Preview

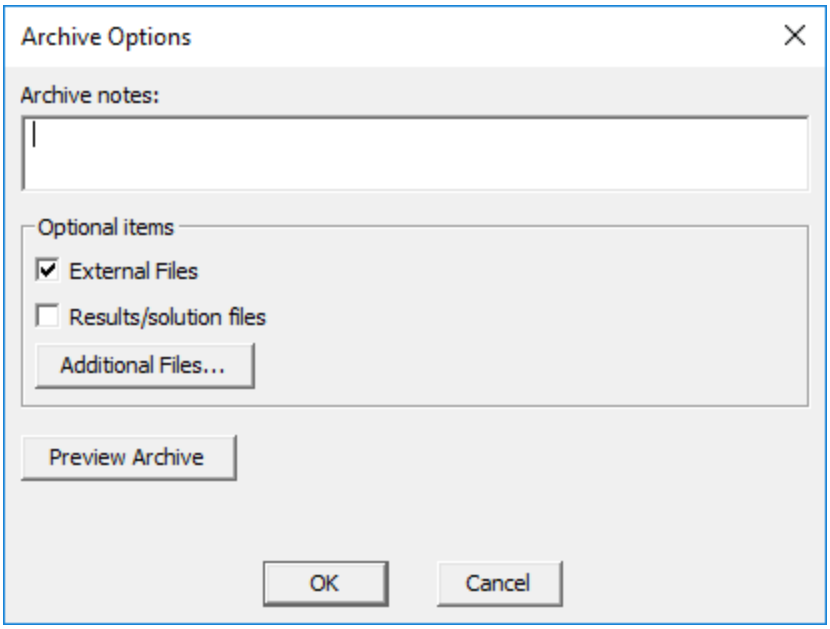
The **Archive** command includes a preview feature that lets you review the contents of a planned archive.

To archive the current project:

1. Click **File** > **Archive** or select the **Desktop** tab of the ribbon and click **Save Archive**.



The **Archive Options** dialog box opens.

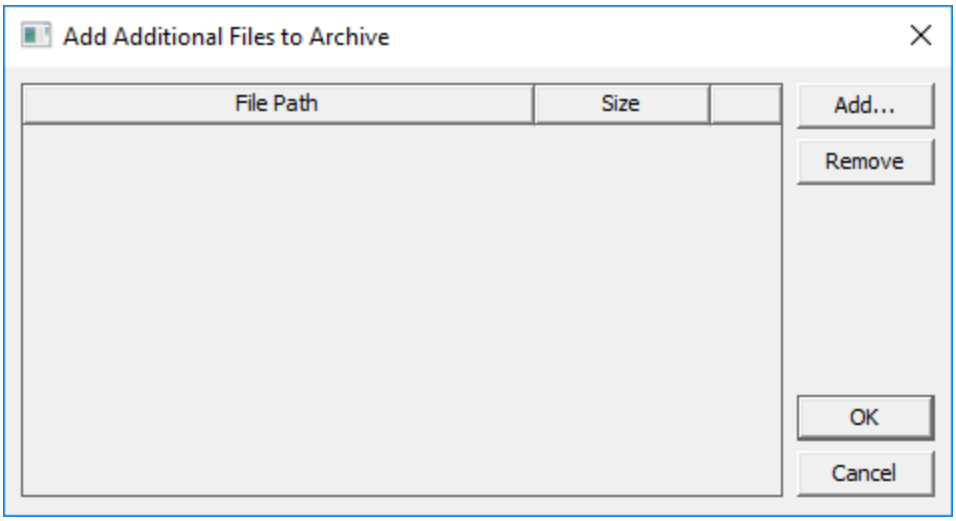


- **Archive Notes** – in this field, you can specify notes that will be visible when previewing the archive. These notes can be viewed from the preview dialog box without restoring the archive.
- **External Files** – selecting this check box causes all external files to be included in the archive. These include any existing files associated with the project, such as

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

linked files, or files added through the **Project** > **Insert Documentation File** command or **Project** > **Data Set** command.

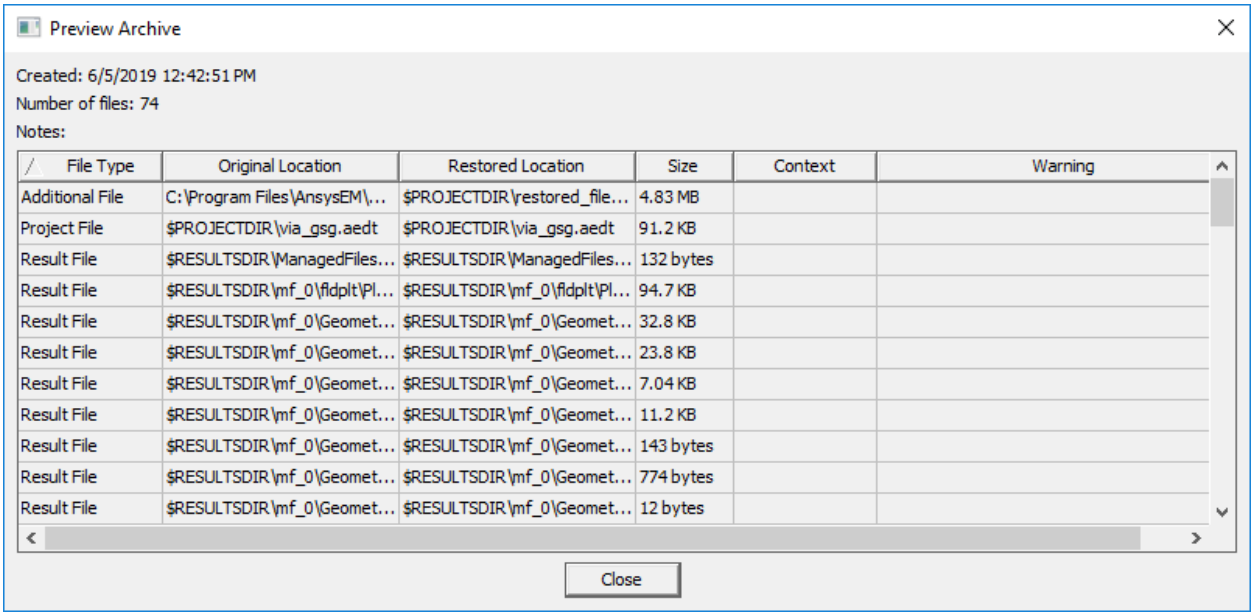
- **Results/Solution Files** – selecting this check box causes the entire results directory to be included in the archive. This may greatly increase the size of the archive file.
- **Additional Files** – clicking this opens the **Add Additional Files to Archive** dialog box.



  - Click **Add** to browse and locate additional files you want to include in the archive.
  - Select and the **Remove** any files listed.
  - Click **OK** to accept changes or **Cancel** any proposed changes.

2. Select any optional items, and make any desired notes in the text field.
3. When you have made your selections, click the **Preview Archive** button in the Archive Options dialog box to view archive contents, file types, the locations of the archive files, and the locations where restoring from archive would place them. Additionally, there may be context or warning information.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To read longer locations, you can drag the column header to expand them.

Previewing an archive before creating the archive can be helpful in order to see exactly what files will be included in an archive, as well as how those files are being relocated. Another purpose of previewing an archive is to view warnings and consider if any additional files need to be added to the archive.

4. When you are ready to create the archive, **Close** the preview and click **OK**.

   A browsing window appears.

5. Specify a name for your file and select the format you want to use from the **Save as type** drop-down menu.
6. Click **OK** to create the archive.

File Relocation

In a project to be archived, external files can be located anywhere on the user's system. One of the goals is for the restored project to be relatively self contained, and to NOT allow the restoring of an archived project to haphazardly write files anywhere on the restoring user's system.

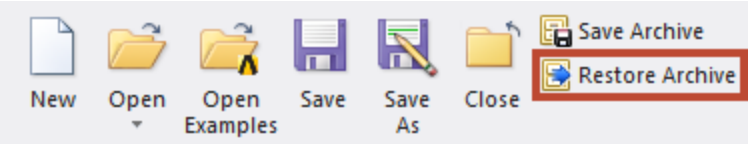
To achieve this, it is sometimes necessary to change the location of files in the archived project so that the external files are located in the project directory. At archive time, any external files not located in the project directory are relocated to the restored_files subdirectory of the project directory in the archived project. Any external files located in the user library or system library will be relocated to the personal library directory. Note that the project file that is written into the archive will be updated to refer to the files at the new locations, and the original project file will remain unaltered.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

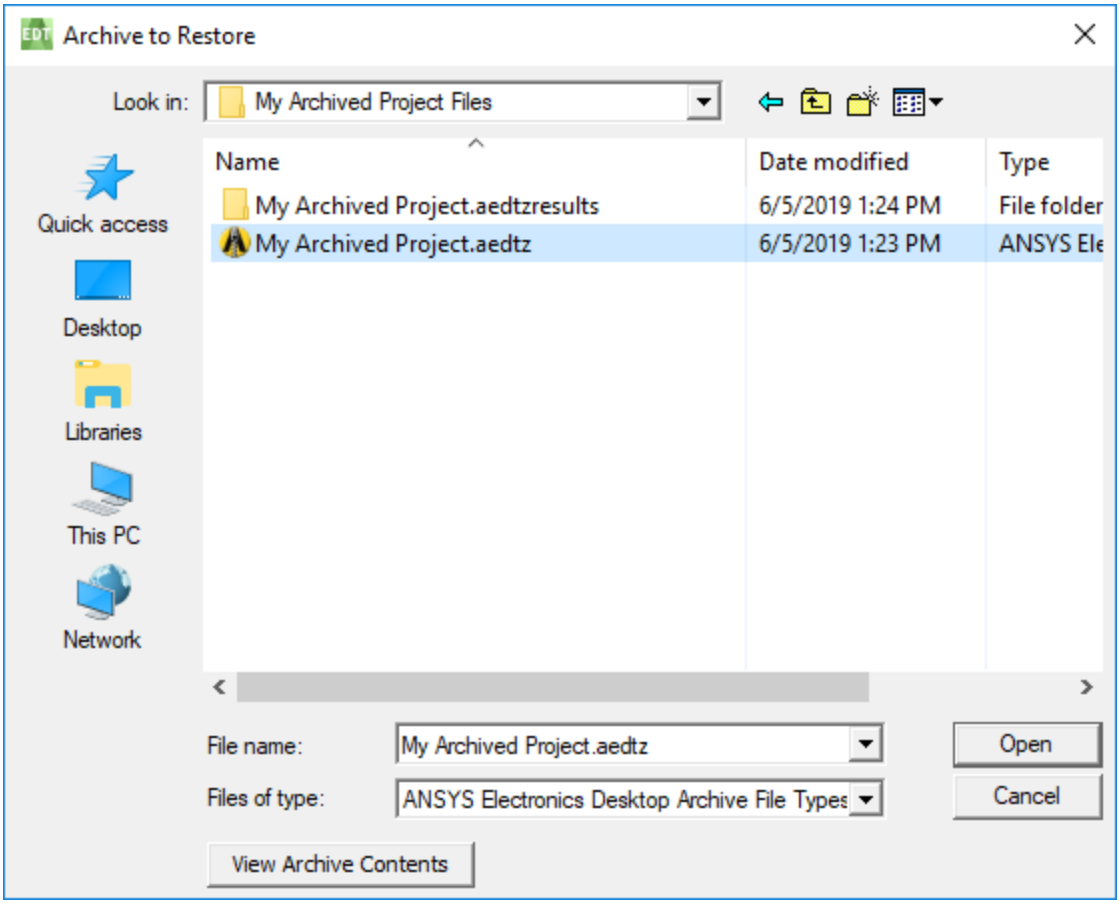
Restoring an Archived Project

To restore an existing archive created with **File** > **Archive...**, use the **File** > **Restore Archive** command.

1. Click **File** > **Restore Archive** or select the **Desktop** tab of the ribbon and click **Restore Archive**.



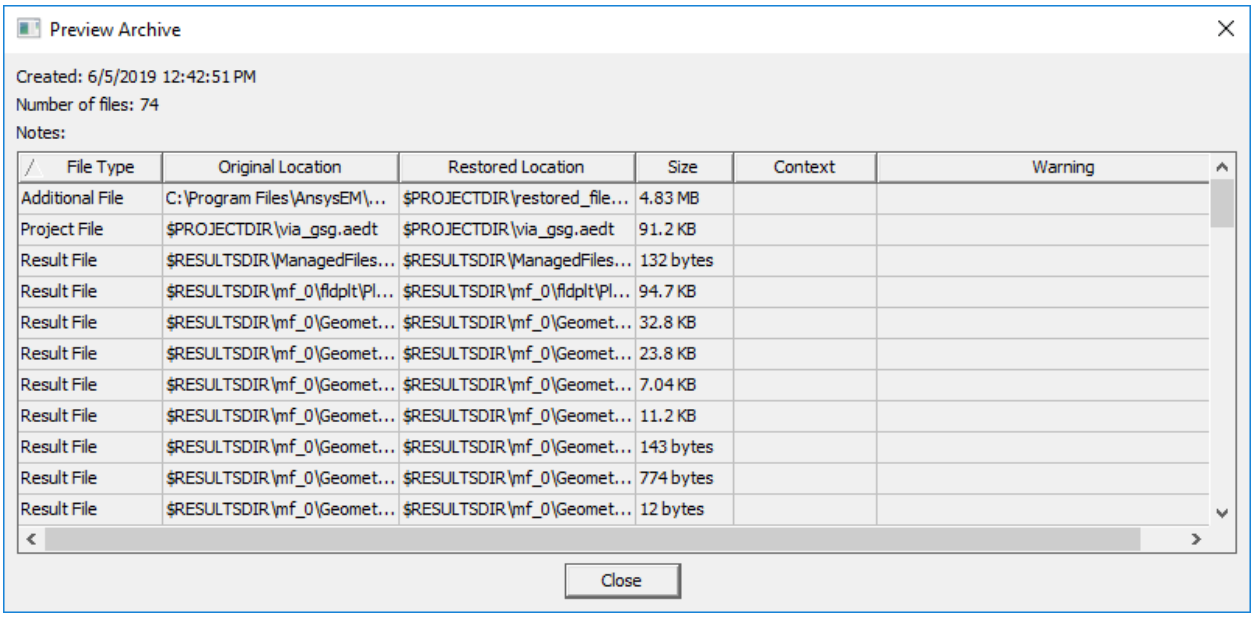
This displays an **Archive to Restore** browser window that lets you navigate your file system for *.aedtz, legacy *.*<solversuffix>z* or *.zip archive files.



2. If you select a valid archive file, you can click **View Archive Contents** to preview the contents.

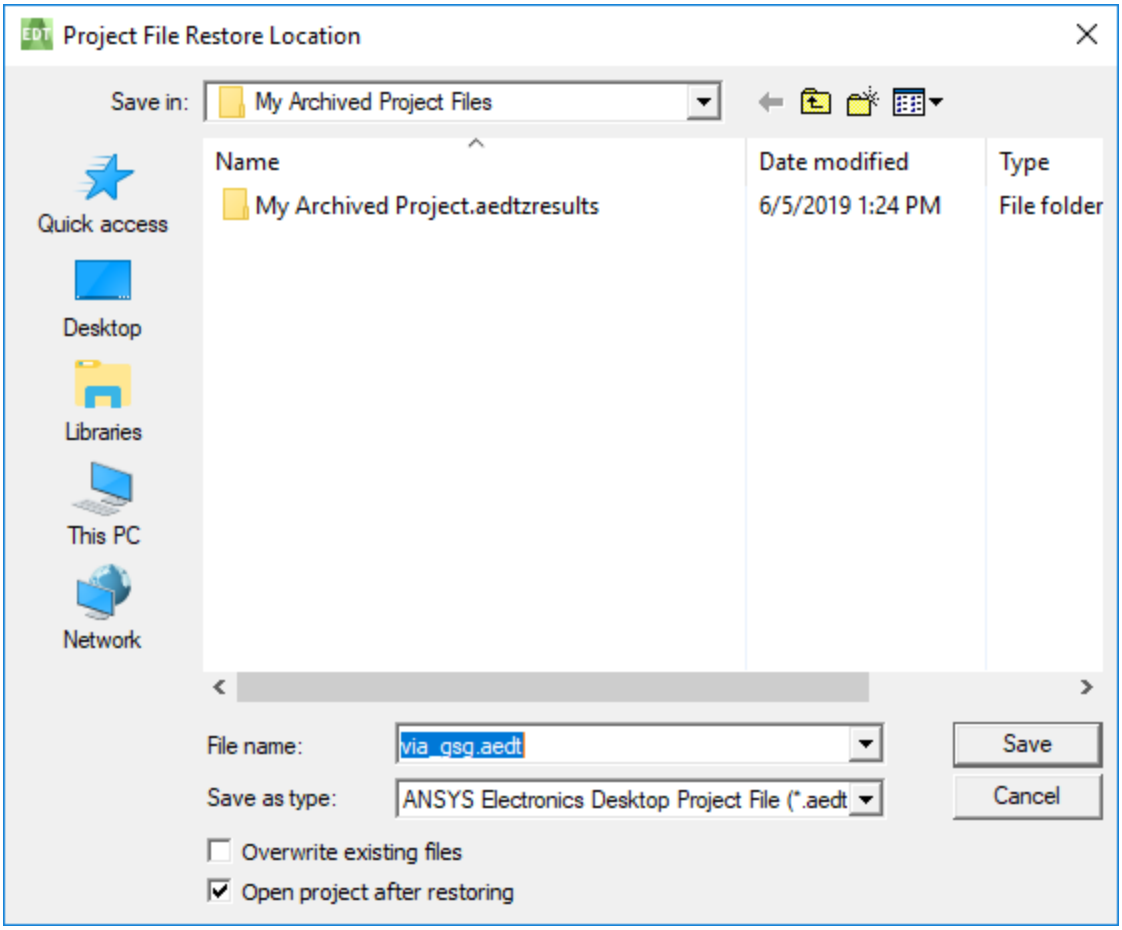
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Preview Archive** dialog box lists all files in the archive, notes, the original and restored locations, and any warnings that were generated at archive time. These warnings may be useful to identify additional steps that are needed to update any files to refer to files which had to be manually added to the archive.



3. Click **Close** to exit the preview.
4. In the browsing window, click **Open** to change to a **Project File Restore Location** browser.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



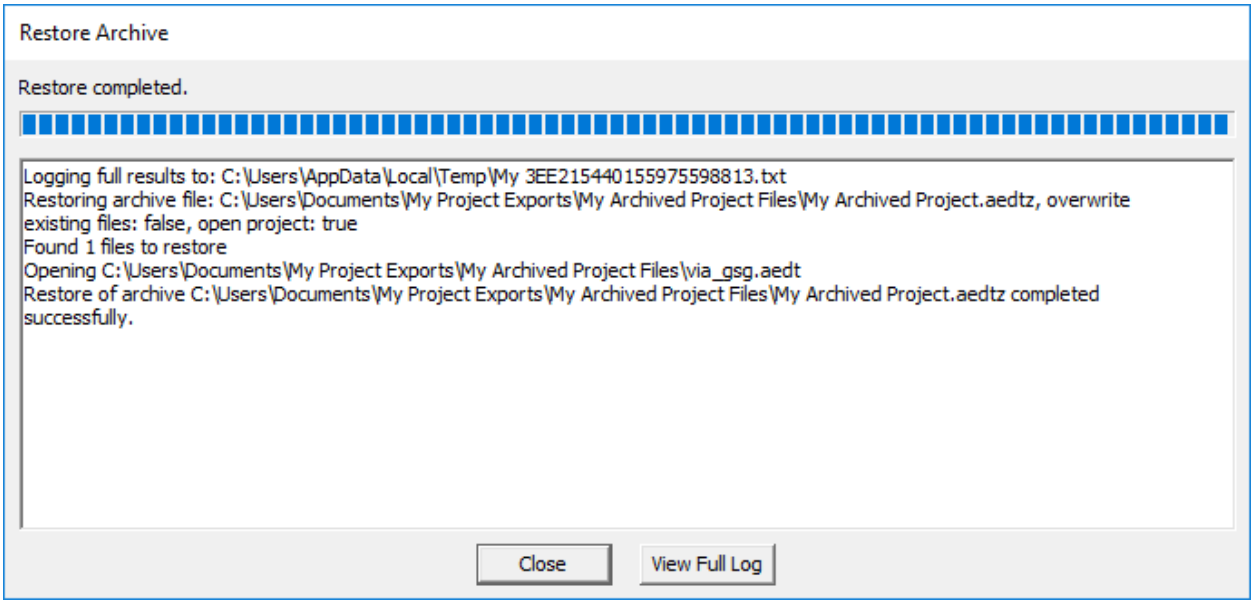
You can edit the file name, and select options:

- **Overwrite existing files** – If this check box is selected, restored files will automatically overwrite existing files during the restore process. If this button is deselected, existing files will not be modified.
- **Open project after restoring** – If this check box is selected, the project will be opened in this instance of the application after all files have been restored.

5. Click **Save** to restore the archived file.

   While restoring an archive, a dialog box displays showing restore results. A progress bar shows the relative progress, and the text window displays important information and warnings.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Electronics Desktop also generates a full log file that contains detailed information about the restore process. The first line in the text window displays the location of the full log file. After the restore has been completed, you can click **View Full Log** to open this log file, or use a text editor and open the file at the specified location.

Deleting a Project or Design

To delete a project or design:

1. In the Project Tree, select a project or design.
2. Click **Edit** > **Delete**, or press the **Delete** key.
3. Confirm the warning box to complete the delete operation.

   The project or design is removed from the Project tree.

Setting Read Only Designs

Designs can be set as either Read Only or Full Access. Full Access is the default status of a design and allows you to modify the design. In Read Only mode, you may only run the design or link it to another design. Setting a design as Read Only protects it from accidental modification.

Read Only designs are marked with a red lock in the Project Manager:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To change a single design's designation:

1. In the **Project Manager**, right-click the design.
2. Select either **Convert to Read Only** or **Convert to Full Access**.

To change the designation of *every design* in a project:

1. In the **Project Manager**, right-click the project.
2. Select either **Convert All Designs to Read Only** or **Convert All Designs to Full Access**.

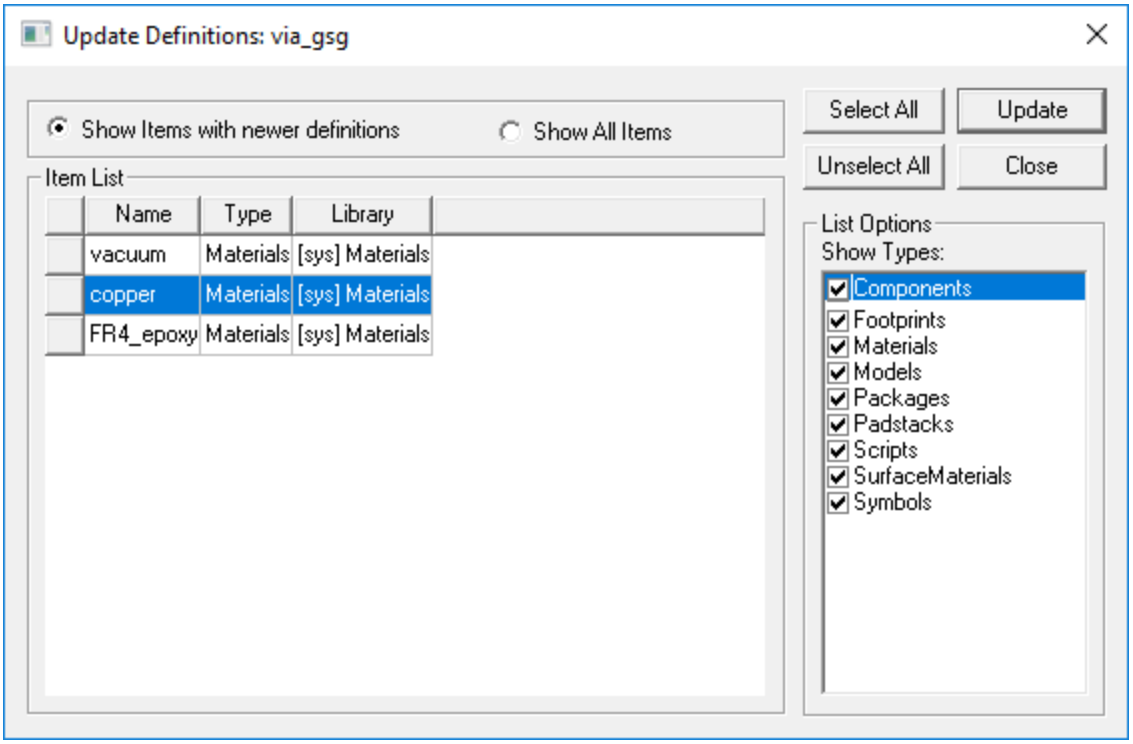
> **Note:**
>
> Read Only settings are not saved when you save the project file.

Updating Design Components

To update components defined in the current design:

1. Click **Tools** > **Project Tools** > **Update Definitions**.

   The **Update Definitions** dialog box appears.



2. Select either **Show Items with newer definitions** or **Show All Items**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. From the **Show Types** list in the **List Options** section, select the types of definitions you want to show in the **Item List**.
4. Select the item(s) you want to update from the **Item List**, or use **Select All.**
5. Click **Update**.

   A message appears telling you the update was successful.

6. Click **OK** to close the message.

When you are finished updating definitions, click **Close**.

Removing Unused Components from a Project

To remove unused components from a project:

1. Click **Tools** > **Project Tools** > **Remove Unused Definitions**.

   The **Unused Definitions** dialog box appears.

2. Select the **Delete** check box for any components you want to remove.

   You can also click the **Select All** button to select all components listed.

3. Click **Apply**.

   A warning message appears that indicates that the undo/redo history will be cleared for the project.

4. Click **OK** to continue, or click **Cancel** to cancel the deletion.
5. When you are finished removing components, click **OK** to close the **Unused Definitions** dialog box.

Undoing and Redoing Commands

The **Edit** menu contains **Undo and** Redo commands.

Use the **Undo** command to cancel the last action you performed on the active project or design.

Use the **Redo** command to reperform an action you previously undid.

Both commands are useful for dealing with unintended actions related to project management, model creation, and post-processing.

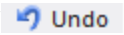
To use **Undo** or **Redo**:

1. In the **Project Manager** window, do one of the following:
   - To undo or redo your last project-level action, such as inserting a design or adding project variables, click the project icon.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To undo or redo your last design-level action, such as drawing an object or deleting a field overlay plot, click the design icon.

> **Note:**
>
> You cannot undo an analysis that you've performed on a model—that is, the **Maxwell** > **Analyze** command.

2. Click **Edit** > [**Undo/Redo**], or click the **Undo** or **Redo** button () on the **Desktop** tab.

   Your action is undone or redone.

> **Important:**
>
> When you save a project, Electronics Desktop clears the entire undo/redo history for the project and its designs.

Inserting a Documentation File

You may want to add a documentation file to the project tree.

1. Click **Project** > **Insert Documentation File**.

   This opens a file browser that lets you navigate your file system.

2. Select the documentation file and click **OK**.

   The documentation file is placed in the Project Tree.

Working with Design Notes

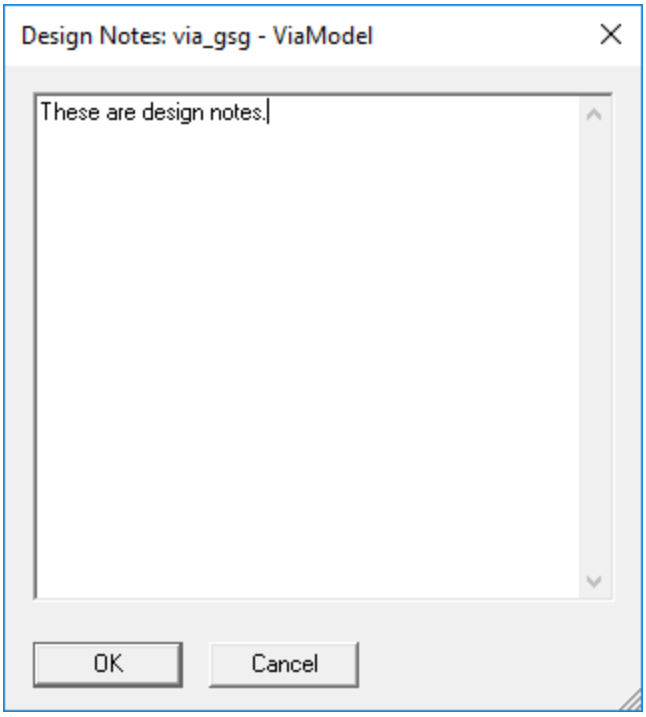
You can save design notes, such as information about the creation date or a description of the device being modeled. This is useful for keeping a running log.

To add notes:

1. Click **Maxwell 3D**, **Maxwell 2D**, or **RMxprt** > **Edit Notes**.

   The **Design Notes** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click in the window and type your notes.
3. Click **OK** to save the notes with the current project.

To edit notes, use the same window or double-click the **Notes** icon (Notes) in the Project Tree.

To delete the existing notes for a design:

1. Select the **Notes** icon in the Project Tree.
2. Click **Edit** > **Delete**, or right-click and select **Delete**.

   The Notes icon is removed from the Project Tree.

> **Note:**
>
> Notes are used to document aspects of designs only.
>
> For project-level documentation, you can insert a documentation file into a project with the **Project** > **Insert Documentation File** command.

Printing

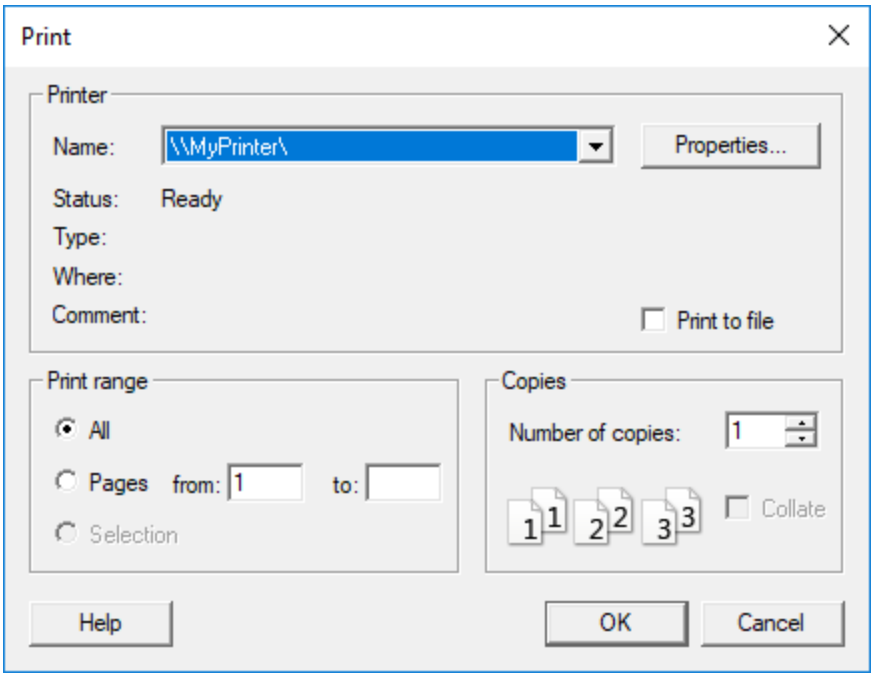
Electronics Desktop printing commands enable you to send an image of the active window to the printer.

To print the project:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **File** > **Print**.

   A dialog box similar to the following one appears:



2. Print options will vary based on your printer. Specify your desired options.
3. Click **OK** to print.

To change additional print settings, click **File** > **Page Setup**.

To preview what your printout will look like, click **File** > **Print Preview**.

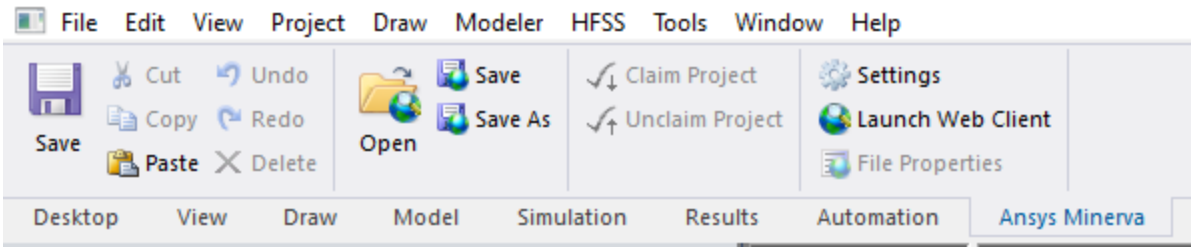
> **Important:**
>
> To print from Ansys software on Linux, you must first configure a printer from the MainWin control panel.
>
> For more information, consult documentation for your specific instance of Linux.

Minerva Remote Storage Environment

Ansys Minerva is a remote storage environment where you can store Ansys project archive files. You can collaborate with other users by downloading a project, making changes, and then uploading the project. One of the features of Ansys Minerva is the ability to store and retrieve Ansys projects from the Minerva server. This ability has been integrated into Electronics Desktop so it is now possible to download and open a project stored on Minerva directly from the

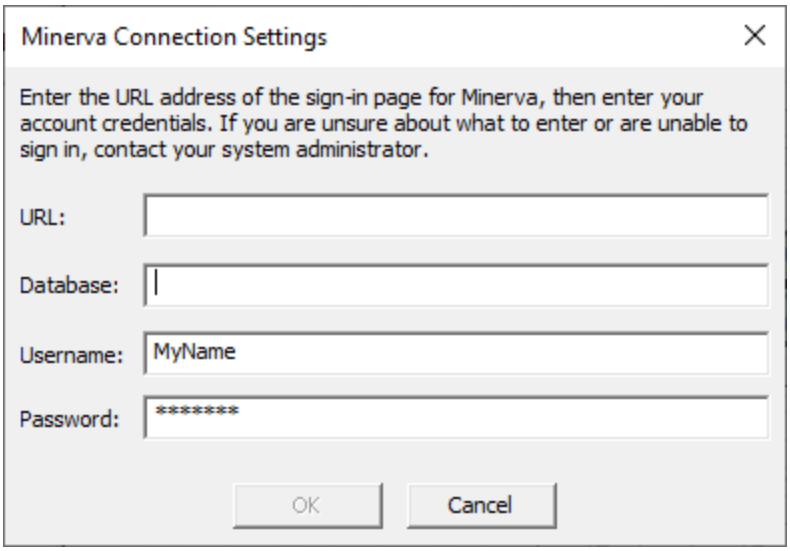
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Electronics Desktop interface. In the same way, it is also possible to save a project to Minerva. The image below shows the Minerva ribbon containing Minerva commands accessible through the desktop.



Minerva has its own HTML web interface with additional features as a knowledge management application that secures critical simulation data, and provides simulation process and decision support to simulation teams across geographies and functional silos.

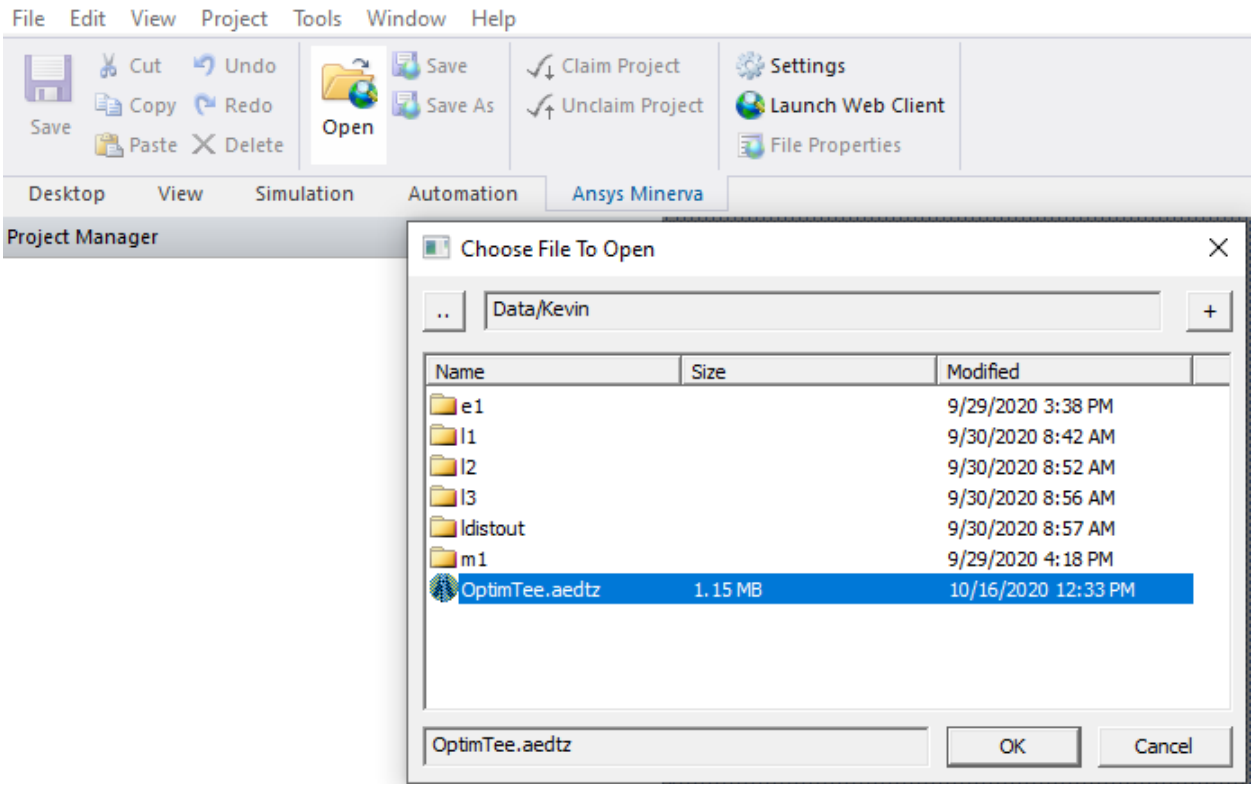
Minerva Settings



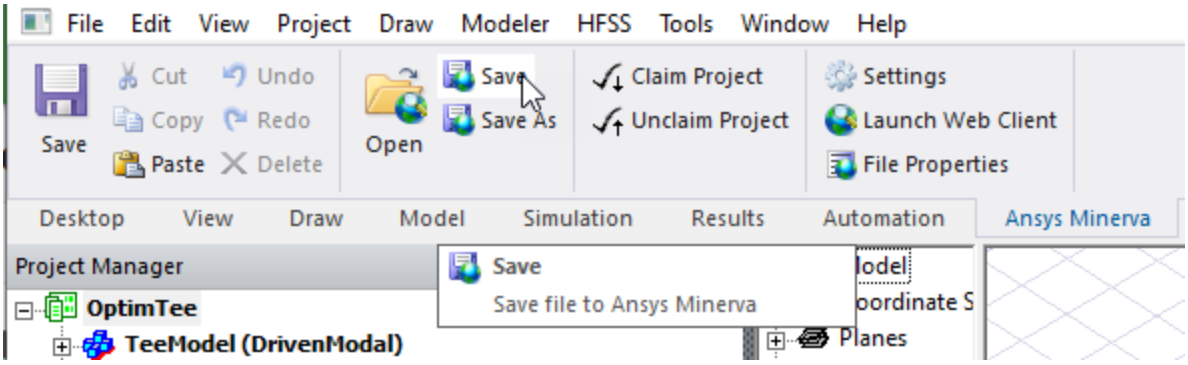
When you first login via the desktop, or to the Web Client, you must provide a Minerva URL, a database name, a user name, and a password. You will need to obtain these from your system administrator. Once you provide the connection settings, the other Minvera commands are enabled.

Open – opens the Choose File to Open browser that lets you select and download an archive file, extract it, and open it. The [..] button moves up one level in the file hierarchy. The [+] button creates a new folder at the current level.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

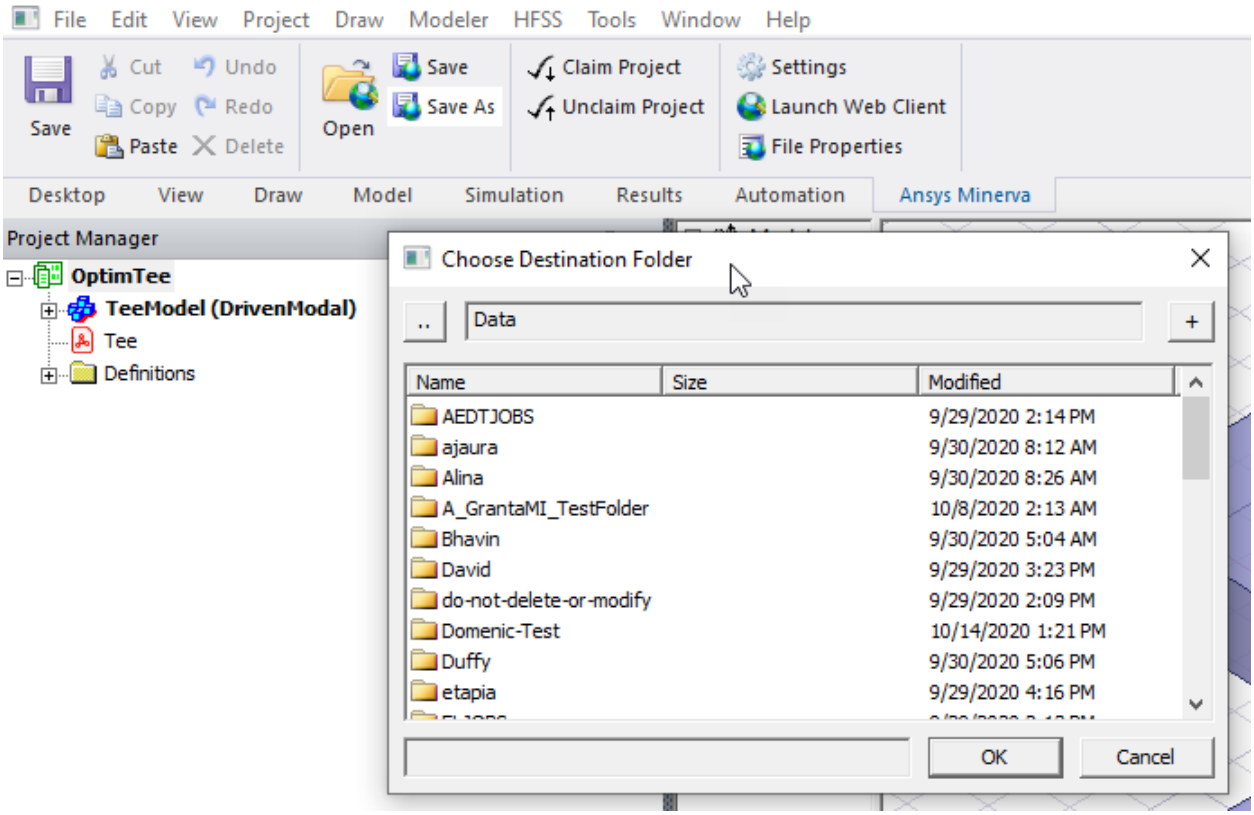


Save – archives the current project, and saves it to a remote Minerva system.



Save As – prompt for remote file, archive project, save to remote Minerva system.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In order to prevent users from overwriting each other's work, Minerva has added the concept of claiming and unclaiming files. A file claimed by one user cannot be overwritten on the Minerva server by another user. Unclaimed files can be overwritten by anyone at any time. Once a file has been claimed by one user, it cannot be claimed by a different user until it has been unclaimed by the original claimer. Note: even if a project is claimed by one user, other users are free to download, open and edit the file locally.

The recommended flow when working on a project is

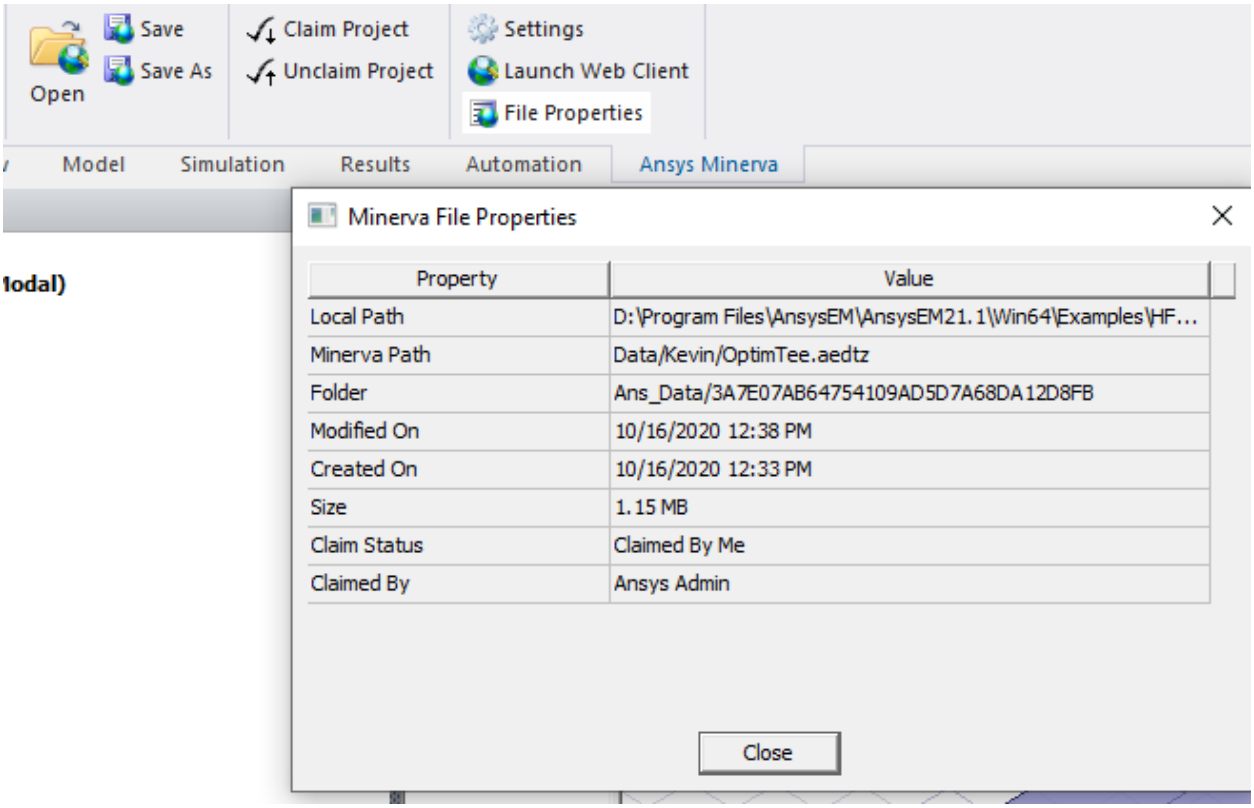
1. Claim the file, make sure you are working with the most recent version
2. Make edits
3. Save the file to Minerva
4. Unclaim the file

Claim Project – claims file so it cannot be modified by other users.

Unclaim Project – releases claimed file.

File properties – opens a dialog letting you view remote file properties. Notice the Claim Status and Claimed By properties.

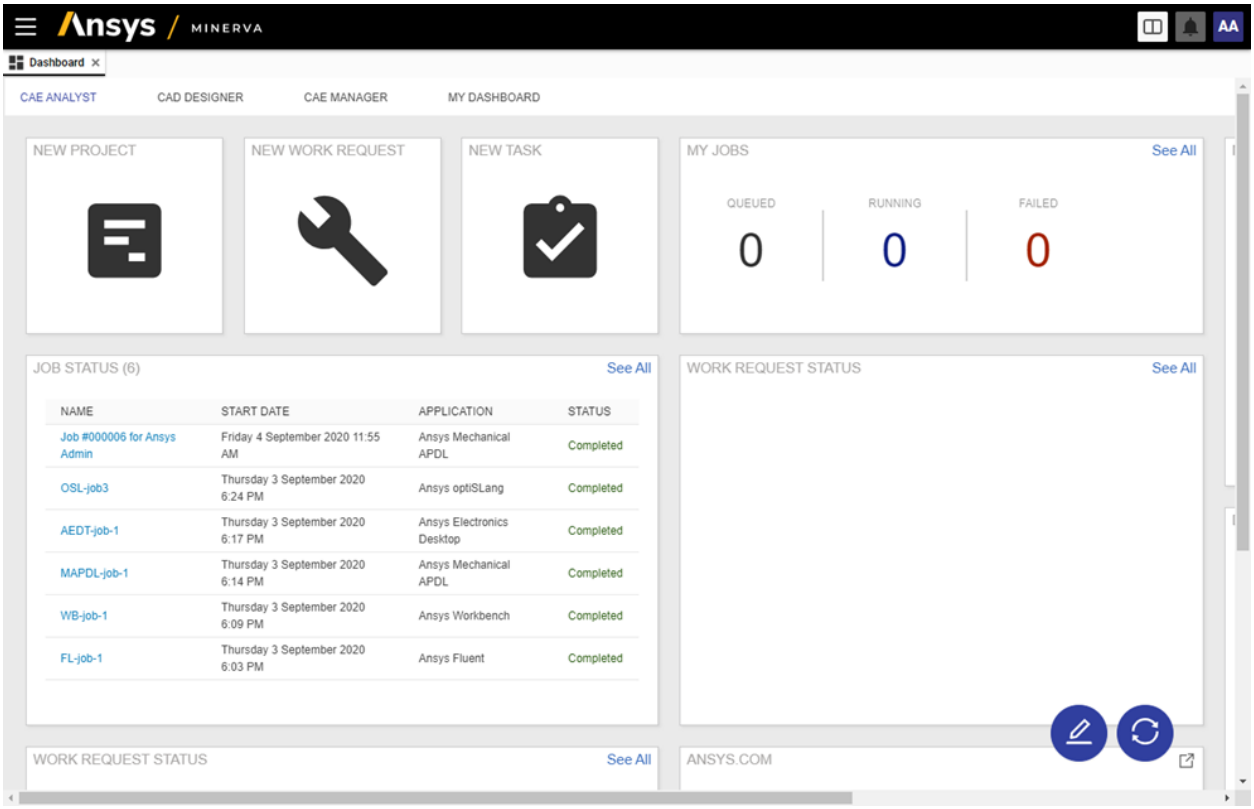
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Launch Web Client:

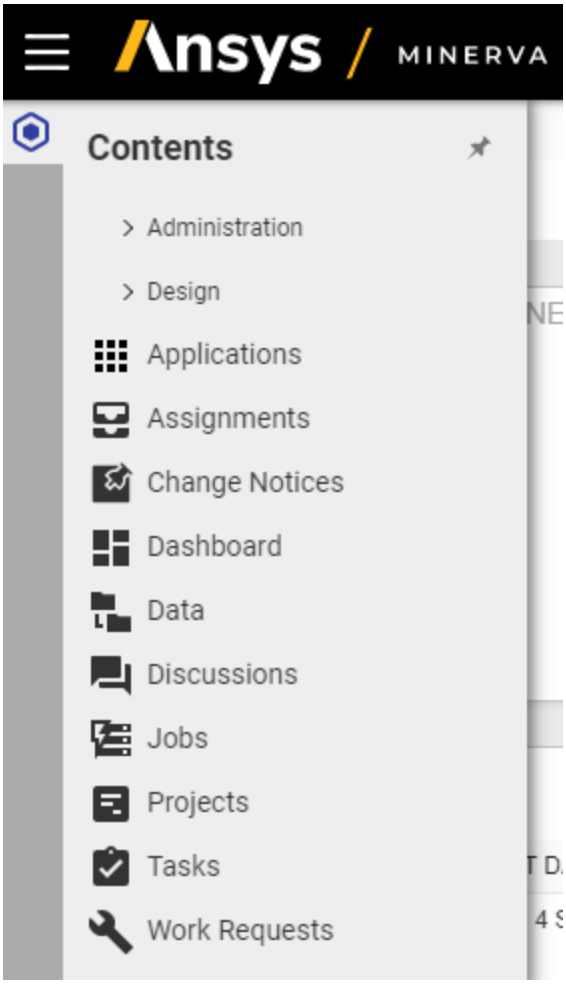
Ansys Minerva, powered by Aras, is a knowledge management application that secures critical simulation data, and provides simulation process and decision support to simulation teams across geographies and functional silos. Launching the Minerva Web Client gives access to these features. This opens a dialog calling for you to specify provide login information. It attempts to open a web browser window, so your default browser must be compliant.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



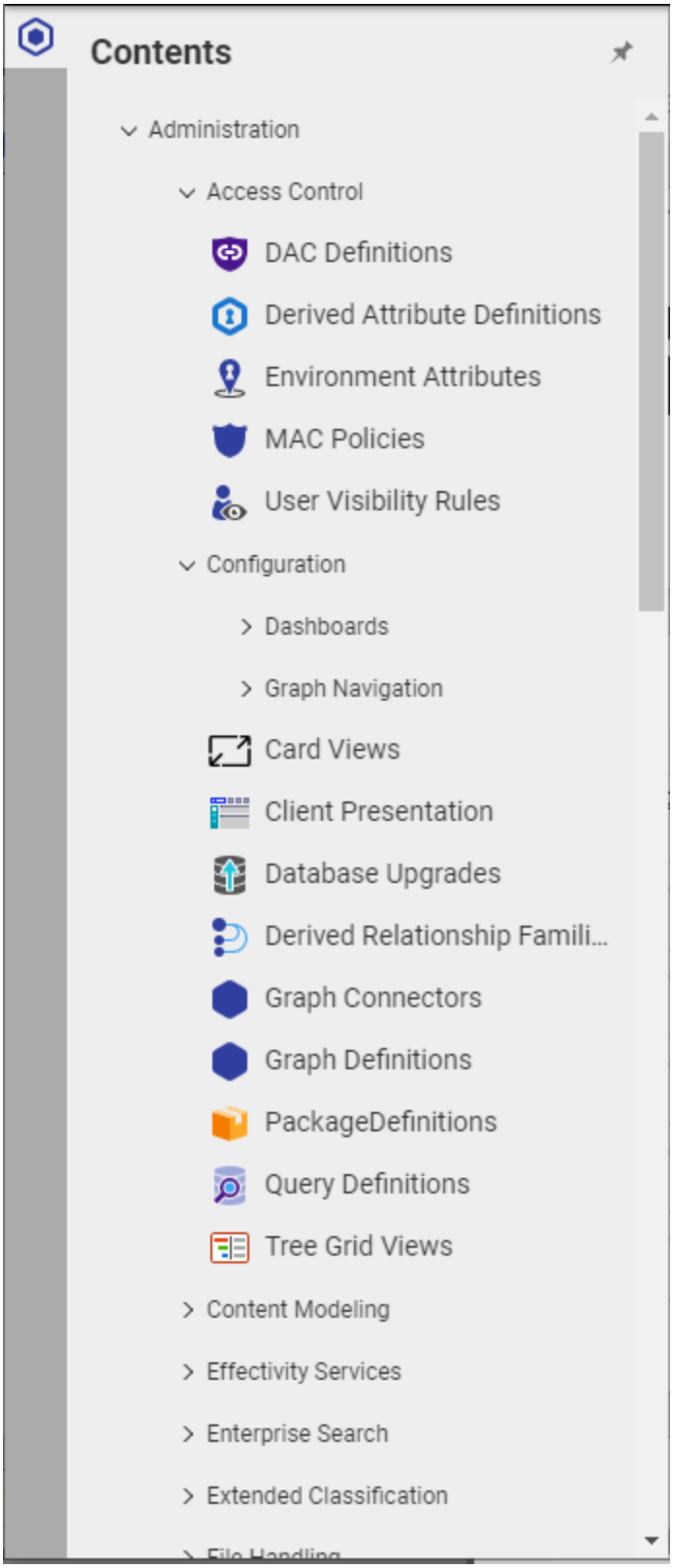
Clicking on the triple bar icon left of the Ansys logo opens up access to the more robust Minerva Web Client functionality.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The arrow icons for Administration open up further functionality.

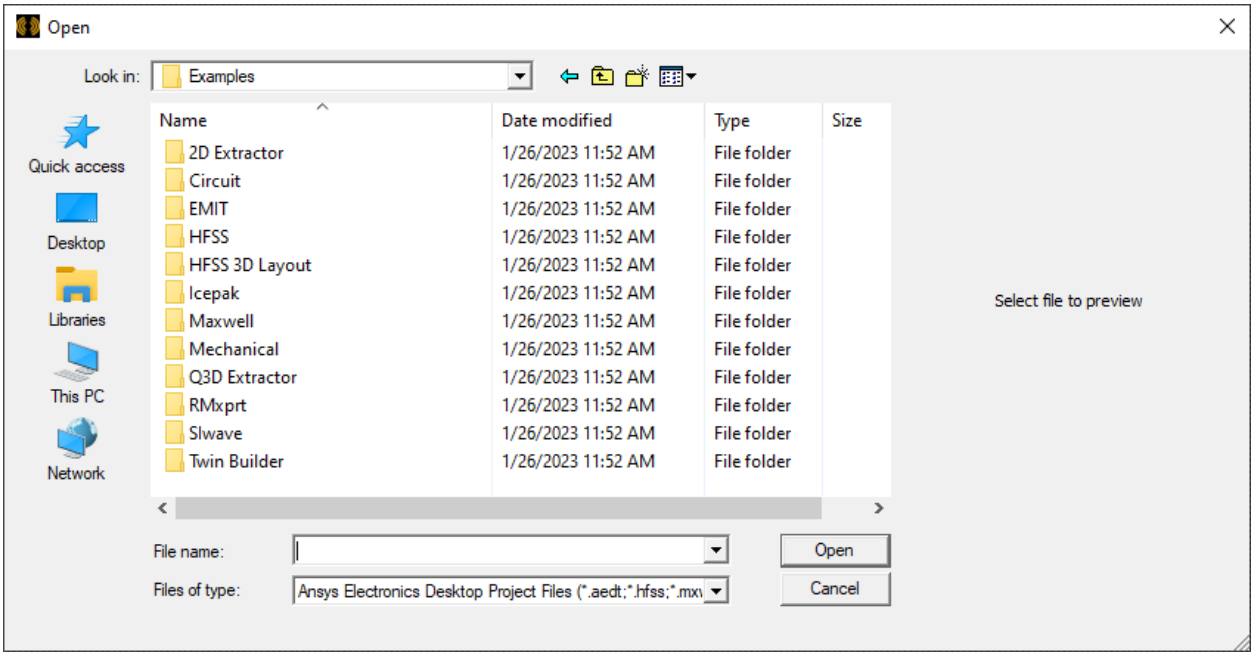
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example Projects

Your Ansys Electronics Desktop installation includes an example directory containing projects folders for several kinds of designs.



Example projects are organized by the design type.

Several of these projects are associated with detailed getting started guides.

Working with PEmag Designs

If **PEmag** is installed on your system, you can launch the PEmag desktop directly from the main Maxwell **Tools** menu. After you have created or opened a design in PEmag, PEmag can be used to generate both Maxwell 2D electrostatic designs and eddy current designs. Also, provided that both Maxwell and PEmag desktops remain open, you can modify the related Maxwell design using PEmag.

The first time you invoke PEmag from Maxwell, an “About PExprt Modeler [PEmag]” dialog box displays, informing you that modeling of magnetic components is disabled when PEmag is invoked from Maxwell. Uncheck the **I want to see this message...** checkbox to prevent this dialog box from displaying again.

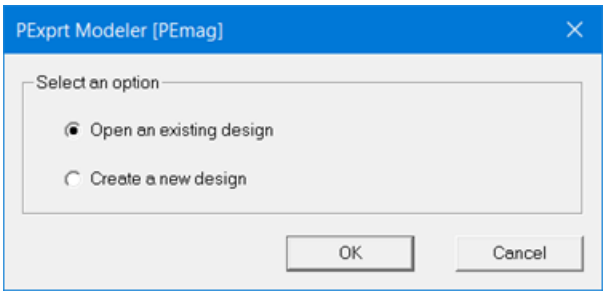
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: Refer to the documentation included with PEmag for detailed information on how to use that application.

Creating Maxwell Designs Using PEmag

To create a Maxwell design using PEmag:

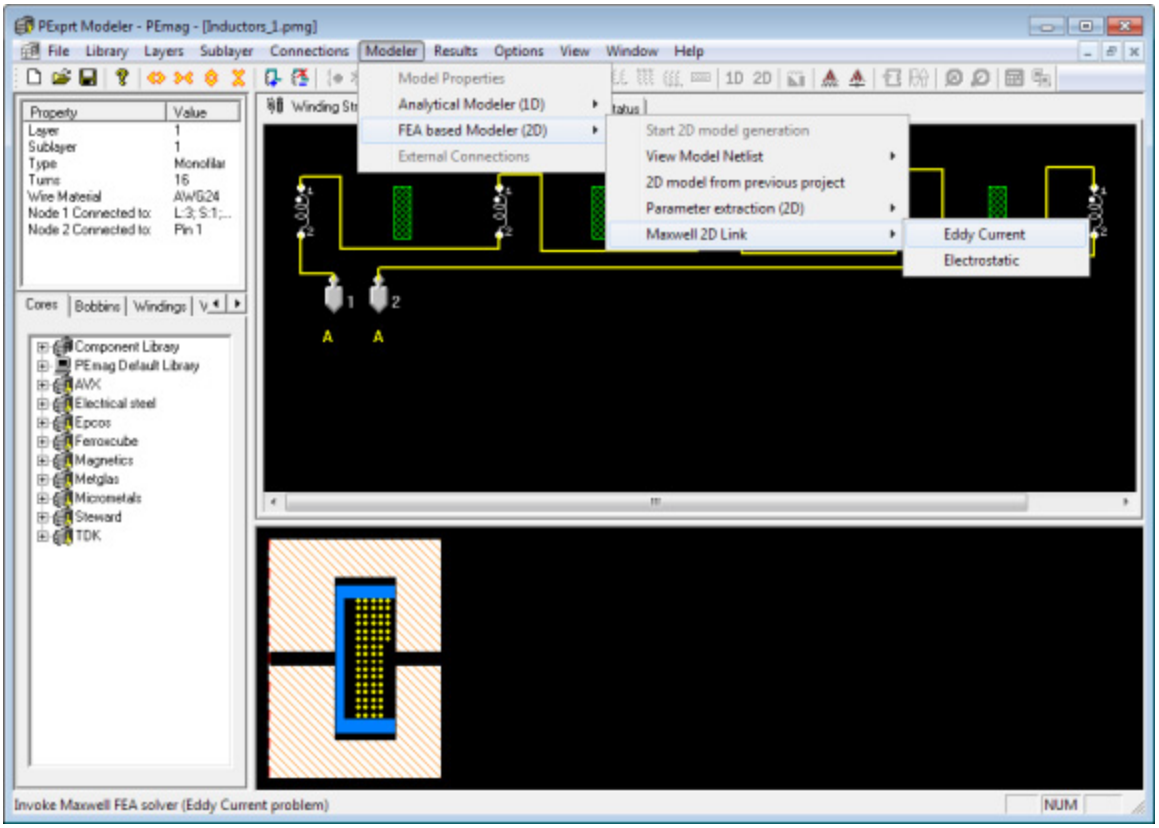
1. Select **Tools** > **PEmag** to launch the PEmag desktop application.
2. In PEmag, choose either to **Open an existing design**, or to **Create a new design**.



3. When finished creating the PEmag design, you can choose either **Modeler** > **FEA based Modeler(2D)** > **Maxwell 2D Link** > **Eddy Current** or **Modeler** > **FEA based Modeler (2D)** > **Maxwell 2D Link** > **Electrostatic** to have PEmag generate a corresponding design for Maxwell.

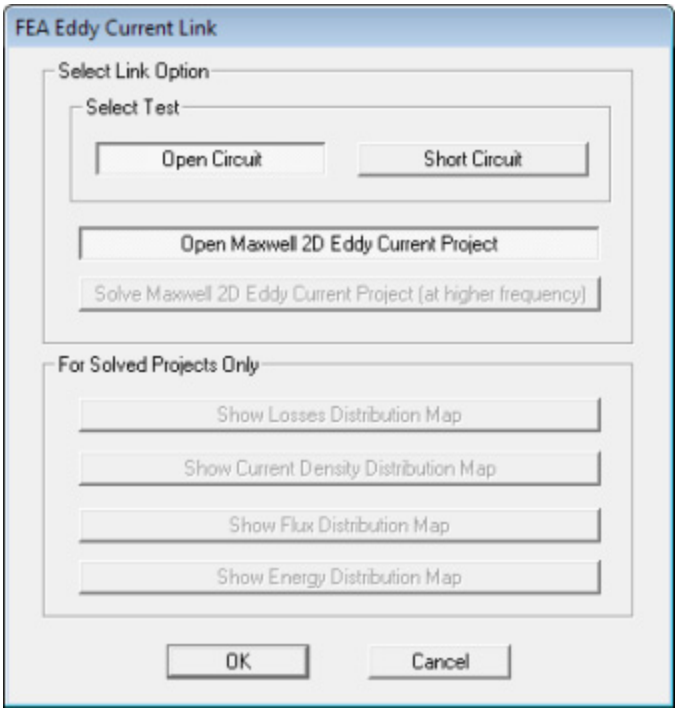
   You can also click on the **FEA Eddy Current** or **FEA Electrostatic** toolbar icons.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

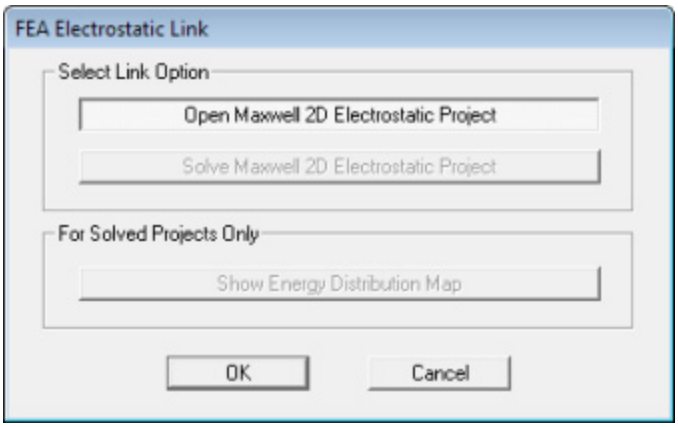


a. If you choose **Modeler** > **FEA based Modeler(2D)** > **Maxwell 2D Link** > **Eddy Current**, a window similar to the following displays.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



b. If you choose **Modeler** > **FEA based Modeler(2D)** > **Maxwell 2D Link** > **Electrostatic**, a window similar to the following displays.



4. Click **OK** to close the dialog box. PEmag generates a Maxwell 2D design and adds it to the Maxwell Project Manager and desktop. The new design is created corresponding to the PEmag model. The project is immediately saved on disk with the name, <*PEmag project name*>.aedt, in a new folder named <*PEmag project name*>.pjt, which is under the same directory as the PEmag project.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- Adding a Maxwell 2D design via PEmag cannot be undone.
- After a Maxwell design is created through PEmag, attempting to close Maxwell while PEmag is still open displays a message to inform that the Maxwell desktop cannot be closed while PEmag is still open.

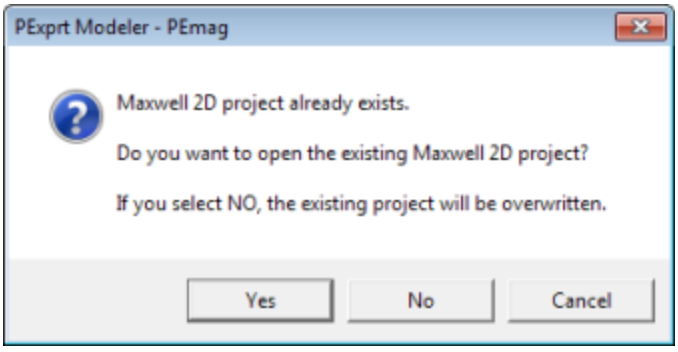
Modifying Maxwell Designs Using PEmag

After a Maxwell project is created through PEmag, the Maxwell project and design can be modified in the Maxwell desktop while the PEmag desktop is still open. The PEmag model can also be modified through PEmag desktop while the Maxwell desktop is open.

Note:

- Modifying a Maxwell 2D design via PEmag cannot be undone.
- After a Maxwell design is modified through PEmag, attempting to close Maxwell while PEmag is still open displays a message that says that the Maxwell desktop cannot be closed while PEmag is still open.
- Modifications made in the Maxwell project are not updated in the PEmag project.

If you modify the Maxwell 2D design, or add a new Maxwell design to the project in Maxwell or if you modify the PEmag model in the PEmag desktop and regenerate the Maxwell design (by selecting either **Modeler** > **FEA based Modeler(2D)** > **Maxwell 2D Link** > **Eddy Current** or **Modeler** > **FEA based Modeler(2D)** > **Maxwell 2D Link** > **Electrostatic**), PEmag will pop up a dialog box to ask whether to open the existing Maxwell 2D project.



- If you select **Yes**, the Maxwell 2D project is reloaded from disk, and any changes made in Maxwell to the Maxwell 2D design and project are lost.

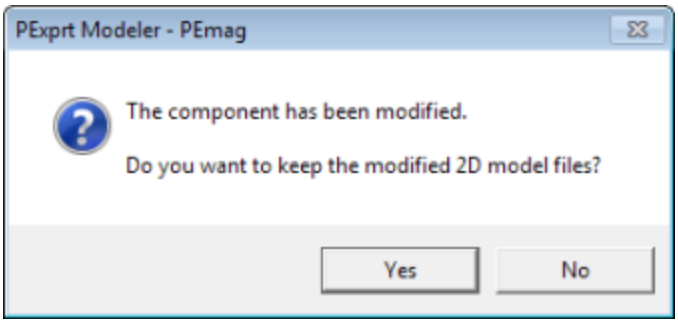
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you select **No**, the Maxwell design and project on the Maxwell desktop will be overwritten by the design and project generated through PEmag with the latest changes in the PEmag model. The Maxwell 2D project is immediately saved to the disk.

If you launch PEmag from Maxwell with no Maxwell projects loaded, then open an existing PEmag project that has previously been used to generate a Maxwell project, modify the PEmag model and regenerate the Maxwell design (by selecting either **Modeler** > **FEA based Modeler (2D)** > **Maxwell 2D Link** > **Eddy Current** or **Modeler** > **FEA based Modeler(2D)** > **Maxwell 2D Link** > **Electrostatic**), PEmag will pop up a dialog (see above) to ask whether to open the existing Maxwell 2D project.

- If you click **Yes**, the Maxwell 2D project on disk is loaded to the Maxwell desktop.
- If you click **No**, PEmag generates a Maxwell project and design on the Maxwell desktop, corresponding to the modified PEmag model. The project is immediately saved to the disk, overwriting the previous project on disk and deleting the previous solutions (if any).

If you modify a PEmag model and regenerate the Maxwell design; then attempt to close PEmag, a dialog box pops up asking whether to keep the modified 2D model files.



- If you click **Yes**, PEmag closes and nothing is changed.
- If you click **No**, PEmag closes, the Maxwell design and project is deleted from disk, but the modified Maxwell design remains on the Maxwell desktop, and you can save the project to disk, if desired. If not, closing Maxwell will permanently lose the project.

Solving Maxwell Designs Generated by PEmag

Maxwell designs generated by PEmag are fully set up and are ready for analysis. Note the following:

- After solving the Maxwell design generated by PEmag, if you subsequently modify the PEmag model and regenerate the Maxwell design, the old Maxwell design is overwritten and the solutions are deleted.
- While the Maxwell design is being solved on the Maxwell desktop, if you modify the corresponding PEmag design on the PEmag desktop and invoke PEmag to regenerate the Maxwell design, the solver process on the Maxwell desktop will be aborted, and the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Maxwell design that has been modified by PEmag is loaded into Maxwell and saved to the disk simultaneously.

Working with Variables

A variable is a numerical value, mathematical expression, or mathematical function that can be assigned to a design parameter in **Maxwell 3D**, **Maxwell 2D**, or **RMxprt**. You can assign a variable to any dimension, material property, or output value. Variables are useful in the following situations:

- You expect to change a parameter often.
- You expect to use the same parameter value often.
- You intend to run a parametric analysis, in which you specify a series of variable values within a range to solve.
- You intend to optimize a parameter value by running an optimization analysis or using Design of Experiments to generate a response surface.
- You intend to run a convergence on an output variable.
- You intend to animate a plot against a variable.

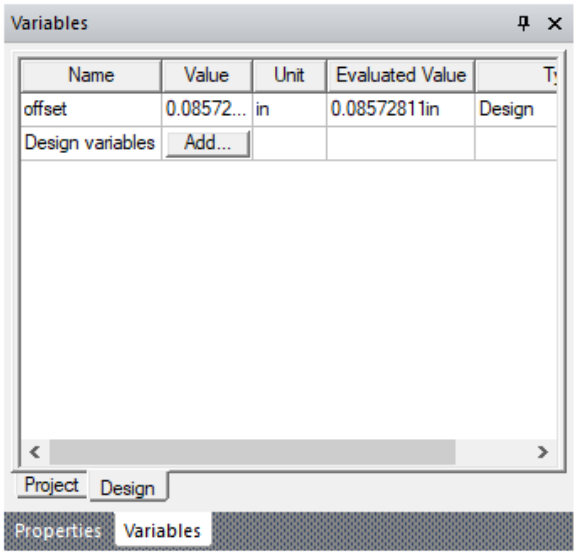
Variables can be regular variables, array index variables, or arrays. A project variable's value can reference other project variables or intrinsics. Values can be a number, variable, or mathematical expression.

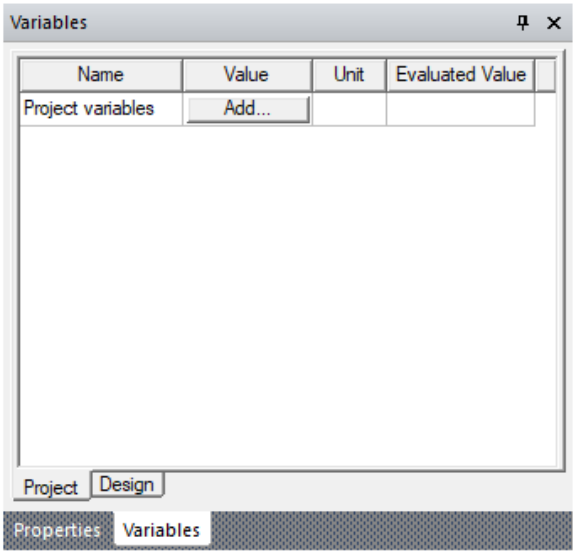
There are two types of variables in Maxwell:

Project Variables	A project variable can be assigned to any parameter value in the Maxwell project in which it was created. Maxwell differentiates project variables from other types of variables by prefixing the variable name with dollar sign symbol ($). You can manually include the $ in the project variable's name, or Maxwell will automatically append the project variable's name after you define the variable. Project variables can be designated as Design, ArrayIndex or Separator variables .
Design (Local) Variables	A design variable can be assigned to any parameter value in the Maxwell design in which it was created. From the Design Variables **Properties** dialog box, you can Add, Add Array, Edit, or Remove design variables. Design Variables can be designated as Design, ArrayIndex or Separator variables and as Post-Processing Variables.

Clicking **View** > **Variables** brings up a dockable variable window that is associated with the active project and/or design. When there is an active project, there will be a corresponding project variable tab. When there is an active design, there will be a corresponding design variable tab. Each tab contains an **Add...** button allowing creation of new variable of this type. If variables exist for the Project or Design, they are shown in the corresponding tab.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

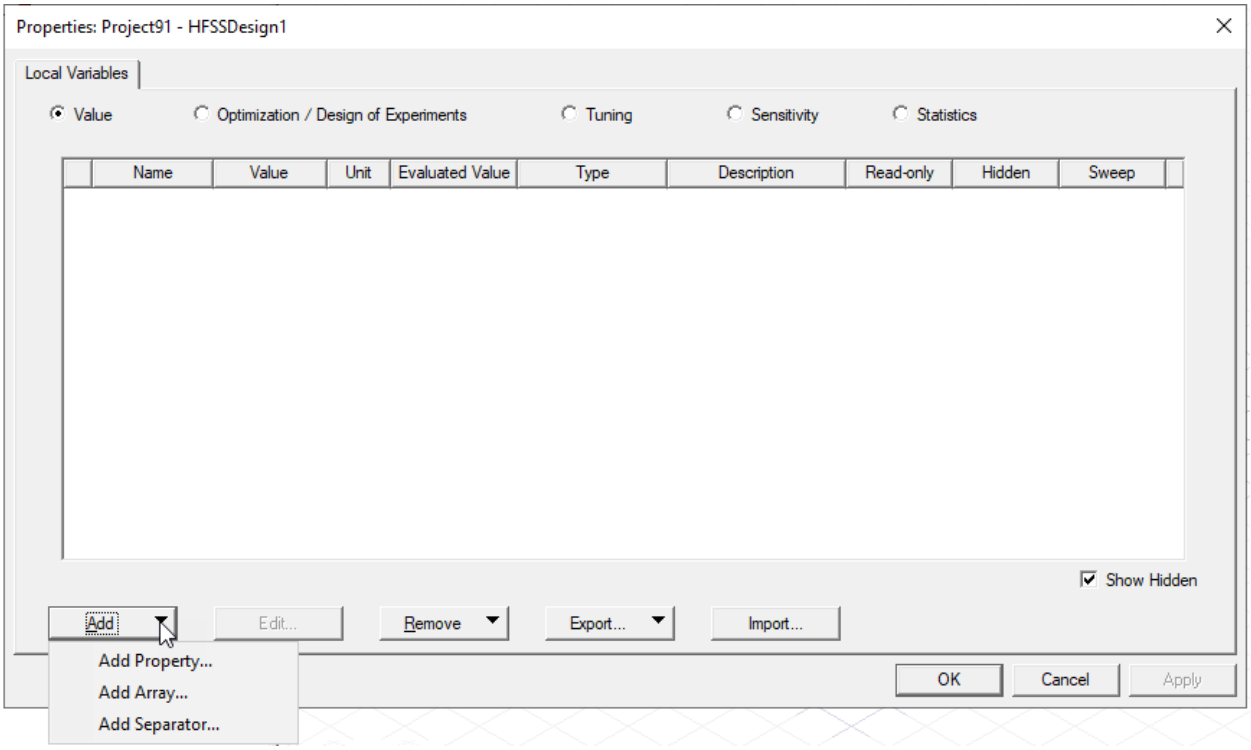




Importing and Exporting Variables

Once you have defined Design and/or Project Variables, you can export them to a csv file, and import them to another project or design. If there are naming conflicts on import, these are flagged as errors. The **Properties** dialogs for Design Variables and Project variables include **Import...** and **Export...** buttons.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Working with Project Variables

Project variables are available across all hierarchical levels of a project. A project variable can be assigned to a parameter value in the Maxwell project in which it was created. Maxwell differentiates project variables from other types of variables by prefixing the variable name with the dollar sign symbol ($). You can manually include the $ in the project variable's name when you create it, or Maxwell will automatically append it after you define the variable.

The **Project Variables** menu item allows you to access three tabs:

- **Project Variables** – variables, which you define, that apply to the entire project.
- **Intrinsic Variables** – pre-defined variables that cannot be changed. See: List of Intrinsic Variables.
- **Constants** – pre-defined constants that cannot be changed. See: List of Constants.

Adding a Project Variable

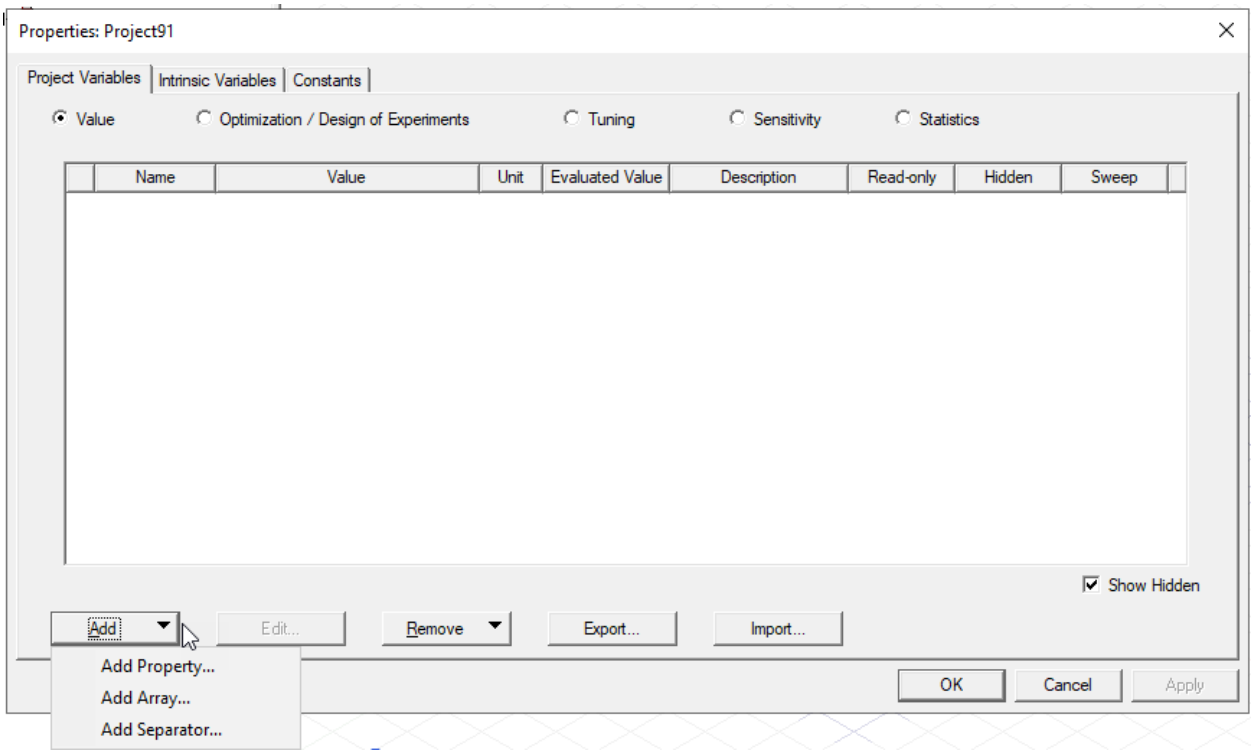
To define a Project Variable:

1. Access **Project Variables** one of three ways:
    - Click **Project** > **Project Variables**.
    - Right-click the project name in the Project Tree and select **Project Variables**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- From a menu in the lower left corner of the following Optimization dialogs: **Parametric**, **Optimization**, **Sensitivity**, **Statistical**, **Design of Experiments**, and **Design Xplorer Setup**. Click **Edit Variables** and from the menu select **Edit Project Variables**.

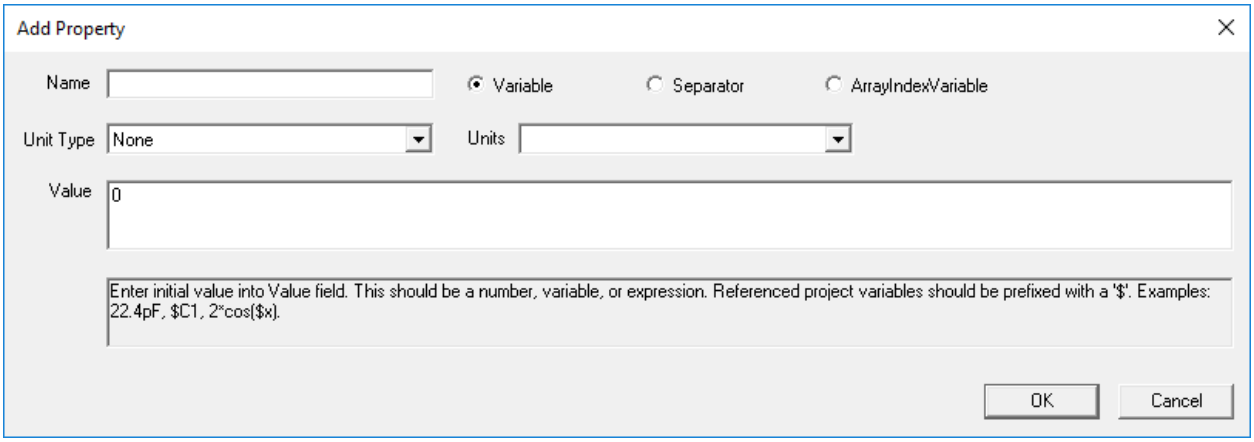
The **Properties** window appears, on the **Project Variables** tab.



2. From the **Project Variables** tab, click **Add**.

   The **Add Property** dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Enter information for the variable, as applicable:
   - **Name** – project variable names must start with the dollar sign symbol ($), followed by a letter. The name can contain alphanumeric characters and underscores (_). You cannot use the names of Intrinsic Variables or pre-defined Constants.
   - **Variable Type** – use the **Variable**, **Separator**, and **ArrayIndexVariable** radio buttons to select the variable type.

     Your selection impacts which properties you can edit:

Variable Type	Editable Properties
Variable	Unit Type, Units, Value.
Separator	Name. A separator variable provides a bolded name for a blank line to facilitate grouping variables in variable lists.
Array Index Variable	Associate Array Variable, Value

   - **Unit Type** – for Variables, use the drop-down menu to select a type from the list (for example, Charge, Density, Energy, ...). “None” is the default.

     When you select a Unit Type, the choices in drop-down menu for the Units text box adapt to that unit type. For example, selecting Length as the Unit Type causes the Unit menu to show a range of metric and english units for length. Similarly, if you select the Unit Type as Resistance, the Units drop down lists a range of standard Ohm units.

   - **Units** – for Variables, use the drop-down menu to select a unit of measure.
   - **Value** – for Variables and ArrayIndexVariables, enter a number, variable, or mathematical expression. The quantity entered will be the current (or default) value for the variable. If the mathematical expression includes a reference to an existing variable, this variable is treated as a dependent variable. The units for a dependent

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

variable will automatically change to those of the independent variable on which the value depends. Additionally, dependent variables, though useful in many situations, cannot be the direct subject of optimization, sensitivity analysis, tuning, or statistical analysis.

For ArrayIndexVariables, the index reference can be a constant (for example, 1), an index (for example, ii) or an expression (for example, ii + 1). This allows you to sweep the index and simulate for different values that are stored in the array variable itself. In particular, it also enables you to sweep different text strings. This allows you to set a property to different string values as the index is swept.

> **Warning:**
>
> If you include the variable's units in its definition (in the **Value** text box), do not include the variable's units when you enter the variable name for a parameter value.

- **Associate Array Variable** – for ArrayIndexVariables, select an associated array from the drop-down list. You must have previously created an array.

4. Click **OK**.

   The new variable appears in the list. You can sort project variables by clicking the Name column header. Clicking once sorts them in ascending order, noted by a triangle pointing up. Clicking against sorts in descending order, noted by a triangle pointing down. Clicking a third time sorts in original order, with no triangle.

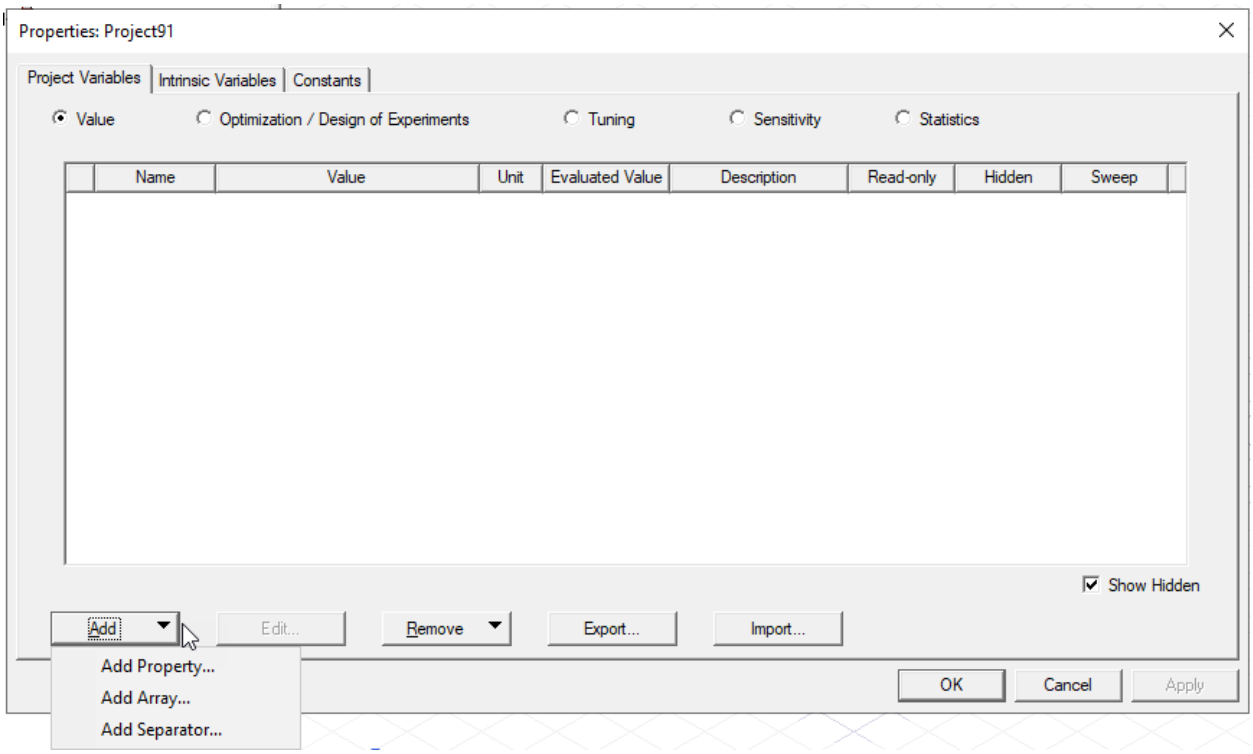
5. If desired, use the check boxes to designate a variable as **Read-only**, **Hidden**, or **Sweep**.
   - **Read-only** – when selected, the variable's name, value, unit, and description cannot be modified.
   - **Hidden** – hidden variables do not appear in the **Properties** window unless **Show Hidden** is selected.
   - **Sweep** – allows you to designate variables to include in solution indexing as a way to permit faster post-processing. Variables with the Sweep check box cleared are not used in solution indexing. If a solution exists, selecting or clearing a variable's Sweep setting produces a warning that the change will invalidate existing solutions. To continue, click **OK** to dismiss the warning.
6. Click **Apply** to apply changes.
7. Click **OK** to exit the window.

The new variable can be assigned to a parameter value in the project in which it was created. You can enable defined variables for Optimization/Design of Experiments, Tuning, Sensitivity, or Statistics. See: Optimetrics.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Importing and Exporting Project Variables

Once you have defined Project Variables, you can export them to a csv file, and import them to another project or design. If there are naming conflicts on import, these are flagged as errors. The **Properties** dialogs for and Project variables include **Import...** and **Export...** buttons.



When you click **Export....** a dialog lets you name the file and select a location. When you **Save**, if the file exists, an overwrite prompt should pop up.

Deleting a Project Variable

To remove a Project Variable:

1. Access **Project Variables** as described above.

   The **Properties** window appears, on the **Project Variables** tab.

2. Remove one or more variables:
   - To remove a specific variable, select it and click **Remove** > **Remove Selected**.
   - To remove all unused variables, click **Remove** > **Remove All Unused**.
   - To force the removal of all unused variables, including those in the project's undo/redo history, click **Remove** > **Force Remove All Unused**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Editing a Project Variable

To edit a Project Variable:

1. Access **Project Variables** as described above.

   The **Properties** window appears, on the **Project Variables** tab.

2. Select the variable you want to edit and click **Edit**.
3. Change the properties as desired and click **OK**.

Adding a Project Variable Array

You can define array variables that contain numbers or strings. Number array variables can be used in component property expressions, while string array variables can be used in certain component property values.

> **Note:**
>
> Text array variables can be used in certain component property values, but not all, and cannot be combined with operators to form more complex expressions.
>
> The index for a text array variable reference can either be a constant (1) or can be an index (ii) or even an expression (ii + 1). This allows you to sweep the index and simulate for different values that are stored in the array variable itself. In particular, it also enables you to sweep different text strings. This allows you to set a property to different string values as the index is swept. The following are properties that currently allow text array variables:
>
> - V_PRBS
> - V_PRBS_JITTER
> - V_PRBSD
> - V_PRBSG_JITTER
> - V_PSK
> - V_QAM
> - V_CPM

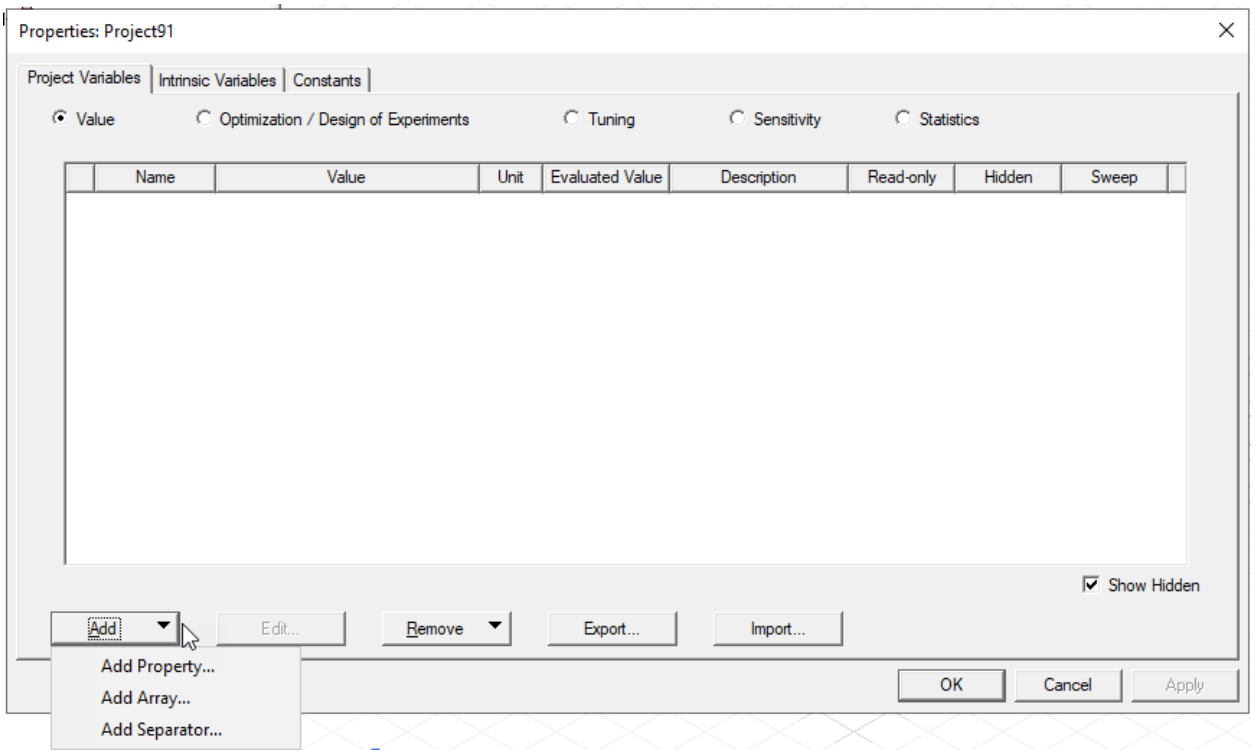
To define an array variable, you must first define a Project Variable Array:

1. Access **Project Variables** one of three ways:
   - Click **Project** > **Project Variables**.
   - Right-click the project name in the Project Tree and select **Project Variables**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- From a menu in the lower left corner of the following Optimization dialogs: **Parametric**, **Optimization**, **Sensitivity**, **Statistical**, **Design of Experiments**, and **Design Xplorer Setup**. Click **Edit Variables** and from the menu select **Edit Project Variables**

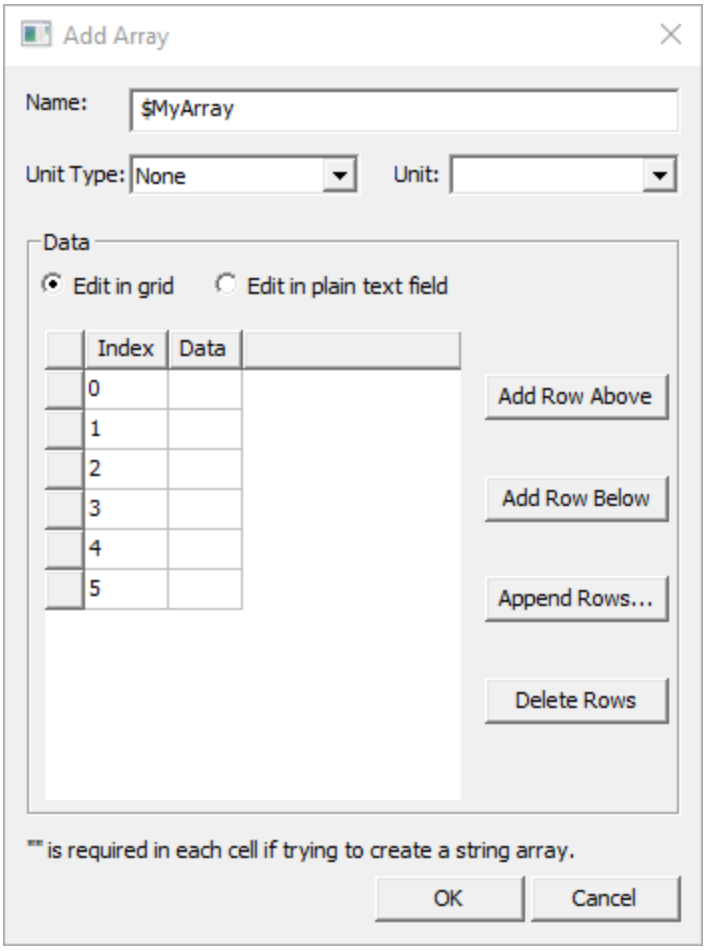
The **Properties** window appears, on the **Project Variables** tab.



2. Click **Add Array** > **Add Array...**.

   The **Add Array** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The value grid displays each array item's **Index** number, and the **Data** associated with that index number.

3. Give the variable array a **Name**, and select a **Unit Type** and **Unit** from the drop-down menus.
4. Use the control buttons to add, delete, and reposition rows in the value grid at left. The default is **Edit in grid**, but you may select **Edit in plain text field**.

> **Note:**
>
> Quotation marks (“ ”) are *required* as delimiters when array values are entered in either the grid or text field.

5. When you have finished entering values, click **OK**.

   The array appears in the **Project Variables** list.

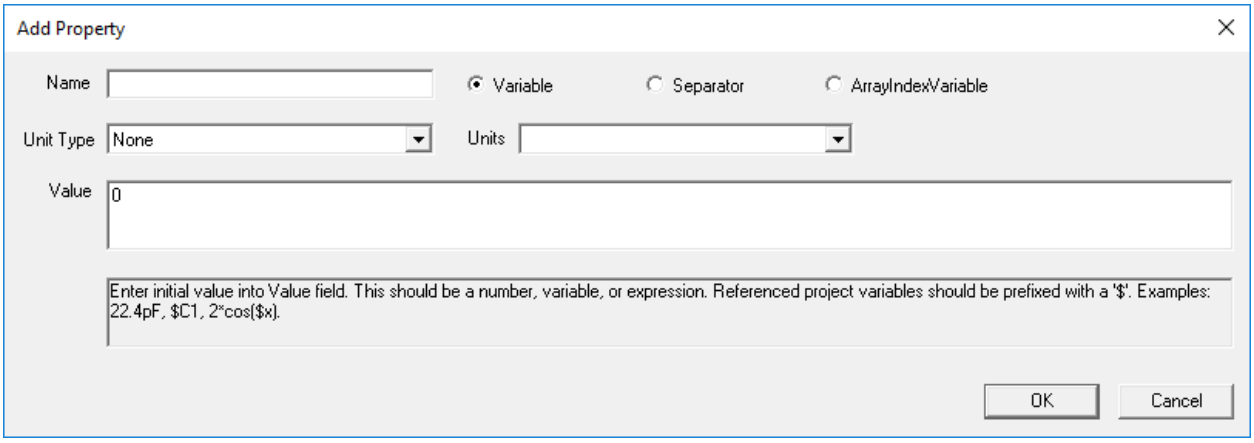
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can add a variable to the array by adding an ArrayIndexVariable on the **Project Variables** tab.

1. Access **Project Variables** one of three ways:
   - Click **Project** > **Project Variables**.
   - Right-click the project name in the Project Tree and select **Project Variables**.
   - From the **Setup Optimization** dialog box, click **Edit Variables** > **Edit Project Variables**.

   The **Properties** window appears, on the **Project Variables** tab.
2. From the **Project Variables** tab, click **Add**.

   The **Add Property** dialog box appears.



3. Enter information for the variable, as applicable:
   - **Name** – project variable names must start with the dollar sign symbol ($), followed by a letter. The name can contain alphanumeric characters and underscores (_). You cannot use the names of Intrinsic Variables or pre-defined Constants.
   - **Variable Type** – select ArrayIndexVariable.
   - **Value** – enter a number, variable, or mathematical expression. The quantity entered will be the current (or default) value for the variable. If the mathematical expression includes a reference to an existing variable, this variable is treated as a dependent variable. The units for a dependent variable will automatically change to those of the independent variable on which the value depends. Additionally, dependent variables, though useful in many situations, cannot be the direct subject of optimization, sensitivity analysis, tuning, or statistical analysis.

     For ArrayIndexVariables, the index reference can be a constant (for example, 1), an index (for example, ii) or an expression (for example, ii + 1). This allows you to

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

sweep the index and simulate for different values that are stored in the array variable itself. In particular, it also enables you to sweep different text strings. This allows you to set a property to different string values as the index is swept.

> **Warning:**
>
> If you include the variable's units in its definition (in the **Value** text box), do not include the variable's units when you enter the variable name for a parameter value.

- **Associate Array Variable** – for ArrayIndexVariables, select an associated array from the drop-down list. You must have previously created an array.

4. Click **OK**.

List of Intrinsic Variables

The Ansys Electronics Desktop software recognizes a set of intrinsic variables that can be used to define expressions. Intrinsic variable names are reserved and may not be used as user-defined variable names.

To view the list of Intrinsic Variables, click **Project** > **Project Variables** and select the **Intrinsic Variables** tab.

Name	Unit	Description
_Empty		Empty value, taken to be model default by simulator (Twin Builder)
_I1	mA	Armature current; **this variable is only used in RMxprt**.
_I2	mA	Exciting current; **this variable is only used in RMxprt**.
_I3	mA	Current; **this variable is only used in RMxprt**.
_I4 through _I9		Not used
_t,**_u**,**_v**		Variables to define parametric equation-based curve.
_V1 to **_V9**	mV	Port Voltage in user-defined model (V).
Ang	Ang	Angle. Postprocessing variable
Budget Index		Postprocessing variables; cannot be set by the user
Distance	mm	
Electrical Degree	deg	Electric degree of the rotating machine; cannot be set by the user.**This variable is not applicable to Maxwell; it is only used in RMxprt.**
F	GHz	Frequency of the circuit/system analysis
F1, **F2**, **F3**	GHz	Frequency tones 1, 2, 3 in harmonic balance analysis. (Hz).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Unit	Description
FNoi	GHz	Offset noise frequency in harmonic balance noise analysis.
Freq	Hz	Frequency for analysis and postprocessing; cannot be set by the user
Hmax	ns	
Hmin	ns	
Ia, **Ib**	mA	Postprocessing variables; cannot be set by the user
Index		Identifier for a data point; cannot be set by the user
IWave Phi, **IWave Theta**	deg	Incident wave spherical coordinate variables. Phi is the angle from the origin in the z direction and Theta the angle from the x-axis.
Normalized Deformation, **Normalized Distance**		Postprocessing variables; cannot be set by the user
OP	mW	Postprocessing variables; cannot be set by the user
Pass		Post processing variables, not settable by the user.
Phase	deg	Angle of a complex number.
Phi	deg	The angle measured from the x-axis and can be from 0 to 360 degrees.
R	mm	R is the cylindrical coordinate system variable.
Rho		Rho is the spherical coordinate system variable.
Rspeed	rpm	Speed of the machine.
Spectrum	GHz	Postprocessing variable.
Temp	cel/deg	Analysis temperature (deg)
Tend	ns	
Theta	deg	The angle measured from the z-axis, which is the axis perpendicular to the plane of the work space, and must be from 0 to 180 degrees.
Time	ns	Time point in transient analysis
Time0	ns	Time 0 point in a transient analysis
Vac, **Vbe**, **Vce**, **Vds**, **Vgs**	mV	Post processing variables; cannot be set by the user
X,**Y**,**Z**	mm	Point coordinate variables in the modeler.
ZAng, **XRho**	deg	A spherical coordinate system; cannot be set by the user

Not shown on the Intrinsic Variables tab are the **Speed** and **Position** variables that are available only in Maxwell 2D and 3D transient designs with Motion Setup. These variables are treated as intrinsic variables and are not user-editable. Their values are dynamically calculated during solve time and are expressed in standard SI units. While these variables are related to motion

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

setups, they are not necessarily used in motion setups. For example, they may also be used with the transient design's excitations. See Motion Variables Tab for additional information.

List of Constants

To view the list of constants, click **Project** > **Project Variables** and select the **Constants** tab.

Name	Value	Description
abs0	-273.15	Absolute zero (C)
boltz	1.3806503e-23	Boltzmann constant (J/K)
c0	299792458	Speed of light in a vacuum (m/s)
e0	8.854187817e-12	Permittivity of vacuum (F/m)
elecq	1.602176462e-19	Electron charge (C)
eta	376.730313461	Impedance of vacuum (Ohm)
false	0	Boolean False
g0	9.80665	Acceleration due to gravity of Earth (m/s^2)
mathE	2.718281828	Euler's number (Napier's constant)
pi	3.1415926535898	Ratio of circumference to diameter
planck	6.6260755e-34	Planck's constant (m^2*kg/s)
true	1	Boolean True
u0	1.2566370614359e-06	Permeability of vacuum (H/m)

Working with Design Variables

Design Variables (Local Variables) are only available within the Maxwell design for which they are defined. A design variable can be assigned to a parameter value in the design. You can also add a variable defined with an array of values. Choosing **Add Property** pops up an **Add Property** dialog with default radio button on Variable; choosing **Add Array** pops up an **Add Array** dialog; choosing **Add Separator** pops up an **Add Property** dialog with default radio button on Separator.

The **Design Properties** menu item allows you to access up to three tabs, depending on your design:

- **Parameter Defaults** – local variables with default values that can be overridden in instances of a design. For example, if three subcircuit instances contain a parameter default C1 that is defined as equal to 11.3pF, C1 may be overridden as 11.8pF in the first instance, overridden as 10.9pF in the second, and left at its default value of 11.3pF in the third. A property value that has been set by means of a parameter default is called a *passed parameter*. Parameter defaults can be defined at the design level.
- **Local Variables** – variables, which you define, that apply to the current design only. Local variables can be defined at the design level.

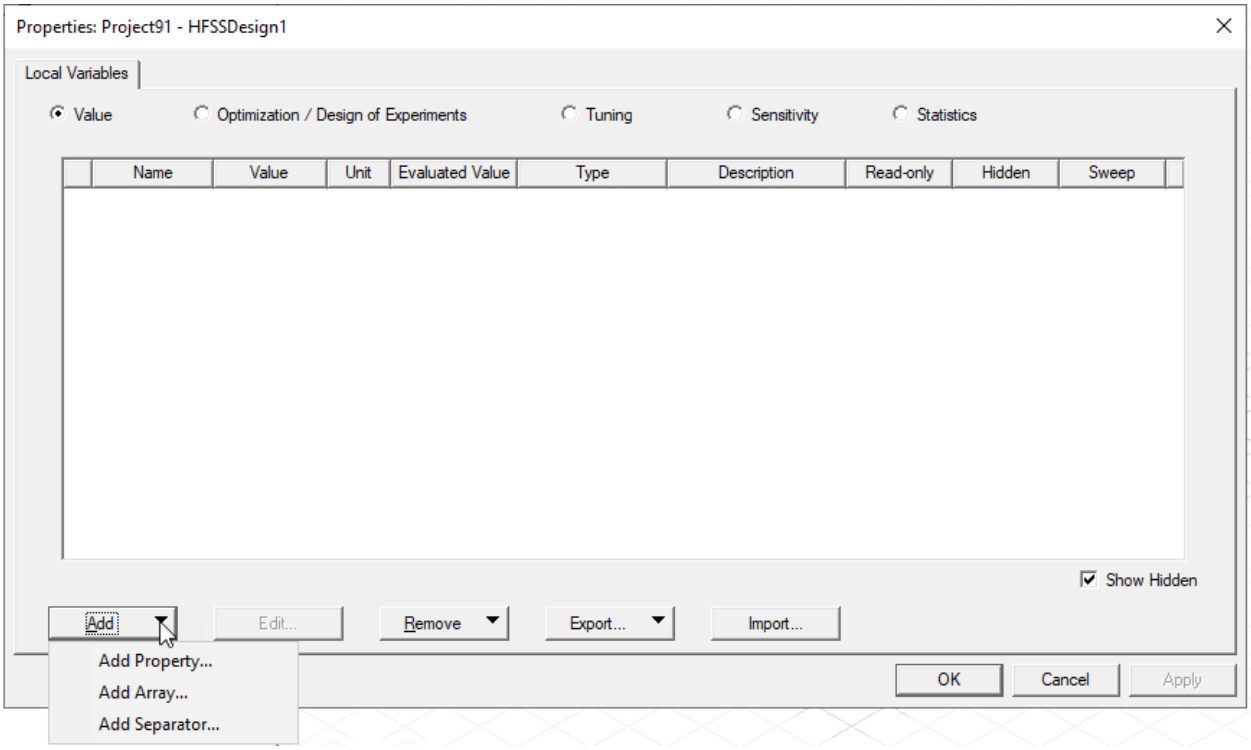
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **General** – pre-defined design variables that cannot be changed. You cannot use the names of these variables when creating a new variable.

> **Note:**
>
> Not all tabs appear for all designs.

Importing and Exporting Variables

Once you have defined Design Variables, you can export them to a csv file, and import them to another project or design. If there are naming conflicts on import, these are flagged as errors. The Properties dialogs for Design Variables include **Import...** and **Export...** buttons.



Defining Local Variables at the Design Level

Adding a Local Variable

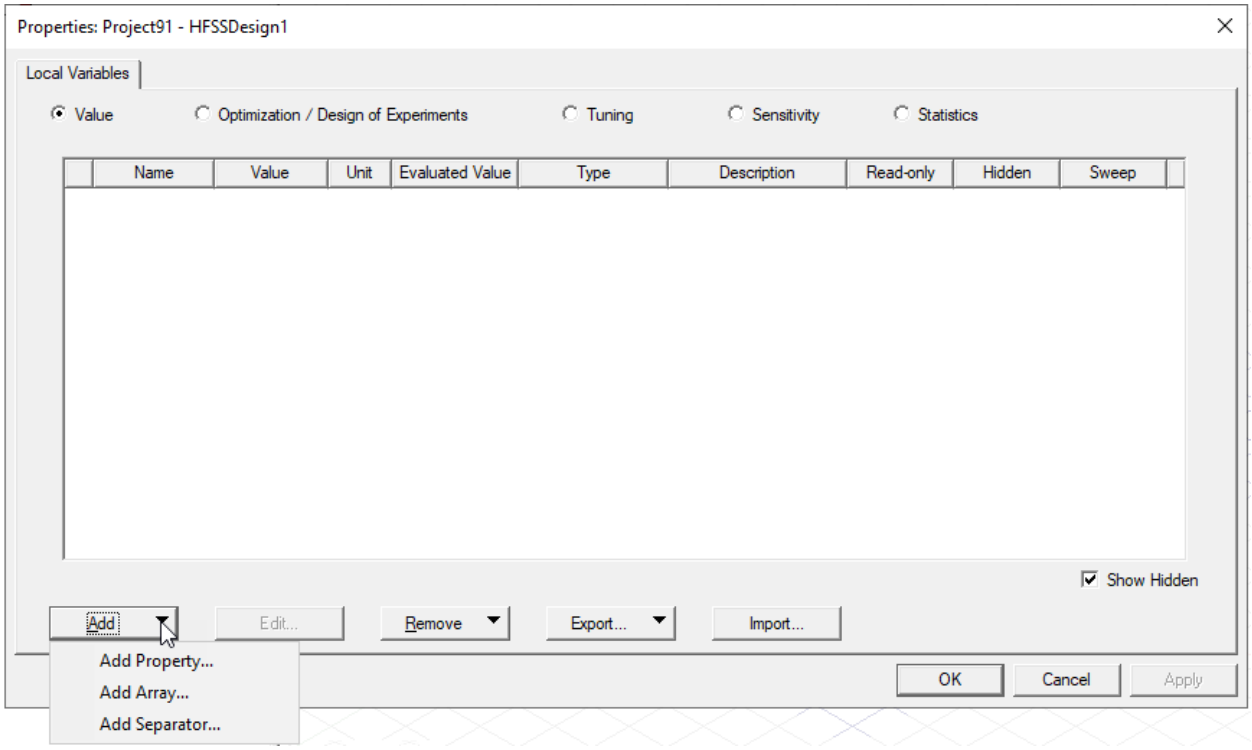
To define a Local Variable:

1. Access **Design Properties** one of three ways:
   - Click **Maxwell 3D, Maxwell 2D, or RMxprt** > **Design Properties**.
   - Right-click the design name in the Project Tree and select **Design Properties**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- From a menu in the lower left corner of the following Optimization dialogs: **Parametric**, **Optimization**, **Sensitivity**, **Statistical**, **Design of Experiments**, and **Design Xplorer Setup**. Click **Edit Variables** and from the menu select **Edit Design Variables**

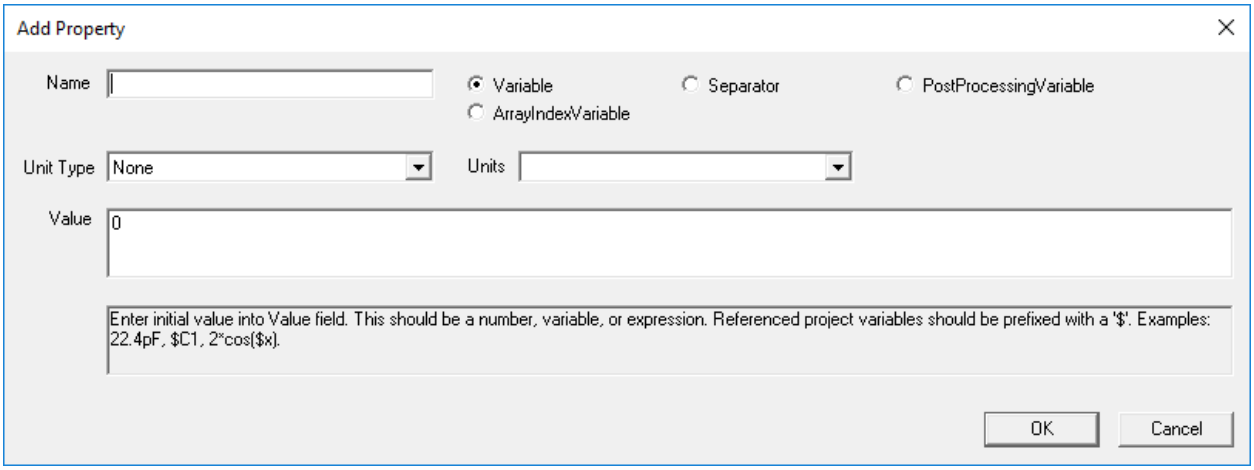
The **Properties** window appears with the **Local Variables** tab.



2. Click **Add**.

   The **Add Property** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Enter information for the variable, as applicable:
   - **Name** – the name must start with a letter and can contain alphanumeric characters and underscores (_). You cannot use any names already defined on the **General** tab. The names of intrinsic functions and the predefined constant pi (p) cannot be used as variable names.
   - **Variable Type** – use the **Variable**, **Separator**, **ArrayIndexVariable**, and **PostProcessingVariable** radio buttons to select the variable type.

     > **Note:**
     >
     > Not all variable types are available for all designs.

     Your selection impacts which properties you can edit:

Variable Type	Editable Properties
Variable	Unit Type, Units, Value.
Separator	Name. A separator variable provides a bolded name for a blank line to facilitate grouping variables in variable lists.
Array Index Variable	Associate Array Variable, Value
PostProcessingVariable	Unit Type, Units, Value.

   - **Unit Type** – for Variables and PostProcessingVariables, use the drop-down menu to select a type from the list (for example, Charge, Density, Energy, ...).

     When you select a Unit Type, the choices in drop down menu for the Units text box adapt to that unit type. For example, selecting Length as the Unit Type causes the Unit menu to show a range of metric and english units for length. Similarly, if you

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

select the Unit Type as Resistance, the Units drop down lists a range of standard Ohm units.

- **Units** – for Variables and PostProcessingVariables, use the drop-down menu to select a unit of measure.
- **Value** – for Variables, ArrayIndexVariables, and PostProcessingVariables, enter a number, variable, or mathematical expression. The quantity entered will be the current (or default) value for the variable. If the mathematical expression includes a reference to an existing variable, this variable is treated as a dependent variable. The units for a dependent variable will automatically change to those of the independent variable on which the value depends. Additionally, dependent variables, though useful in many situations, cannot be the direct subject of optimization, sensitivity analysis, tuning, or statistical analysis.

  For ArrayIndexVariables, the index reference can be a constant (for example, 1), an index (for example, ii) or an expression (for example, ii + 1). This allows you to sweep the index and simulate for different values that are stored in the array variable itself. In particular, it also enables you to sweep different text strings. This allows you to set a property to different string values as the index is swept.

  > **Warning:**
  >
  > If you include the variable's units in its definition (in the **Value** text box), do not include the variable's units when you enter the variable name for a parameter value.

- **Associate Array Variable** – for ArrayIndexVariables, select an associated array from the drop-down list. You must have previously created an array.

4. Click **OK**.

   The new variable appears in the list. You can sort local variables by clicking the Name column header. Clicking once sorts them in ascending order, noted by a triangle pointing up. Clicking against sorts in descending order, noted by a triangle pointing down. Clicking a third time sorts in original order, with no triangle.

5. If desired, use the check boxes to designate a variable as **Read-only**, **Hidden**, or **Sweep**.
   - **Read-only** – when selected, the variable's name, value, unit, and description cannot be modified.
   - **Hidden** – hidden variables do not appear in the **Properties** window unless **Show Hidden** is selected.
   - **Sweep** – allows you to designate variables to include in solution indexing as a way to permit faster post-processing. Variables with the Sweep check box cleared are not used in solution indexing. If a solution exists, selecting or clearing a variable's

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

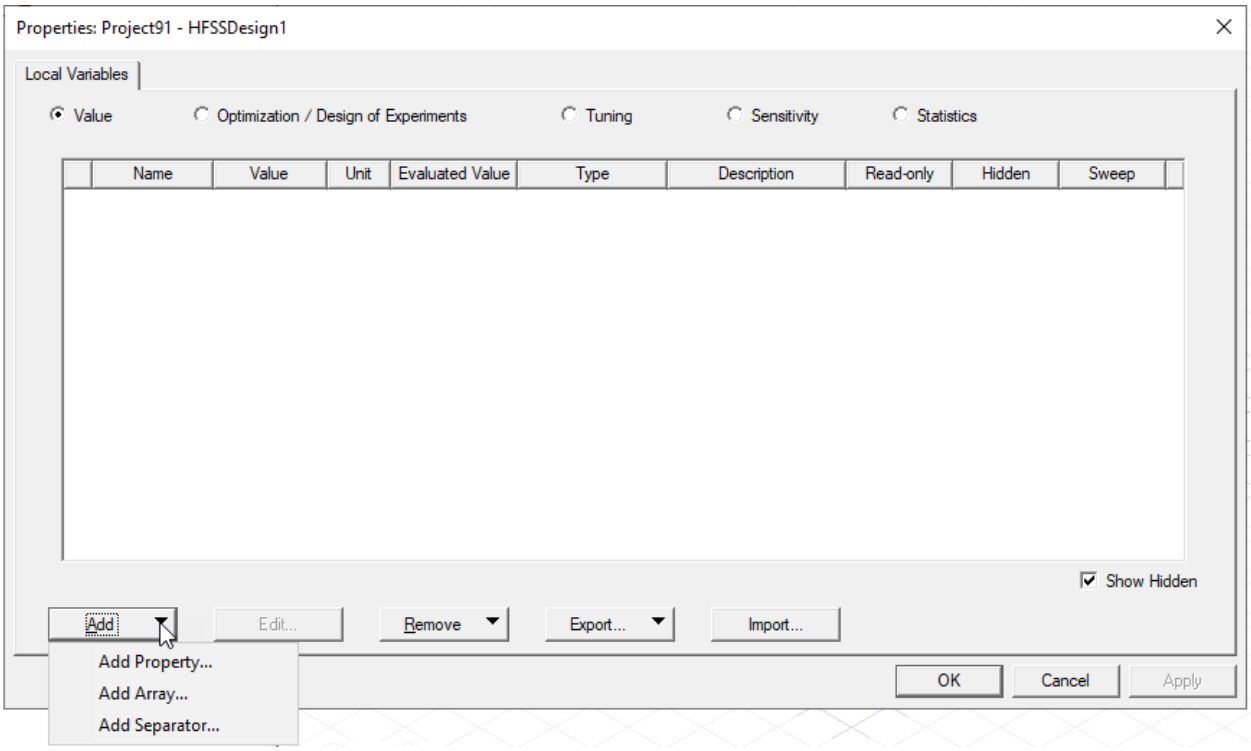
Sweep setting produces a warning that the change will invalidate existing solutions. To continue, click OK to dismiss the warning.

6. Click **Apply** to apply changes.
7. Click **OK** to exit the window.

The new variable can be assigned to a parameter value in the project in which it was created. You can enable defined variables for Optimization/Design of Experiments, Tuning, Sensitivity, or Statistics. See: Optimetrics.

Importing and Exporting Variables

Once you have defined Design Variables, you can export them to a csv file, and import them to another project or design. If there are naming conflicts on import, these are flagged as errors. The Properties dialogs for Design Variables include **Import...** and **Export...** buttons.



Deleting a Local Variable

To remove a Local Variable:

1. Access **Local Variables** as described above.
2. Remove one or more variables:
   - To remove a specific variable, select it and click **Remove** > **Remove Selected**.
   - To remove all unused variables, click **Remove** > **Remove All Unused**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To force the removal of all unused variables, including those in the project's undo/redo history, click **Remove** > **Force Remove All Unused**.

Editing a Local Variable

To edit a Local Variable:

1. Access **Local Variables** as described above.
2. Select the variable you want to edit and click **Edit**.
3. Change the properties as desired and click **OK**.

Overriding a Local Variable

You can override a variable value from an analysis setup's **General** tab.

Select the **Override** check box next to the value you wish to override, and enter a new value in the **Value** field.

Design Variable	Override	Value	Units
padrad	☐	0.6	mm
viarad	☐	0.2	mm

Defining Parameter Defaults at the Design Level

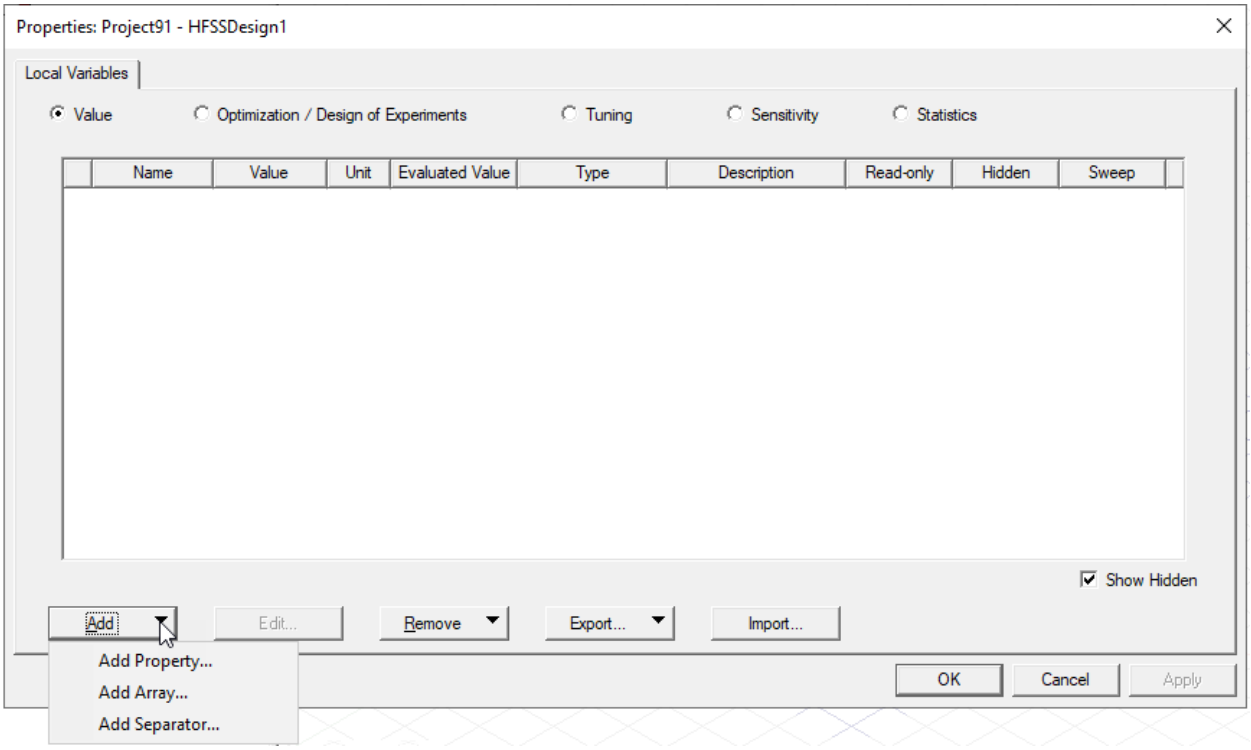
Adding a Parameter Default

To define a new parameter default at the design level:

1. Access **Design Properties** one of three ways:
   - Click **Maxwell 3D, Maxwell 2D, or RMxprt** > **Design Properties**.
   - Right-click the design name in the Project Tree and select **Design Properties**.
   - From a menu in the lower left corner of the following Optimization dialogs: **Parametric**, **Optimization**, **Sensitivity**, **Statistical**, **Design of Experiments**, and **Design Xplorer Setup**. Click **Edit Variables** and from the menu select **Edit Design Variables.**

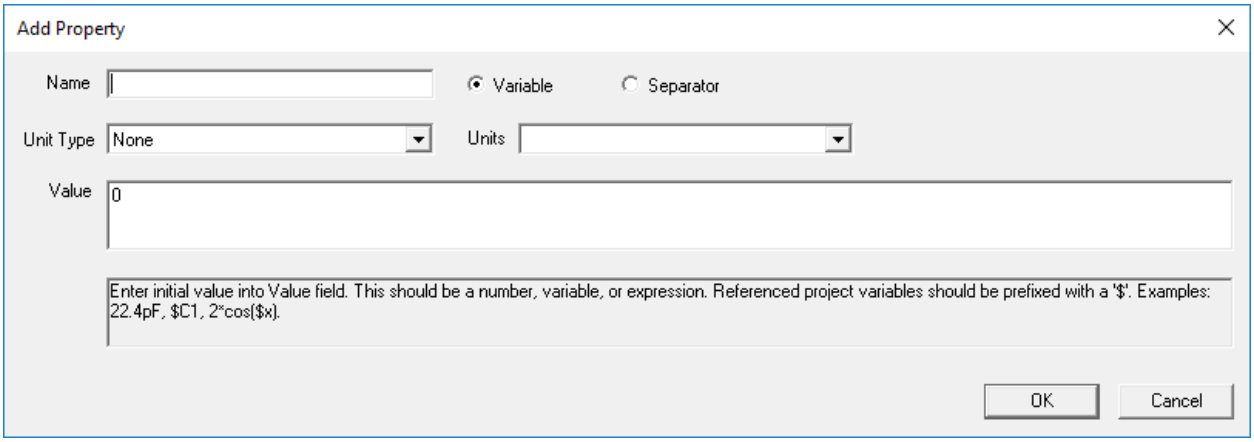
   The **Properties** window for design variables appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click **Add...**.

   The **Add Property** window appears.



3. Enter information for the parameter default, as applicable:
   - **Name** – the name can contain alphanumeric characters and underscores (_). You cannot use any names already defined on the **General** tab.
   - **Variable Type** – select **Variable** to enter a value, or select **Separator** to provide a bolded name for a blank line.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Your selection impacts which properties you can edit:

Variable Type	Editable Properties
Variable	Unit Type, Units, Value.
Separator	Name. A separator variable provides a bolded name for a blank line to facilitate grouping variables in variable lists.

- **Unit Type** – for Variables, use the drop-down menu to select a type from the list (for example, Charge, Density, Energy, ...).
- **Units** – for Variables, use the drop-down menu to select a unit of measure.
- **Value** – for Variables, enter a valid numeric quantity. Alternately, you can assign a variable to a parameter by typing a variable name or mathematical expression in place of a parameter value.

4. Click **OK**.

   The new variable appears in the list. You can sort local variables by clicking the Name column header. Clicking once sorts them in ascending order, noted by a triangle pointing up. Clicking against sorts in descending order, noted by a triangle pointing down. Clicking a third time sorts in original order, with no triangle.

5. If desired, use the check boxes to designate a variable as **Read-only**, **Hidden**, or **Sweep**.
   - **Read-only** – when selected, the variable's name, value, unit, and description cannot be modified.
   - **Hidden** – hidden variables do not appear in the **Properties** window unless **Show Hidden** is selected.
6. Click **Apply** to apply changes.
7. Click **OK** to exit the window.

Deleting a Parameter Default

To remove a Parameter Default:

1. Access **Parameter Defaults** as described above.
2. Select a parameter from the list and click **Remove**.

Editing a Parameter Default

To edit a Parameter Default:

1. Access **Parameter Defaults** as described above.
2. Select the parameter you want to edit and click **Edit**.
3. Change the properties as desired and click **OK**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Overriding a Parameter Default

You can override a parameter default from an analysis setup's **General** tab.

Select the **Override** check box next to the value you wish to override, and enter a new value in the **Value** field.

Design Variable	Override	Value	Units
padrad	☐	0.6	mm
viarad	☐	0.2	mm

Adding a Design Variable Array

You can define array variables that contain numbers or strings. Number array variables can be used in component property expressions, while string array variables can be used in certain component property values. Design variable arrays are only available within the Maxwell design for which they are defined.

> **Note:**
>
> Text array variables can be used in certain component property values, but not all, and cannot be combined with operators to form more complex expressions.
>
> The index for a text array variable reference can either be a constant (1) or can be an index (ii) or even an expression (ii + 1). This allows you to sweep the index and simulate for different values that are stored in the array variable itself. In particular, it also enables you to sweep different text strings. This allows you to set a property to different string values as the index is swept. The following are properties that currently allow text array variables:
>
> - V_PRBS
> - V_PRBS_JITTER
> - V_PRBSD
> - V_PRBSG_JITTER
> - V_PSK
> - V_QAM
> - V_CPM

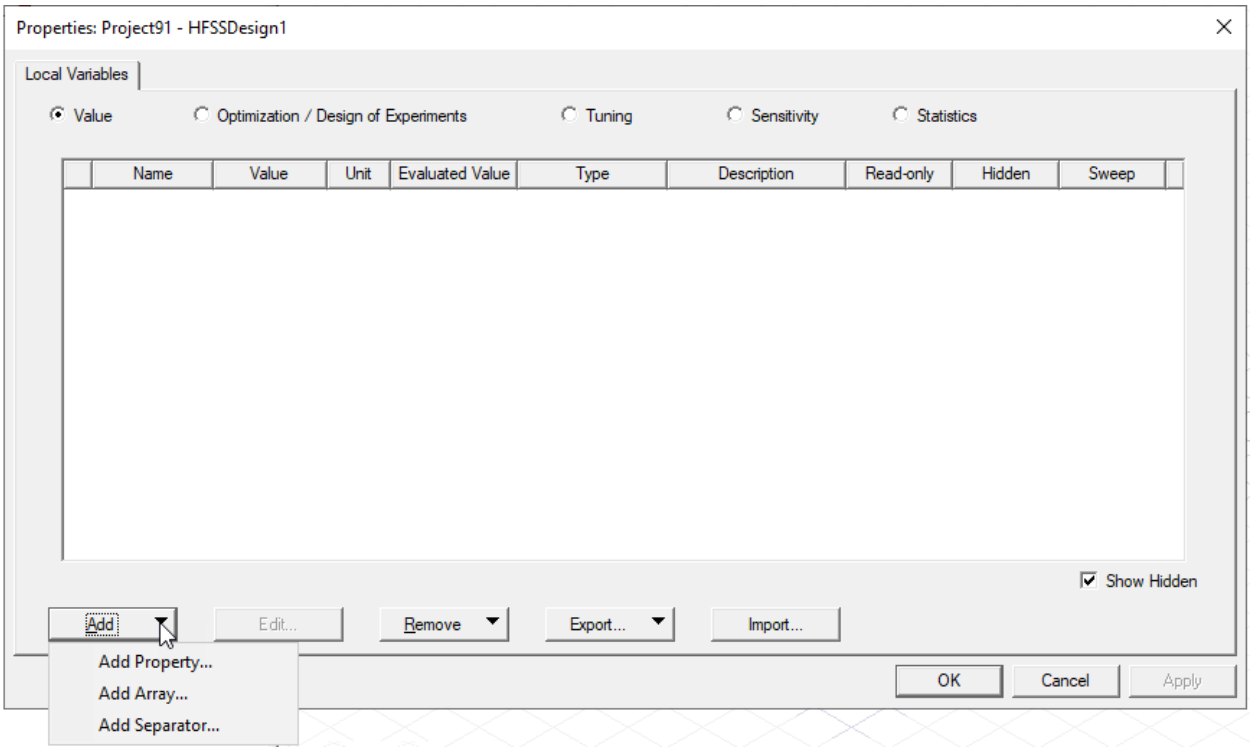
To define an array variable, you must first define a Local Variable Array:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Access **Local Variables** one of three ways:
   - Click **Maxwell 3D, Maxwell 2D, or RMxprt** > **Design Properties**.
   - Right-click the design name in the Project Tree and select **Design Properties**.
   - From a menu in the lower left corner of the following Optimization dialogs: **Parametric**, **Optimization**, **Sensitivity**, **Statistical**, **Design of Experiments**, and **Design Xplorer Setup**. Click **Edit Variables** and from the menu select **Edit Design Variables**

   The **Properties** window appears.
2. Select the **Local Variables** tab.

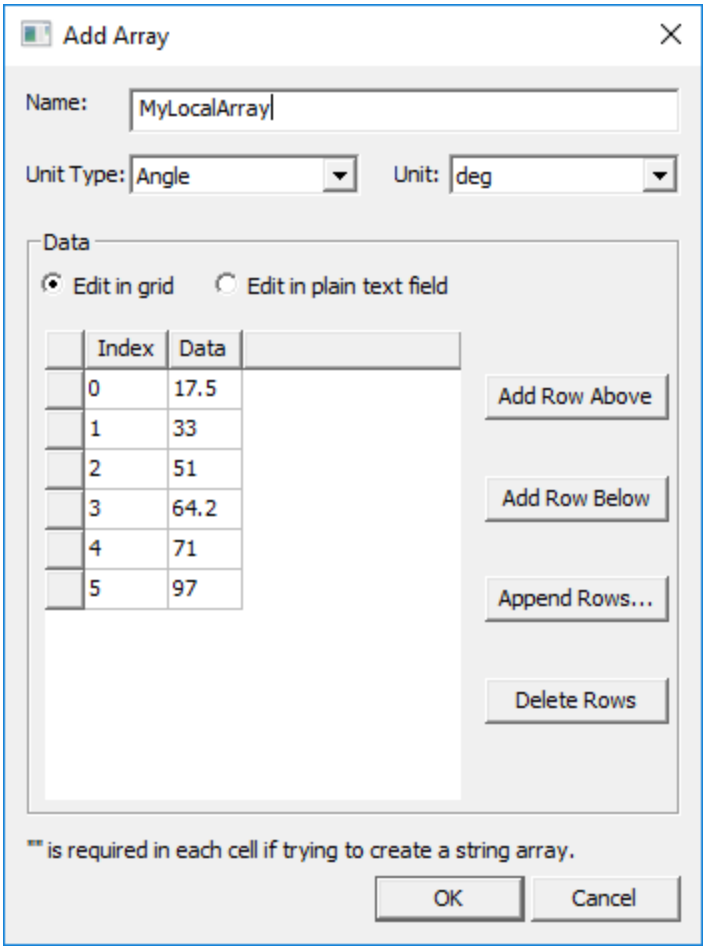
   The **Local Variables** tab displays.



3. Click **Add Array**.

   The **Add Array** window appears.

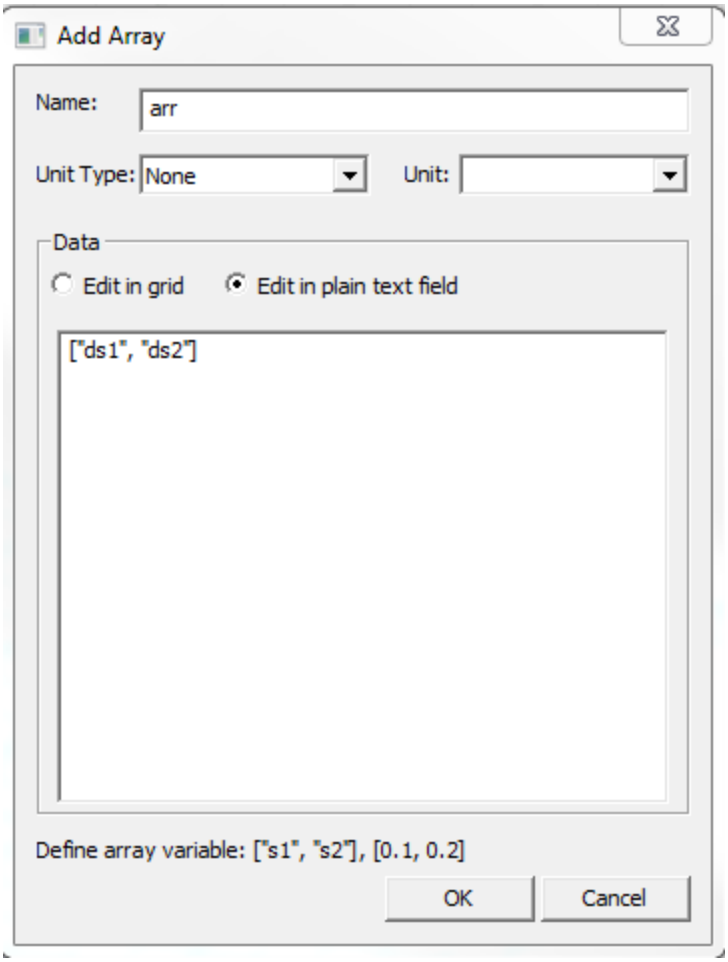
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The value grid displays each array item's **Index** number, and the **Data** associated with that index number.

If you elected to **Edit in plain text field** in the Add Array dialog box, the bracketed and comma delimited format is used.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Give the variable array a **Name**, and select a **Unit Type** and **Unit** from the drop-down menus.
5. Use the control buttons to add, delete, and reposition rows in the value grid at left. The default is **Edit in grid**, but you may select **Edit in plain text field**.

   > **Note:**
   >
   > Quotation marks ("") are *required* as delimiters when array values are entered in either the grid or text field.

6. When you have finished entering values, click **OK**.

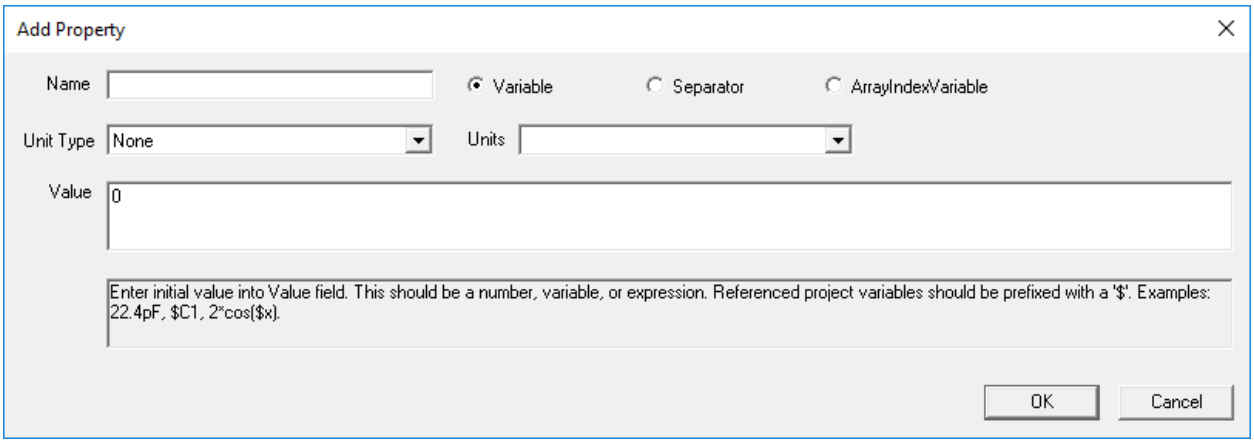
   The array appears in the **Local Variables** list.

You can add a variable to the array by adding an ArrayIndexVariable on the **Local Variables** tab.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Access **Local Variables** one of three ways:
   - Click **Maxwell 3D, Maxwell 2D, or RMxprt** > **Design Properties**.
   - Right-click the design name in the Project Tree and select **Design Properties**.
   - From the **Setup Optimization** dialog box, click **Edit Variables** > **Edit Design Variables**.
2. From the **Local Variables** tab, click **Add**.

   The **Add Property** dialog box appears.



3. Enter information for the variable, as applicable:
   - **Name** – the name can contain alphanumeric characters and underscores (_). You cannot use the names of Intrinsic Variables or pre-defined Constants.
   - **Variable Type** – select ArrayIndexVariable.
   - **Value** – enter a number, variable, or mathematical expression. The quantity entered will be the current (or default) value for the variable. If the mathematical expression includes a reference to an existing variable, this variable is treated as a dependent variable. The units for a dependent variable will automatically change to those of the independent variable on which the value depends. Additionally, dependent variables, though useful in many situations, cannot be the direct subject of optimization, sensitivity analysis, tuning, or statistical analysis.

     For ArrayIndexVariables, the index reference can be a constant (for example, 1), an index (for example, ii) or an expression (for example, ii + 1). This allows you to sweep the index and simulate for different values that are stored in the array variable itself. In particular, it also enables you to sweep different text strings. This allows you to set a property to different string values as the index is swept.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Warning:**
>
> If you include the variable's units in its definition (in the **Value** text box), do not include the variable's units when you enter the variable name for a parameter value.

- **Associate Array Variable** – for ArrayIndexVariables, select an associated array from the drop-down list. You must have previously created an array.

4. Click **OK**.

Converting Variables and Parameter Defaults

Design (Local) variables can be converted to parameter defaults, and vice versa.

To convert a parameter default or local variable:

1. Access **Design Properties** one of three ways:
   - Click **Maxwell 3D, Maxwell 2D, or RMxprt** > **Design Properties**.
   - Right-click the design name in the Project Tree and select **Design Properties**.
   - From the **Setup Optimization** dialog box, click **Edit Variables** > **Edit Design Variables**.

   The **Properties** window appears.
2. Select either the **Parameter Defaults** tab or the **Local Variables** tab, depending on which item you want to convert.
3. Select the variable/parameter and click **Convert to Parameter** or **Convert to Variable**.

   The item will move to the list on the appropriate tab.

Some special cases to consider:

- If a local variable references another local variable, it cannot be converted to a parameter default. You will see a pop-up message if you attempt to convert. This is because local variables can reference parameter defaults, but parameter defaults cannot reference local variables.
- Parameter defaults can be overridden and can have different values in each subcircuit, while a Local Variable cannot. If a parameter default has been overridden and you convert it to a variable, it will lose all overrides. Attempting to do so will display a pop-up message asking if you really want to do the conversion and lose the override values.

Adding Variables and Parameter Defaults from Components

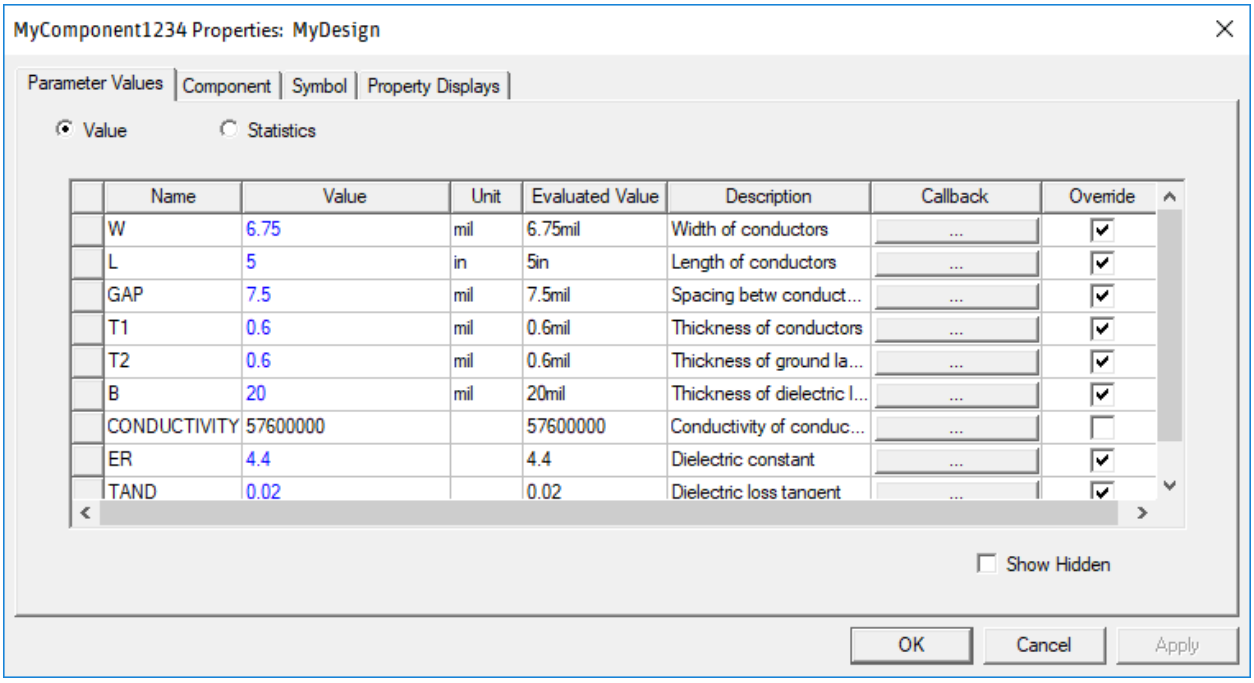
To define a new variable or parameter default from a component:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Display a component's **Properties** window one of three ways:
   - Double-click the component.
   - Select the component and click **Edit** > **Properties**.
   - Select the component, right-click and select **Properties**.

   The **Properties** window appears.
2. Select the **Parameter Values** tab.

   The **Parameter Values** tab displays.



3. Select the **Value** radio button.
4. Select the **Value** field for the parameter you want to set equal to a local variable.
5. Type the variable name. The name may not be the same as an intrinsic variable or constant.

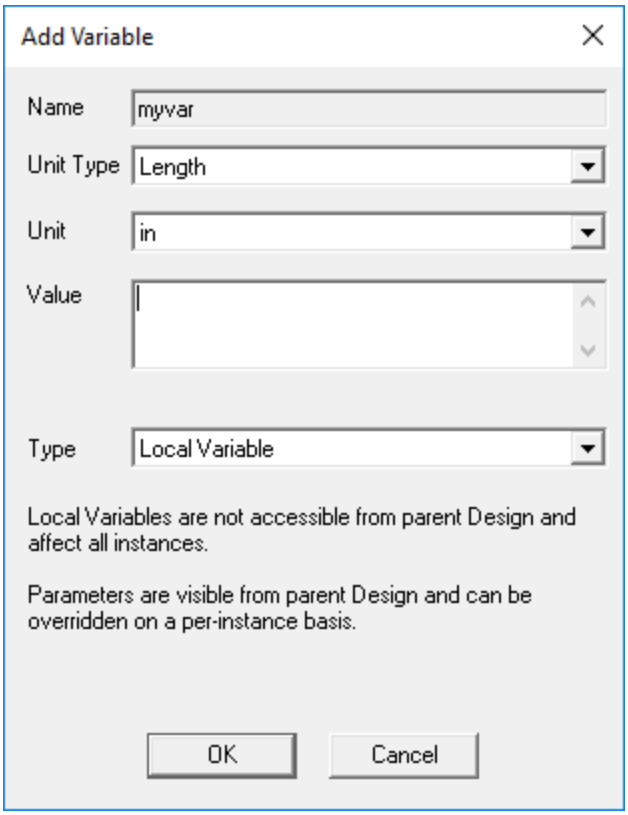
   Variable names may include alphanumeric characters and underscores (_).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Important:**
>
> If you wish to define a Project Variable, you must begin the name with a dollar sign ($).
>
> When you enter a name beginning with $, you will only be allowed to define it as a Project Variable.

6. Press **Enter**.

   The **Add Variable** window appears.



7. Select a **Unit Type** and **Unit** using the drop-down menus.
8. Enter a **Value**.

   The Value can be a number, variable, or mathematical expression. The quantity entered will be the current (or default) value for the variable. If the mathematical expression includes a reference to an existing variable, this variable is treated as a dependent variable. The units for a dependent variable will automatically change to those of the independent variable on which the value depends. Additionally, dependent variables,

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

though useful in many situations, cannot be the direct subject of optimization, sensitivity analysis, tuning, or statistical analysis.

> **Warning:**
>
> If you include the variable's units in its definition in the **Value** text box, do not include the variable's units when you enter the variable name for a parameter value.

9. From the **Type** drop-down menu, select **Local Variable** or **Parameter Default**.

> **Note:**
>
> If you wish to define a Project Variable, your variable name must begin with a dollar sign ($).

10. Click **OK** to exit the **Add Variable** window.
11. Click **Apply** to save your changes.
12. Click **OK** to exit the **Properties** window.

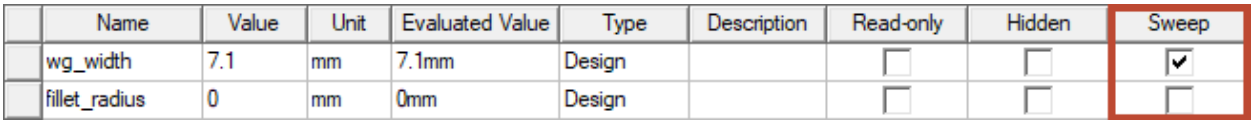
Fixed vs. Non-Fixed Variables

By default, both Project variables and Design variables are included in the list of variables that index solution data. As a result, they are visible when selecting data to be displayed in plots.

However, if you define a variable as *fixed*, it is not swept and will not index solution data. Consequently, defining variables as fixed can speed up simulation—particularly if there are many variables to simulate.

To define a variable as fixed:

1. Open the **Properties** window for either Project or Design variables.
2. Clear the **Sweep** check box next to the variable you want to be fixed.



	Name	Value	Unit	Evaluated Value	Type	Description	Read-only	Hidden	Sweep
	wg_width	7.1	mm	7.1mm	Design		☐	☐	☑
	fillet_radius	0	mm	0mm	Design		☐	☐	☐

- If a **Sweep** check box is selected (default) the corresponding variable is NOT fixed.
- If a **Sweep** check box is cleared, the variable is fixed.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

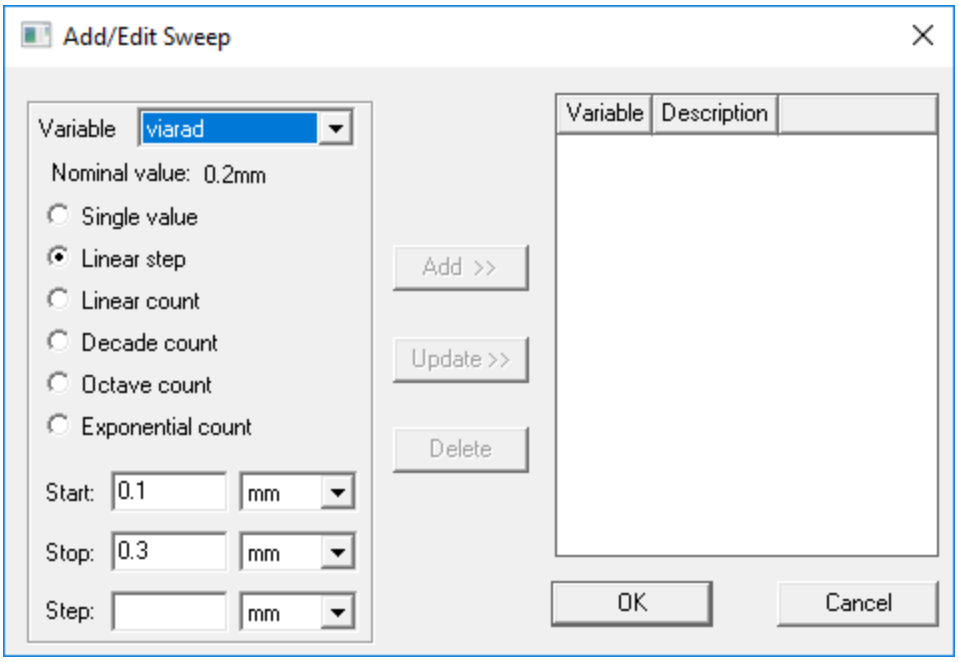
Non-Fixed Variable Sweeps

When setting up a sweep, the **Setup** window contains a list of variables being swept.

To add a non-fixed variable sweep:

1. Click **Add**.

   The **Add/Edit Sweep** window appears.



2. Use the **Variable** drop-down menu to select a non-fixed variable.
3. Select the desired settings and click **Add**.
4. Click **OK** to close the **Add/Edit Sweep** window.
5. Click **OK** to exit the **Setup**.

Orphaned Sweeps

If you create a simulation setup that contains a sweep of a variable, and then subsequently clear the variable's **Sweep** check box, the variable becomes fixed and, as a result, its sweep is "orphaned." The sweep is removed from the setup and the simulation runs as if the sweep did not exist.

- When you close the **Properties** window, a warning message is added to the **Message Window** for each orphaned sweep.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you edit an orphaned variable in the **Properties** window and reselect its **Sweep** box, the orphaned sweep will be restored to the setup list.
- If you double-click a setup that contains orphaned sweeps, a pop-up dialog box asks if you want to delete the orphaned sweeps. If you respond **Yes**, all orphaned sweeps are deleted when the **Properties** window is closed. Even if you reselect a **Sweep** check box, its orphaned sweep will NOT be restored. This action is undoable.
- Orphaned sweeps are not written to disk when the project is saved, so once you save a project and close it, any orphaned sweeps are permanently lost.

Choosing a Variable to Optimize

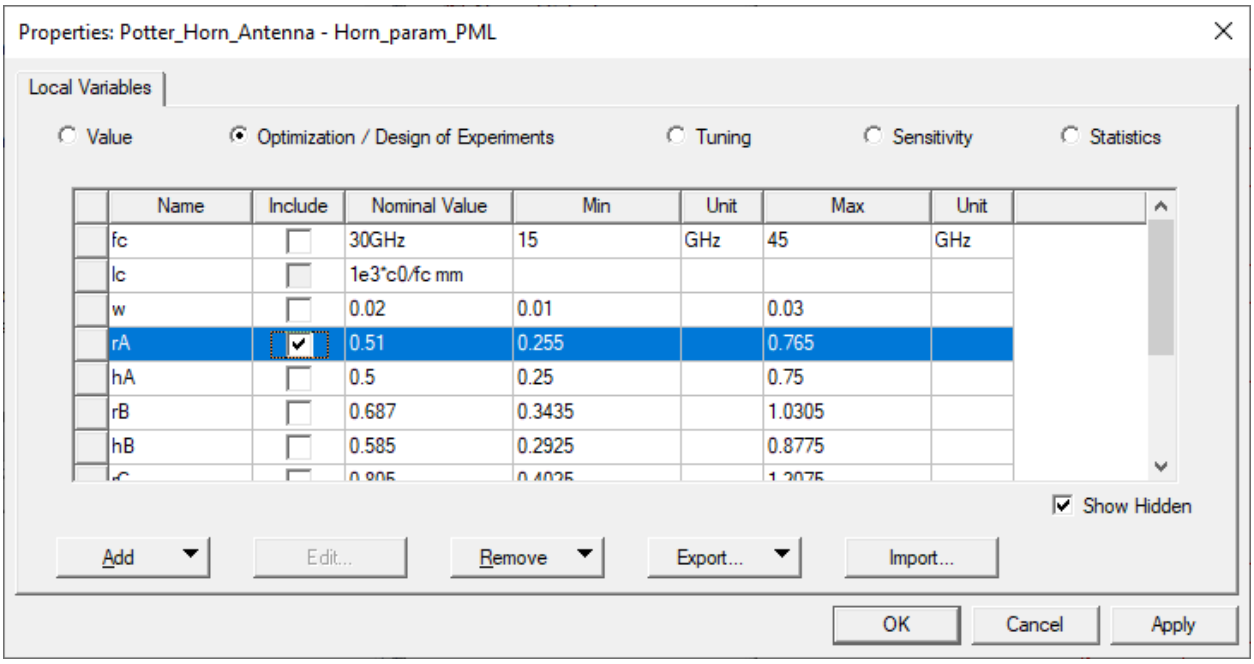
Before a variable can be optimized, you must specify that you intend for it to be used during an optimization analysis in the **Properties** window.

To specify that a variable is to be used during optimization:

1. Navigate either to the Project Variables **Properties** window (**Project** > **Project Variables**), or to the Design Variables **Properties** window (**Maxwell 3D**, **Maxwell 2D**, or **RMxprt** > **Design Properties**).

   The **Properties** window appears.

2. Use the tabs to navigate to the variable you want to optimize.
3. Select the **Optimization / Design of Experiments** radio button.
4. Use the **Include** check boxes to select variables for optimization.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Important:**
>
> Dependent variables cannot be optimized.
>
> Variables containing complex numbers cannot be used in an Optimetrics sweep or for optimization, statistical, sensitivity, and tuning setups.

The selected variable(s) will be available for optimization in an Optimetrics setup defined in the current design or project.

5. If desired, use the **Min** and **Max** fields to override the default minimum and maximum values that Optimetrics will use for the variable in every optimization analysis. During optimization, the optimizer will not consider variable values that lie outside of this range.

Choosing a Variable to Tune

Before a variable can be tuned, you must specify that you intend for it to be tuned in the **Properties** window.

To specify that a variable is to be tuned:

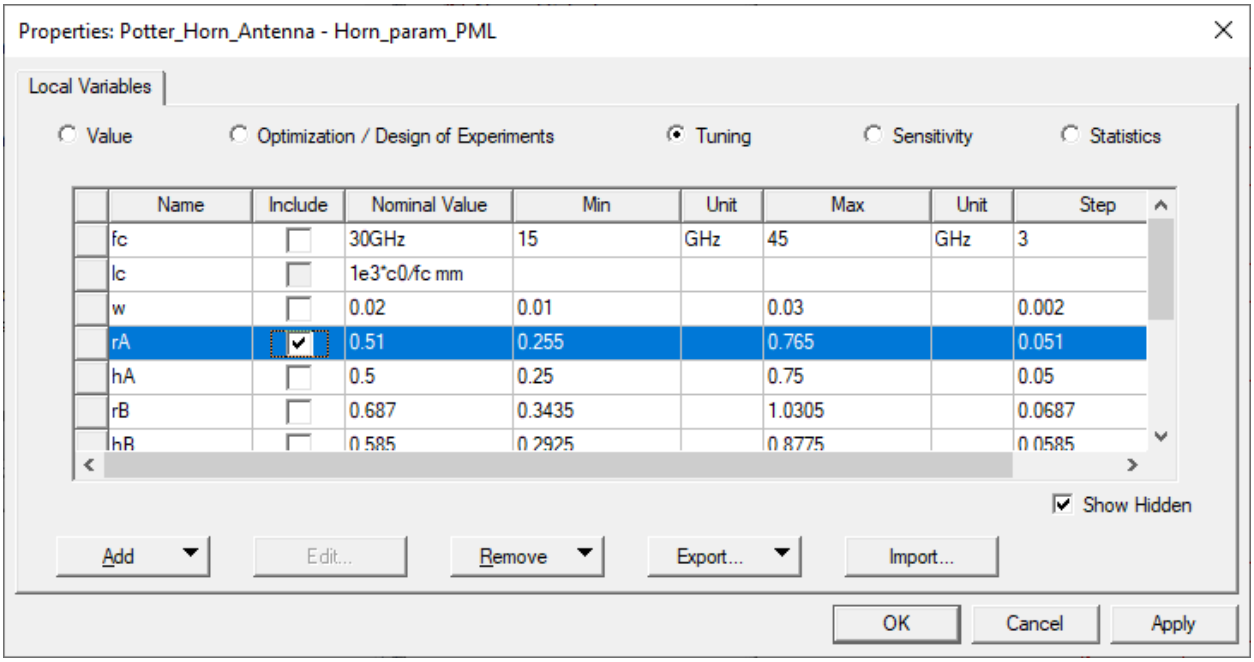
1. Navigate either to the Project Variables **Properties** window (**Project** > **Project Variables**), or to the Design Variables **Properties** window (**Maxwell 3D**, **Maxwell 2D**, or **RMxprt** > **Design Properties**).

   The **Properties** window appears.

2. Use the tabs to navigate to the variable you want to optimize.
3. Select the **Tuning** radio button.
4. Use the **Include** check boxes to select variables for tuning.

> **Important:**
>
> Dependent variables cannot be tuned.
>
> Variables containing complex numbers cannot be used in an Optimetrics sweep or for optimization, statistical, sensitivity, and tuning setups.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The selected variable(s) will be available for tuning.

Including a Variable in a Sensitivity Analysis

Before a variable can be included in a sensitivity analysis, you must specify that you intend for it to be used during a sensitivity analysis in the **Properties** window.

To specify that a variable is to be used during sensitivity analysis:

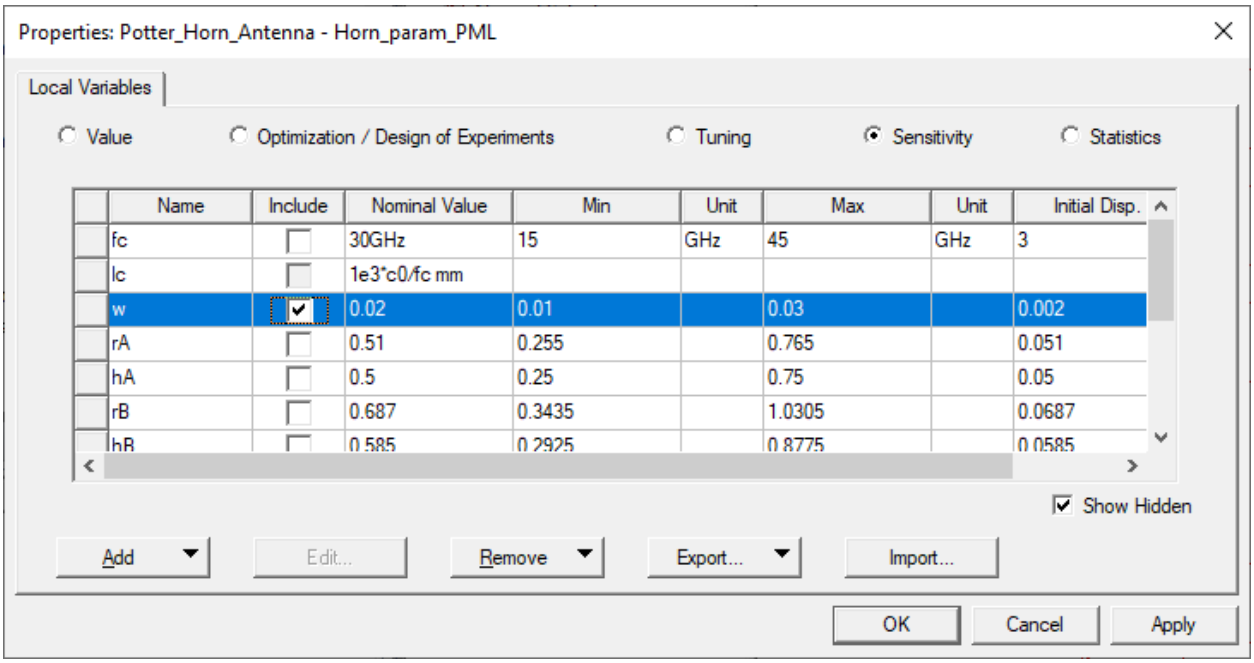
1. Navigate either to the Project Variables **Properties** window (**Project** > **Project Variables**), or to the Design Variables **Properties** window (**Maxwell 3D**, **Maxwell 2D**, or **RMxprt** > **Design Properties**).

   The **Properties** window appears.

2. Use the tabs to navigate to the variable you want to use.
3. Select the **Sensitivity** radio button.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Use the **Include** check boxes to select variables for sensitivity analysis.



> **Important:**
>
> Dependent variables cannot be included in a sensitivity analysis.
>
> Variables containing complex numbers cannot be used in an Optimetrics sweep or for optimization, statistical, sensitivity, and tuning setups.

The selected variable(s) will be available for sensitivity analysis in the current design or project.

5. If desired, use the **Min** and **Max** fields to override the default minimum and maximum values that Optimetrics will use for the variable in every sensitivity analysis. During analysis, the optimizer will not consider variable values that lie outside of this range.
6. If desired, use the **Initial Disp.** field to override the default initial displacement value that Optimetrics will use for the variable in every sensitivity analysis. During analysis, Optimetrics will not consider a variable value for the first design variation that is greater than this step size away from the starting variable value.

Including a Variable in a Statistical Analysis

Before a variable can be included in a statistical analysis, you must specify that you intend for it to be used during a statistical analysis in the **Properties** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

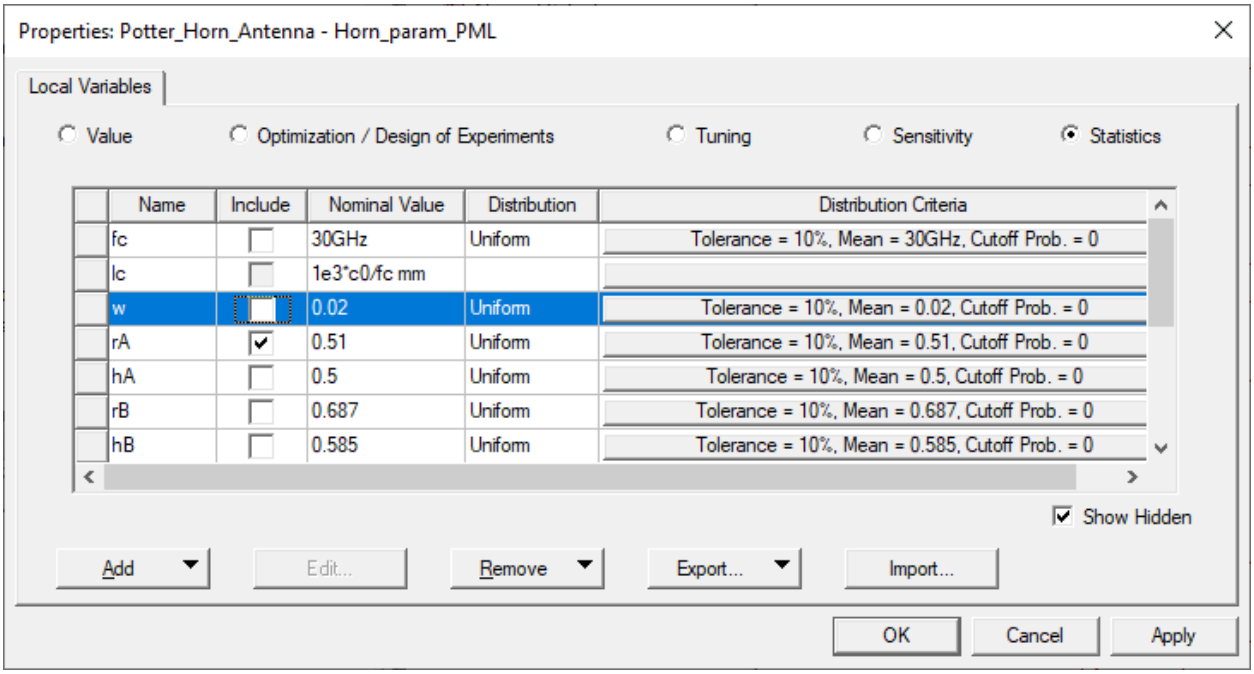
To specify that a variable is to be used during statistical analysis:

1. Navigate either to the Project Variables **Properties** window (**Project** > **Project Variables**), or to the Design Variables **Properties** window (**Maxwell 3D**, **Maxwell 2D**, or **RMxprt** > **Design Properties**).

   The **Properties** window appears.

2. Use the tabs to navigate to the variable you want to use.
3. Select the **Statistics** radio button.
4. Use the **Include** check boxes to select variables for statistical analysis.

> **Important:**
>
> Dependent variables cannot be included in a statistical analysis.
>
> Variables containing complex numbers cannot be used in an Optimetrics sweep or for optimization, statistical, sensitivity, and tuning setups.



   The selected variable(s) will be available for statistical analysis in the current design or project.

5. If desired, click the **Distribution Criteria** button to override the distribution criteria that Optimetrics will use for the variable in every statistical analysis.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copying and Pasting a Variables List

From either the Project or Design variables window, you can copy a list of variables and their values to the clipboard. You can then paste them in a plain text, tab-separated format. This is useful for creating documentation.

To copy and paste the variables list:

1. Navigate to the Project Variables or Design Variables **Properties** window.
2. Right-click and select **Copy to Clipboard**.
3. Open a text editor and **paste**.

The pasted variables appear in a tab separated column format. Fields that do not contain values are left blank.

Name	Value	Unit	"Evaluated Value	"Description	Read-only	Hidden
$width	14.8570192	mm	14.8570192mm		false	false
$length	7.824547736	mm	7.824547736mm		false	false
$height	0.45*$width		6.68565864mm		false	false

Working with Datasets

Datasets are collections of plotted data points that can be extrapolated into an equation based on the piecewise linear makeup of the plot. Each plot consists of straight line segments whose vertices represent their end points. A curve is fitted to the segments of the plot and an expression is derived from the curve that best fits the segmented plot. The created expression is then used in piecewise linear Intrinsic Functions. You can add datasets at either the project level or the design level. They can be for various purposes, including to define frequency-dependent port impedances or frequency-dependent global variables.

> **Note:**
>
> The descriptions of the various controls and fields for working with datasets are also applicable to the BH Curve dialog box when Specifying BH Curves for Nonlinear Relative Permeability.

Project-level datasets are typically used for defining material properties at the project level (applicable to all designs in the project).

Design-level datasets can be used in geometry entities like part commands, coordinate systems, points, and planes. Design-level datasets do not work with equation-based surfaces or curves.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Design-level datasets can be used directly with piecewise linear functions in expressions or indirectly through variables that can refer to the dataset.

Design-level datasets can also be used in the following operations:

- **Creating or editing geometry** – when a geometry uses a dataset directly, edit dataset invalidates the solution. When a geometry uses a variable that is defined by dataset, editing the dataset does not invalidate the solution.
- **Creating an Animation** – based on a variable which can index in datasets.
- **Copying and Pasting Geometry** – if a part refers to a design dataset, it will be pasted to destination design.

Datasets are provided for Maxwell designs.

The **Datasets** dialog box provides a browsable listing of all datasets currently defined for the project or design. A preview window displays a plot of the currently selected dataset. Controls allow you to **Add**, **Edit**, **Remove**, and **Clone** datasets; to **Import** and **Export** characteristic data; and to launch the **SheetScan** tool to extract data from graphics such as data sheets.

To access the **Datasets** window:

- For project-level datasets, including 3D datasets, click **Project** > **Datasets**.
- For design-level datasets, click **Maxwell 3D**, **Maxwell 2D**, or **RMxprt** > **Design Datasets**.

Adding Datasets

The following procedure describes how to add a dataset manually.

You can also add datasets:

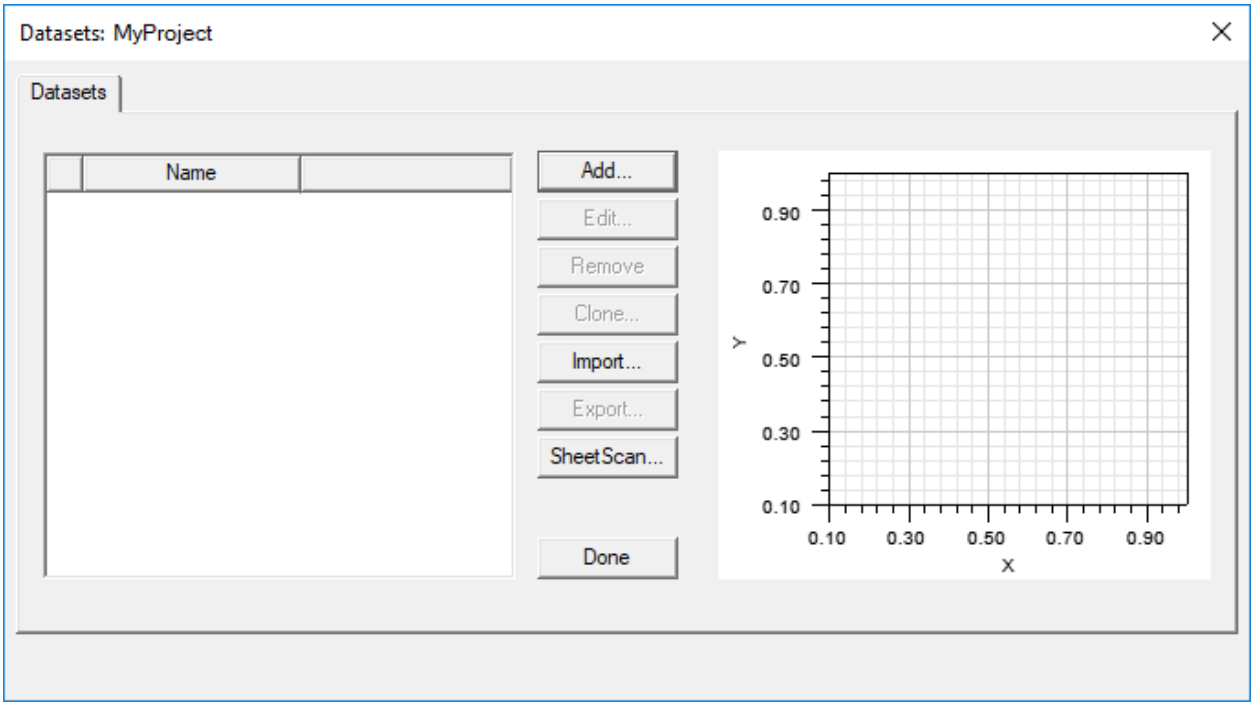
- By importing data from an external tab delimited file using the **Import...** button.
- Using the Sheetscan tool.

To add a dataset manually:

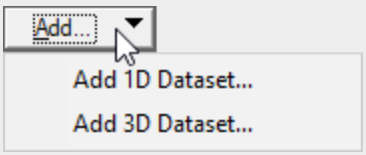
1. Access the **Datasets** window:
   - For project-level datasets, click **Project** > **Datasets** from the menu bar, or click **Datasets** on the **Desktop** ribbon tab.
   - For design-level datasets, click **Maxwell** > **Design Datasets** from the menu bar.

   The **Datasets** window appears, listing any project or design-level datasets.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

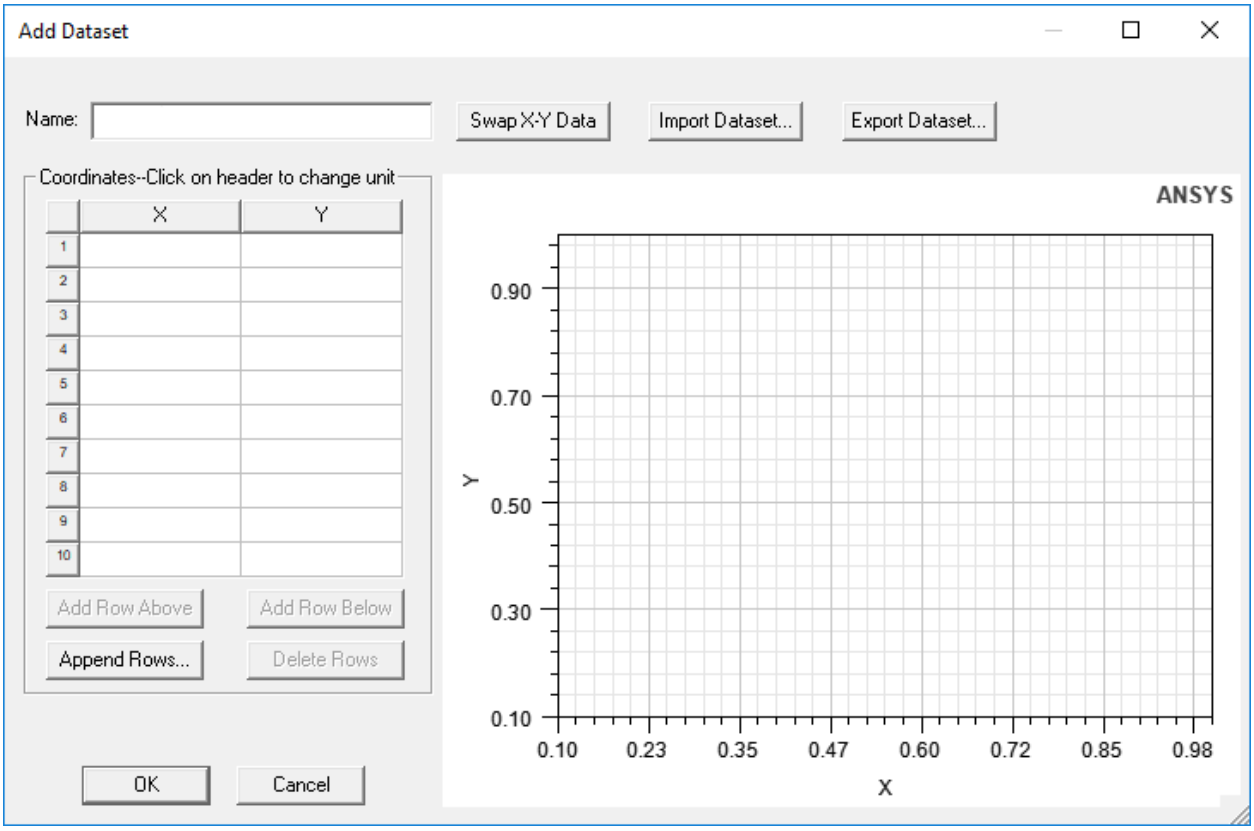


2. Click **Add**.

> **Note:**
>
> If you are adding a project-level dataset, and the project includes a Maxwell design, you will be presented the option to add either a **1D Dataset** or **3D Dataset**.
>
> Similarly, if you are adding a design-level dataset in a Maxwell design, you will be presented the same option.
>
> 
>
>
> See: Adding a 3D Dataset.

The **Add Dataset** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The left side of the **Add Dataset** window contains fields for the dataset name and a table for X- and Y- coordinates. The right side contains a graphic display that draws a line for the coordinates you add. It also includes buttons for the following functions:

- **Swap X-Y Data** – this swaps the X- and Y- coordinates and adjusts the graphical display.
- **Import Dataset** – this provides a way to import data sets from an external source. The format is a tab-separated points file. Clicking the button opens a file browser window.
- **Export Dataset** – this provides a way to export the current dataset to a tab separated points file. Clicking the button opens a file browser window.
- **Add Row Above** – adds a new row to the table above the selected row.
- **Add Row Below** – adds a new row to the table below the selected row.
- **Append Row** – opens a dialog box that lets you specify a number of rows to add to the table.
- **Delete Row** – deletes the selected row or rows.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

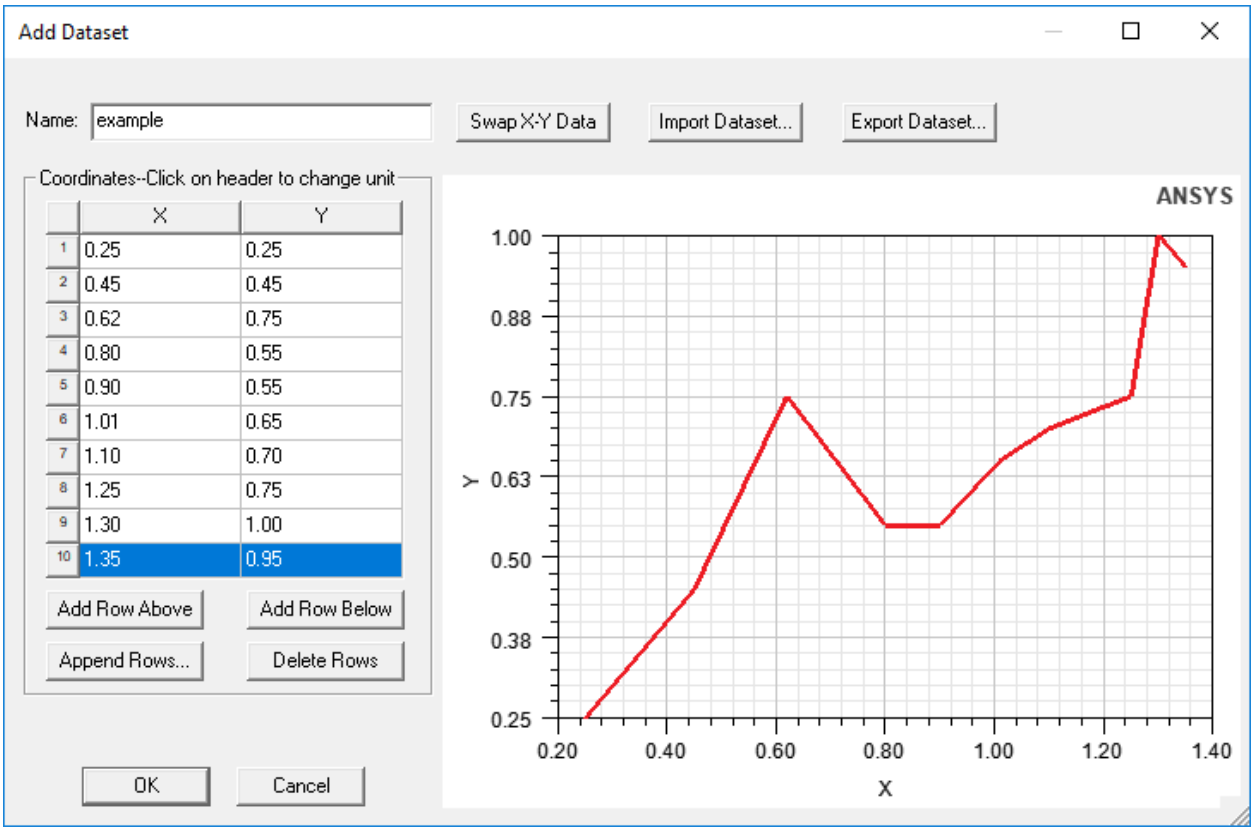
3. Enter a **Name** for the dataset.

> **Note:**
>
> Project-level datasets must begin with a dollar sign character ($). If you do not type one, it will be added for you.

4. Enter X/Y coordinates, either by manually typing or by clicking **Import Dataset**.

   If desired, click the **X** and **Y** column headers to change the unit.

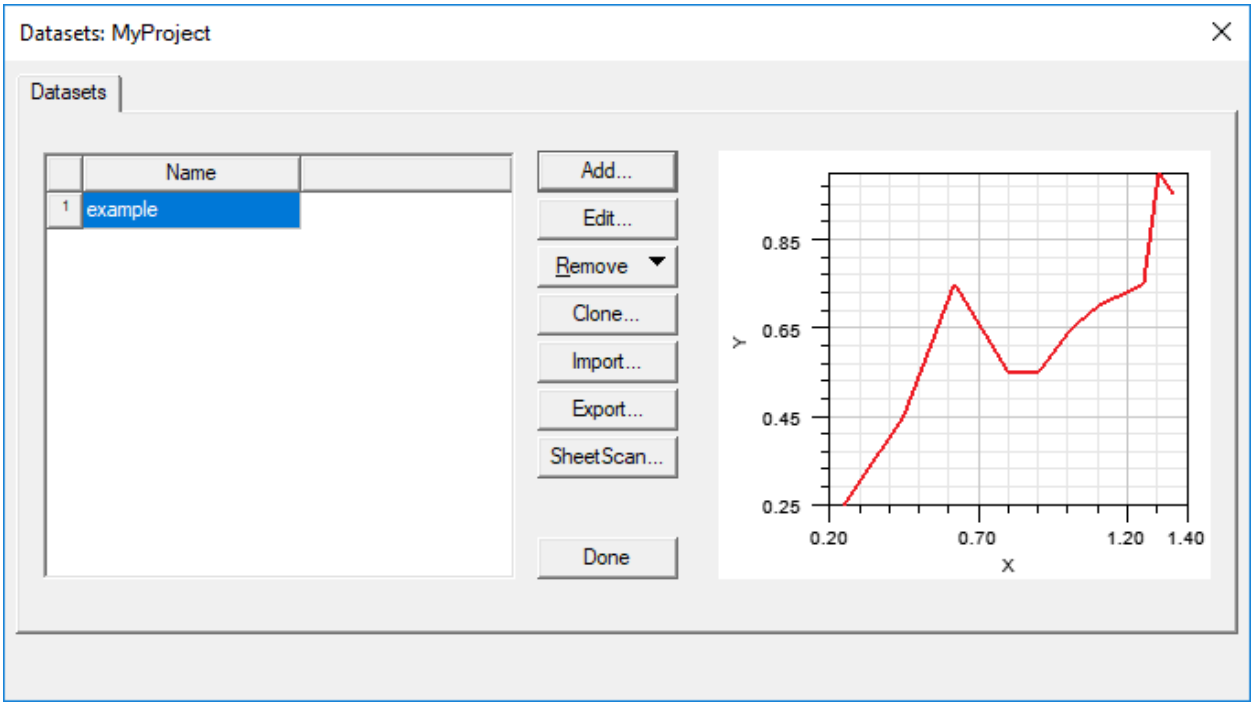
   The display updates to show your coordinates.



5. When you are finished, click **OK** to exit the **Add Datasets** window.

   The new dataset appears in the **Datasets** list.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The dataset plot is extrapolated into an expression that can be used in parametric analyses, boundary definitions, or assigned to a material property value.

You can now edit, remove, clone, or export the dataset. You can also change the plot display properties.

6. Click **Done** to exit the **Datasets** window.

Dataset Preview Plot Properties

You can change the properties of various elements of the preview plot such as the major and minor grid colors, title font, and trace line style.

1. Double-click the element to open its properties window.
2. Depending on the type of element selected, tabs allow you to set the **Color**, **Font**, **Line Style**, **Scaling**, **Title**, and **Legend** properties.
3. Right-click inside the plot window to open a context menu containing commands to add and delete data markers and labels, change trace type, and print the plot.
4. Click **Save as Defaults** to make the changes the default values.
5. Click **OK** to apply the changes only while the current dataset plot window is open. Click **Save as Defaults** to keep the changes.

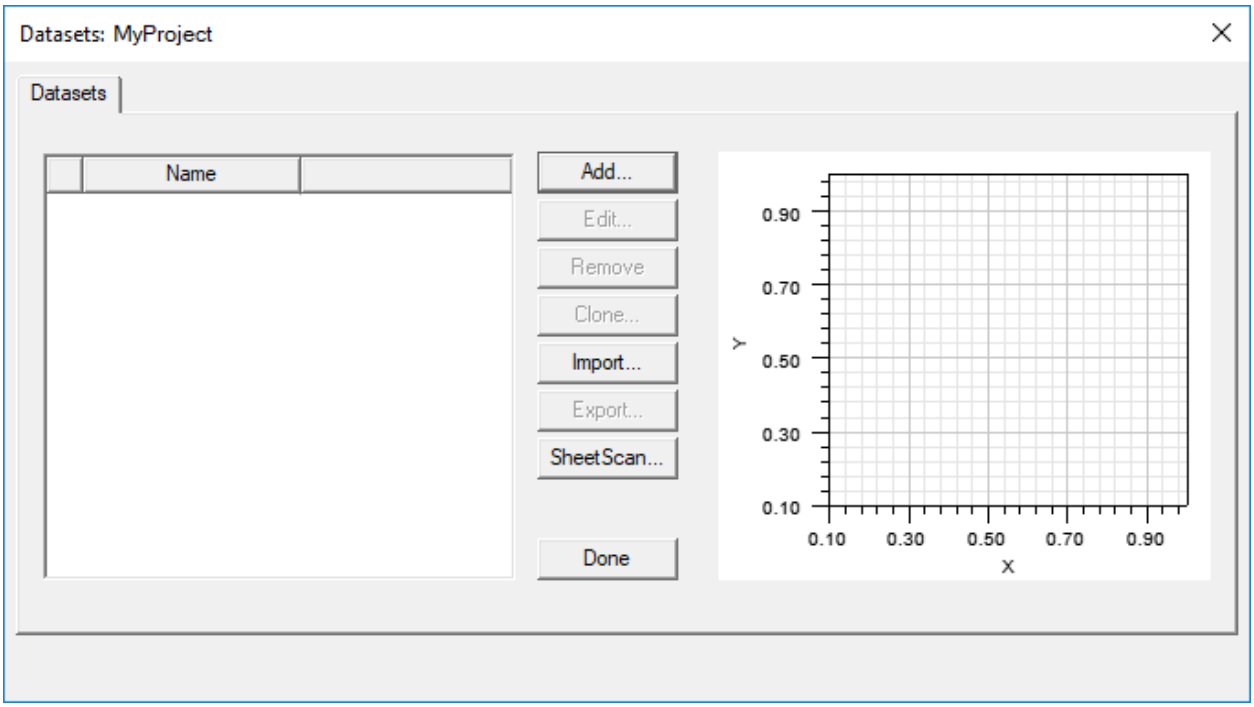
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Importing Datasets

To import a dataset from a file:

1. Access the **Datasets** window:
   - For project-level datasets, click **Project** > **Datasets**.
   - For design-level datasets, click **Maxwell 3D**, **Maxwell 2D**, or **RMxprt** > **Design Datasets**.

   The **Datasets** window appears.



2. Click **Import**.
3. In the file browser that appears, use the drop-down menu to select the file format of the file you wish to import.

> **Note:**
>
> The table below lists the file types supported for *Project level datasets*. For *Design level datasets*, only **.tab** files are supported.

.mdx, .mda	Twin Builder Characteristic format
.xls, .xlsx	Microsoft Excel

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

.txt	text file
.csv	Comma-separated value
.out	Maxwell SPICE (read-only - reads data inside the KW_ DATA section)
.cfg	Comtrade (IEEE Std C37-111-1999)
.dat	TEK Oscilloscope
.tab	tab delimited data files

a. If you select an **.mdx**, **.mda**, **.out**, **.cfg**, or **.tab** file, the data is imported immediately into **Datasets** window.

b. If you select an **.xls** or **.xlsx** file containing multiple sheets, a **Table Properties** dialog box appears, where you can choose the desired sheet from a drop-down menu. Otherwise, selecting an .xls or .xlsx file imports the data immediately.

> **Note:**
>
> Microsoft Excel is required to import .xls or .xlsx files.
>
> Only the first two columns of data are imported, the left-most column containing the X-coordinate values. The x-coordinate values for successive data points must increase within ten significant digits. Non-numeric entries are assigned a value of zero.
>
> The first row of data is assumed to contain column headings and is ignored.

c. Selecting a .**txt** or .**csv** file opens an **Import** dialog box where you can specify settings for reading the data in the file for import. You can choose the separator(s) and decimal symbol, as well as the line at which to begin the import. The dialog box shows both the original text and the text as it would appear when imported based on the current import settings.

4. When satisfied with the import settings, click **OK** to import the data.

   Imported datasets take the name of the imported file. If a dataset of the same name already exists, a number will be appended to the end of the new dataset (for example, a duplicate 'dataset' becomes 'dataset1').

   Once the data has been imported, you can edit, remove, clone, or export the dataset. You can also change plot display properties.

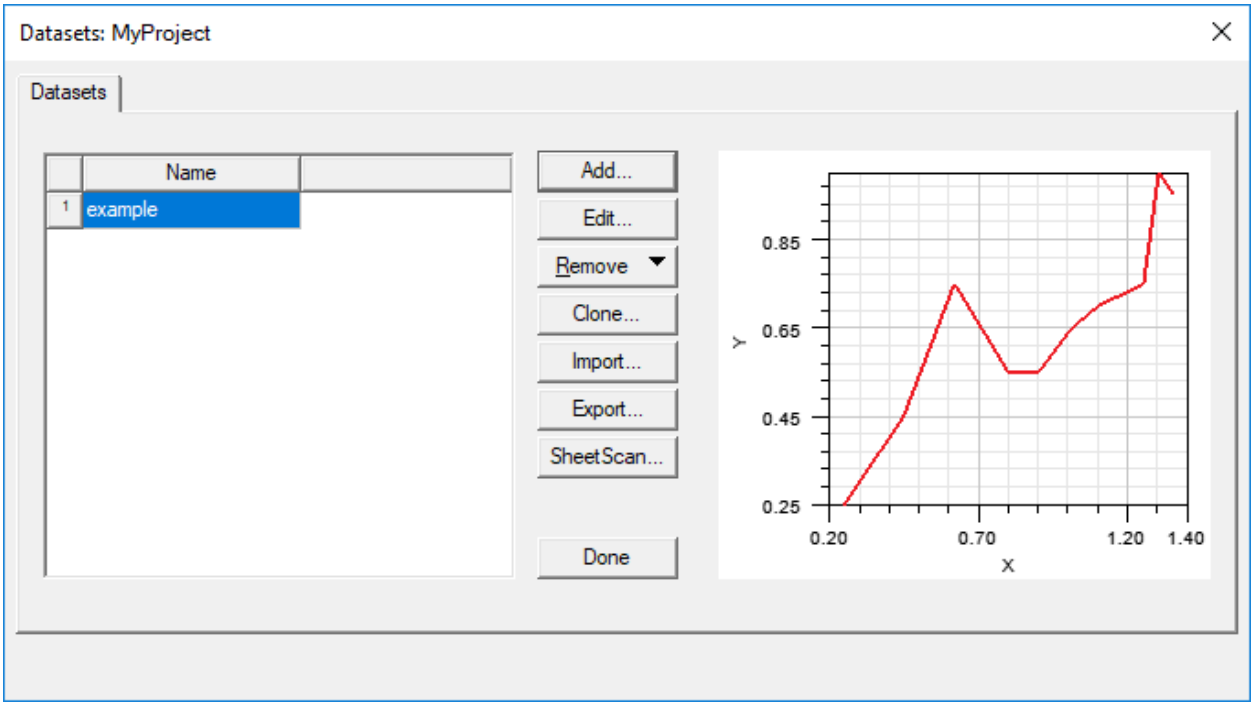
Editing Datasets

To modify an existing dataset:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Access the **Datasets** window:
   - For project-level datasets, click **Project** > **Datasets**.
   - For design-level datasets, click **Maxwell 3D**, **Maxwell 2D**, or **RMxprt** > **Design Datasets**.

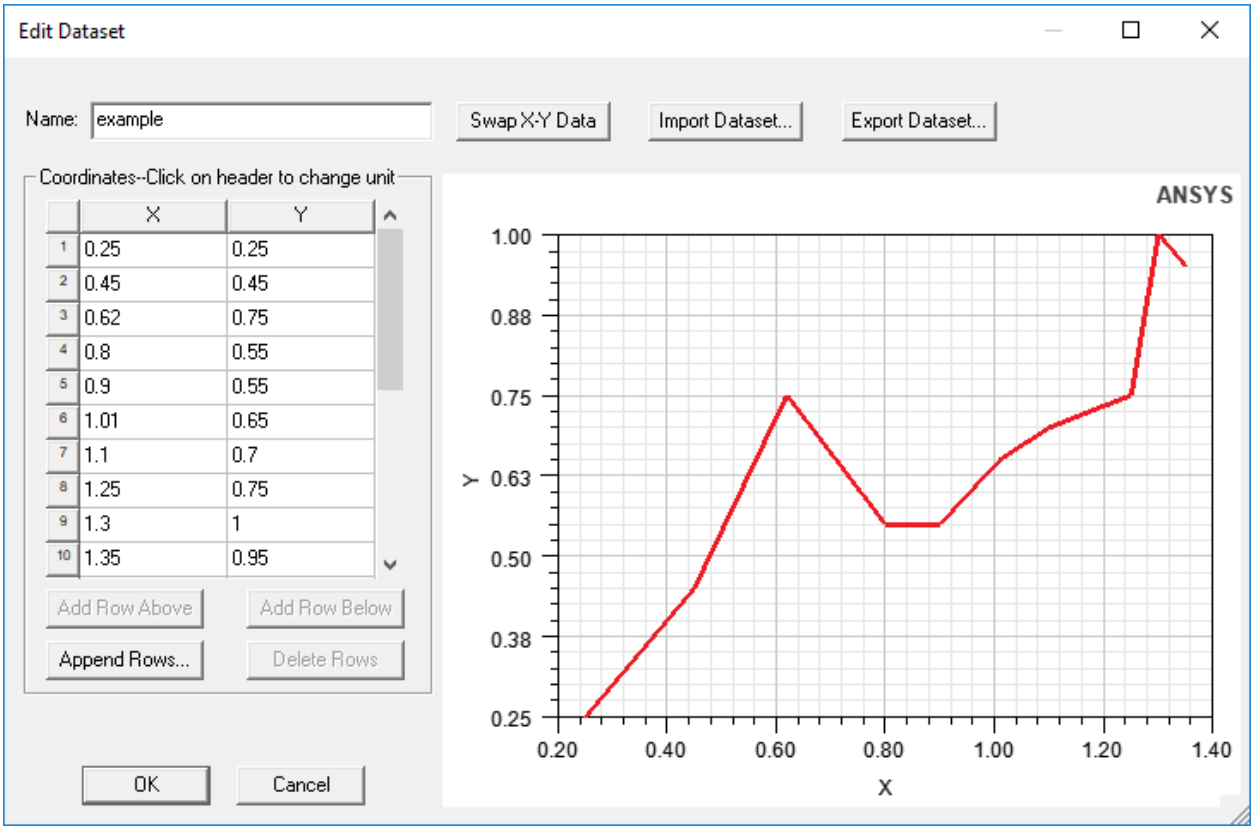
   The **Datasets** window appears, listing any project or design-level datasets.



2. Click **Edit**.

   The **Edit Dataset** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Change the **Name** and **X/Y** coordinates as desired.
4. When you are finished, click **OK** to exit the **Edit Dataset** window.
5. Click **Done** to exit the **Datasets** window.

Cloning Datasets

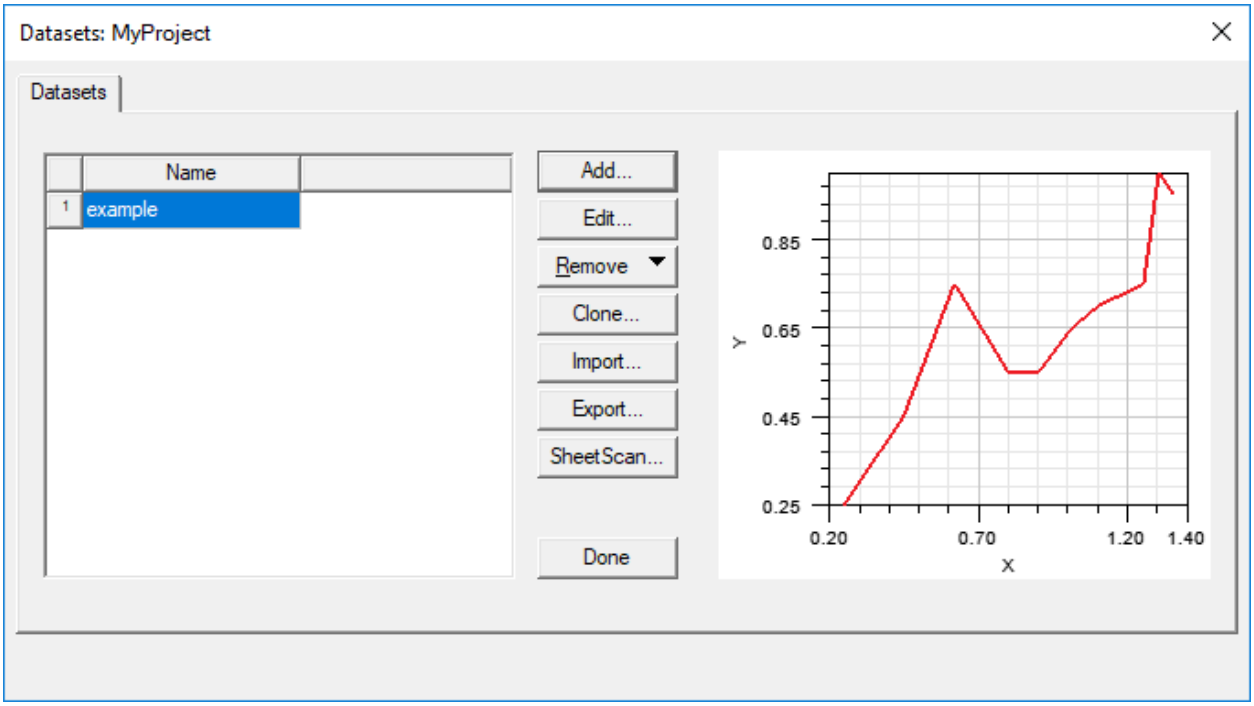
Cloning a dataset creates an identical copy of an existing dataset. The clone can then be modified as necessary.

To clone a dataset:

1. Access the **Datasets** window:
   - For project-level datasets, click **Project** > **Datasets**.
   - For design-level datasets, click **Maxwell 3D**, **Maxwell 2D**, or **RMxprt** > **Design Datasets**.

   The **Datasets** window appears, listing any project or design-level datasets.

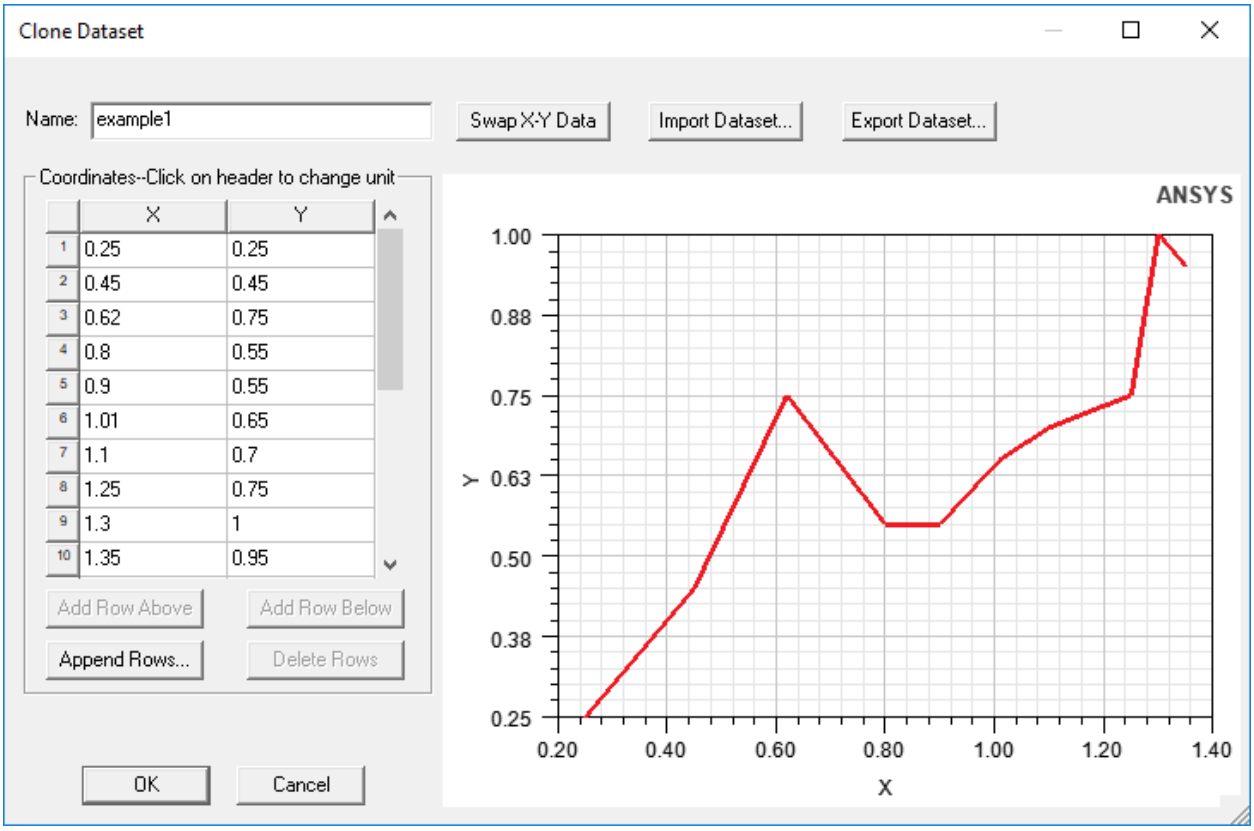
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click **Clone**.

   The **Clone Dataset** window appears. The default name is the original dataset name with a number appended to the end.

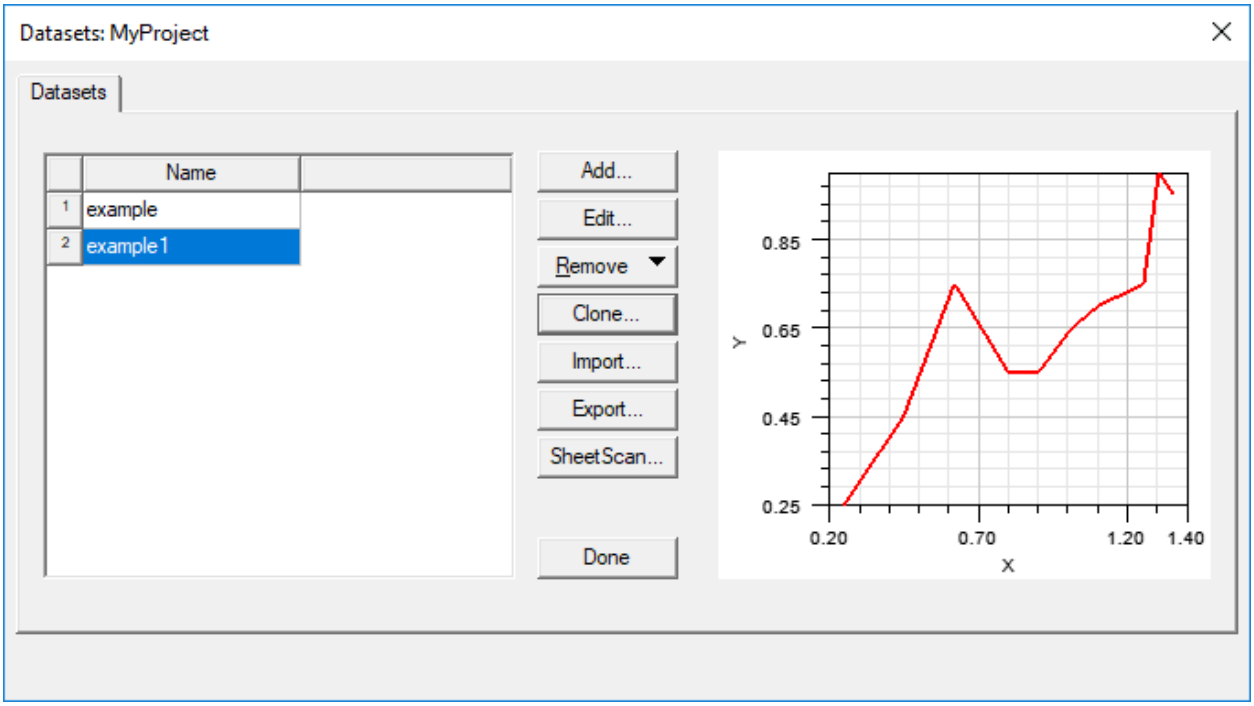
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Change the **Name** and **X/Y** coordinates as desired.
4. When you are finished, click **OK** to exit the **Clone Dataset** window.

   The cloned dataset appears in the list.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Click **Done** to exit the **Datasets** window.

Exporting Datasets

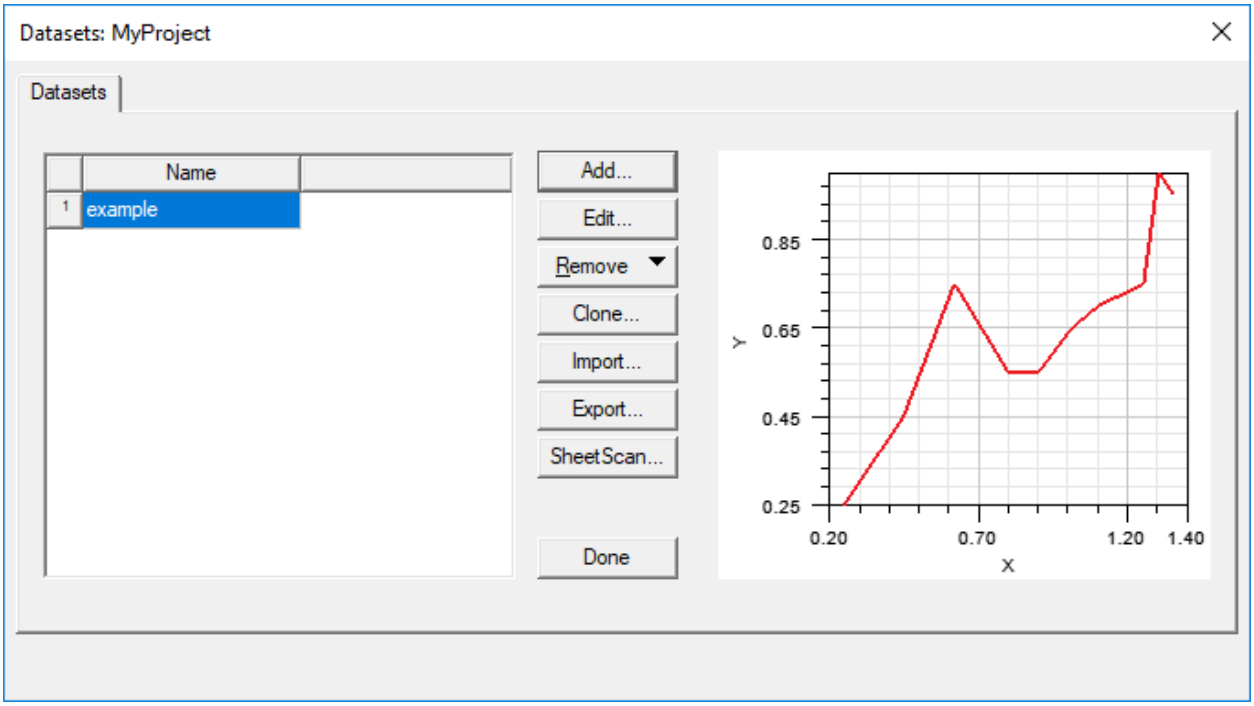
You can export datasets in **.tab** format.

To export a dataset:

1. Access the **Datasets** window:
   - For project-level datasets, click **Project** > **Datasets**.
   - For design-level datasets, click **Maxwell 3D**, **Maxwell 2D**, or **RMxprt** > **Design Datasets**.

   The **Datasets** window appears, listing any existing datasets.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select a dataset and click **Export**.
3. Browse to the location where you would like to save the *.tab file.
4. Enter a name for the file.
5. Click **Save**.

   The dataset file is saved to the specified location for later import.

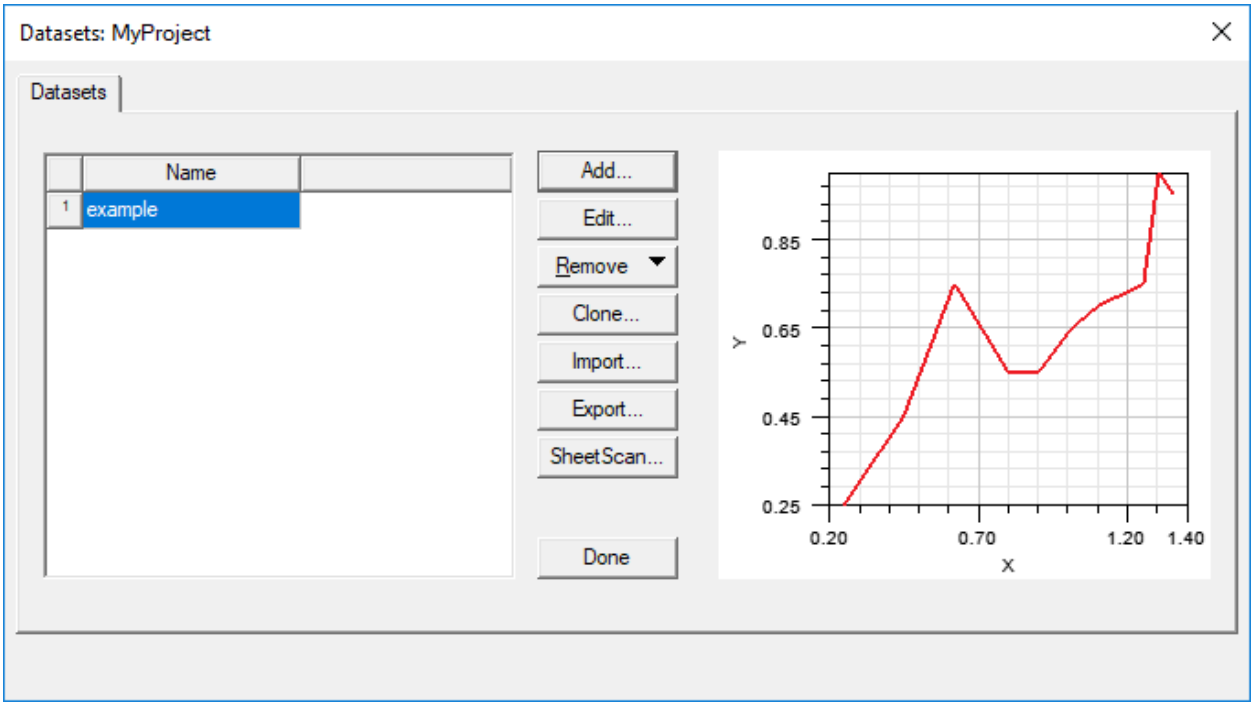
Removing Datasets

To remove a previously added dataset:

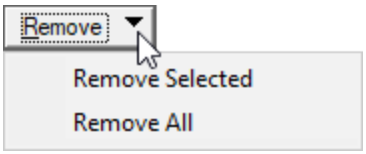
1. Access the **Datasets** window:
   - For project-level datasets, click **Project** > **Datasets**.
   - For design-level datasets, click **Maxwell 3D**, **Maxwell 2D**, or **RMxprt** > **Design Datasets**.

   The **Datasets** window appears, listing any existing datasets.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select a dataset and click **Remove**.
3. Select either **Remove Selected** to remove the selected dataset or **Remove All** to remove all project or design-level datasets.



The selected dataset(s) are deleted from the list.

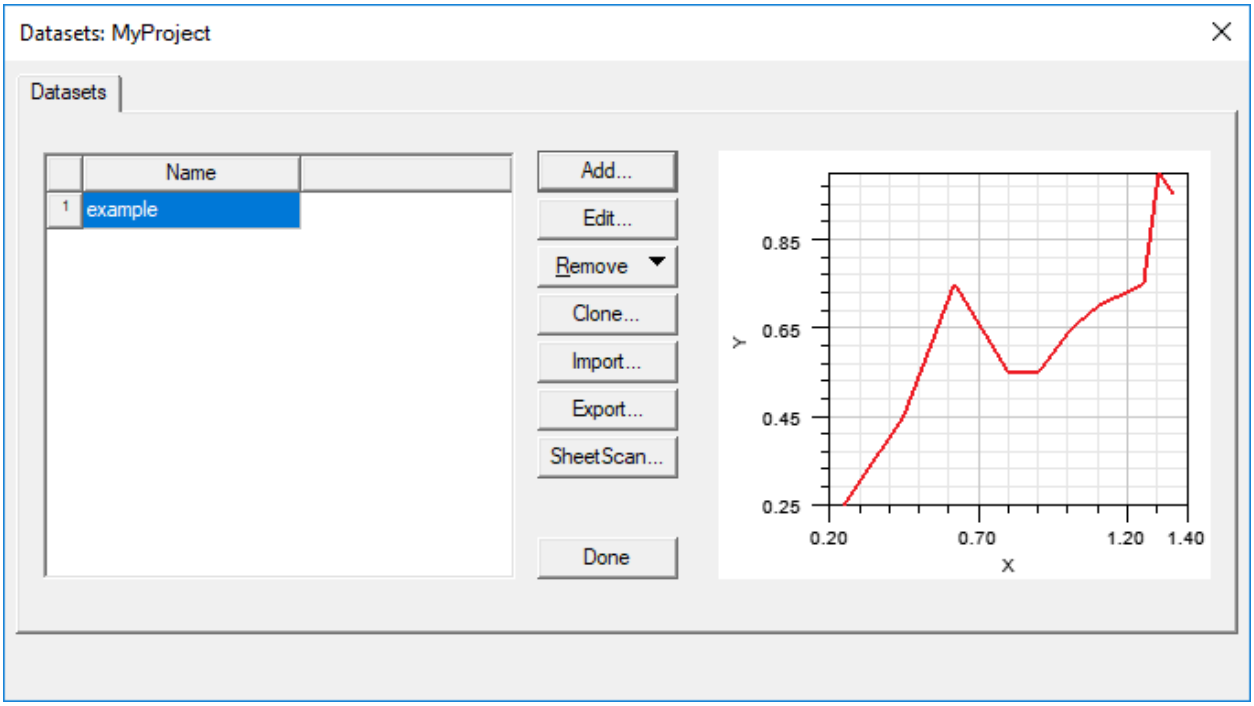
> **Note:**
>
> The **Remove All** option does not work across dataset types. That is, selecting **Remove All** from the **Project Datasets** window does not remove design-level datasets, and vice versa.

4. Click **Done** to exit the **Datasets** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Changing Dataset Plot Properties

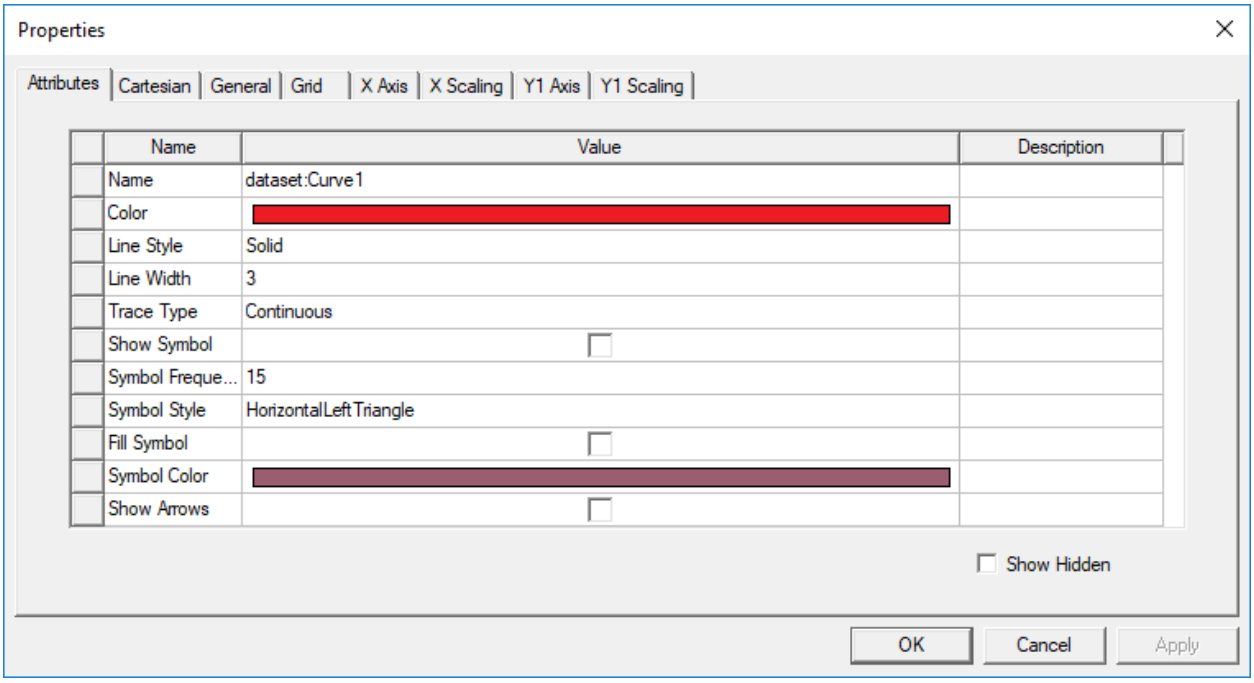
From the **Datasets** window, you can change the appearance of a dataset's plot.



To change a plot's appearance:

1. Double-click a plot element (for example, a line, axis, or title) to open that element's **Properties** window.

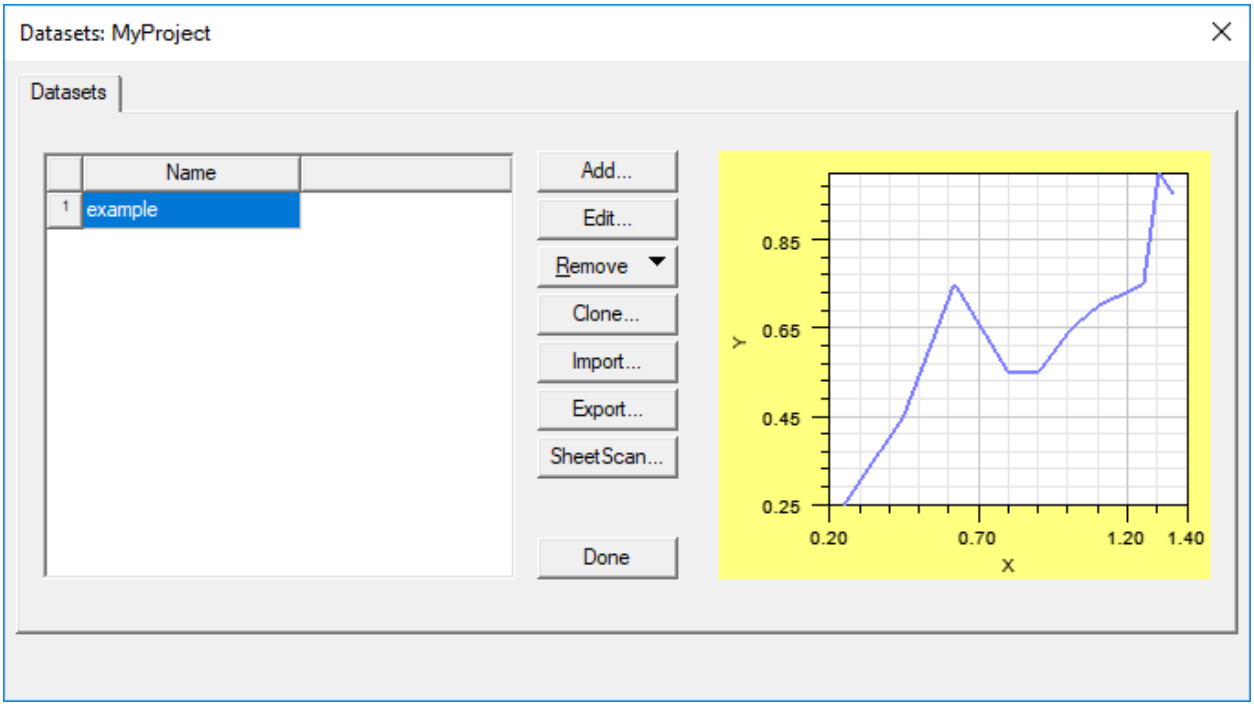
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Set the colors, fonts, and styles as applicable and as desired.
3. Click **Apply**.
4. Click **OK** to exit the **Properties** window.

   The plot now displays as specified.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Using SheetScan

SheetScan allows you to extract characteristics data from graphics such as data sheets which have been scanned and saved in any of the following formats: .bmp, .dib, .jpg, .gif, .tif, .tga, .pcx, .htm, or .html.

Access SheetScan from the **Datasets** window:

- For project-level datasets, click **Project** > **Datasets**.
- For design-level datasets, click **Maxwell 3D**, **Maxwell 2D**, or **RMxprt** > **Design Datasets**.

In addition to importing graphic files directly, SheetScan also can be used to browse the Internet for datasheet information and transfer a snapshot of the web page to the SheetScan editor where you can map axes on the image as an overlay. You can then manually add datapoints to approximate the characteristic curve(s) on the datasheet.The sampled data can then be converted to Ansys Electronics Desktop format, and the extracted data exported to an Ansys Electronics Desktop dataset or saved to a tab-delimited file.

The process for creating a dataset using SheetScan involves four basic operations:

- Loading a datasheet into SheetScan.
- Defining a coordinate system for the imported datasheet picture.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Defining a characteristic curve using the datasheet picture as reference.
- Exporting the characteristic curve data to a file or to a dataset.

SheetScan Menus and Settings

In the SheetScan window, various menus and toolbars allow you to access SheetScan functionality.

The top menu bar contains the following submenus:

- **File** – allows you to open, close, save, and print files.
- **Edit** – provides access to cut, copy, and paste functions.
- **View** – allows you to select which toolbars appear in the SheetScan window (see below).
- **Options** – provides access to document settings.
- **Picture** – provides options for loading a picture or HTML file into SheetScan, and for deleting a picture.

> **Note:**
>
> The **Delete** action cannot be undone. If you delete a picture from the SheetScan editor, you must reload it from the source file or Internet web page.

- **Coordinate System** – provides options for creating and editing a coordinate system.
- **Curve** – provides options that allow you to define a curve.
- **Window** – provides options for rearranging the currently open SheetScan windows (for example, overlapping and tiling).
- **Help** – accesses the Electronics Desktop online help.

SheetScan View Options

You can toggle the following toolbars via **View** > **Toolbar**:

- **Standard** – provides access to basic functions such as file Open and Save, Cut, Copy, Paste, Print, and Help.

- **Curve** – contains a drop-down menu allowing you to select a curve and tools for working with curve values.

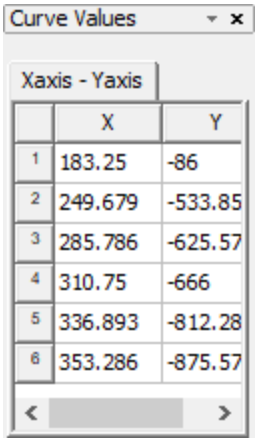
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Zoom** – provides tools for scaling the current view.

From the **View** menu, you can also toggle the following:

- **Status Bar** – displays, at the bottom of the screen, the current cursor coordinates.

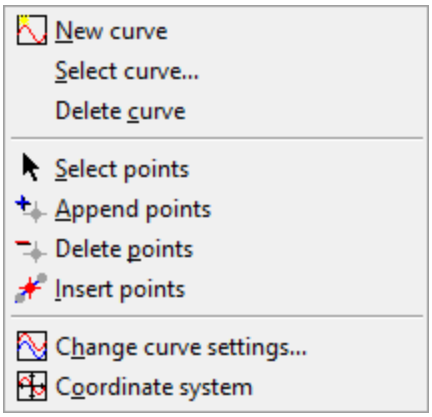
- **Curve Values** – dockable window that displays the data points used while creating a characteristic curve. Data for each curve on a sheet is displayed on its own tab. You can manually change the X and Y values in the table to fine-tune the characteristic curve.



- **Grid** – displays the coordinate grid you created over the sheet.

SheetScan Right-click Menu

Once you have loaded a sheet, you can right-click in the SheetScan window to access curve and coordinate system options:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SheetScan Settings

Default settings can be changed from **Options** > **Settings**.

The **Settings** dialog box contains three tabs:

- **Document** – allows you to set the width and height of the sheet created when a picture is imported. You can either enter the dimensions manually, or allow SheetScan to adapt the dimensions to the picture being loaded.
- **Axis** – allows you to set the name, units, scaling, and offset for the X and Y axis. You can also select **Monotonicity in X** to prevent adding consecutive data points whose X-values are not increasing.
- **Representation** – allows you to choose whether to connect points on the characteristic curve and to choose the connecting line color. You can also decide whether to display markers and choose marker colors.

> **Note:**
>
> You can also override the default settings on the **Axis** and **Representation** tabs for individual curves (See: Defining a Characteristic Curve in SheetScan).

Loading a Datasheet Picture into SheetScan

By default, SheetScan opens a new, blank datasheet editing window.

There are two ways to load a datasheet picture into the editor: directly, or by using the HTML Viewer.

Loading a Datasheet Picture Directly

1. Browse directly to the datasheet picture file by choosing **Picture** > **Load Picture** to open a file browser window.
2. When you have located the desired file, click **OK** to load the image into the SheetScan editor. Supported file types include: .bmp, .dib, .jpg, .gif, .tif, .tga, .pcx, .htm, and .html.

Loading a Datasheet Picture Using the HTML Viewer

1. Choose **Picture** > **Internet** to open the HTML Viewer.
2. Browse the Internet for the desired datasheet.
3. Resize the HTML Viewer window and adjust its scrollbars until the desired portion of the datasheet is in view.
4. Click the **To SheetScan** button ( ) to copy the visible contents into the SheetScan editor window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> To hide the datasheet picture, select **View** > **Picture**.

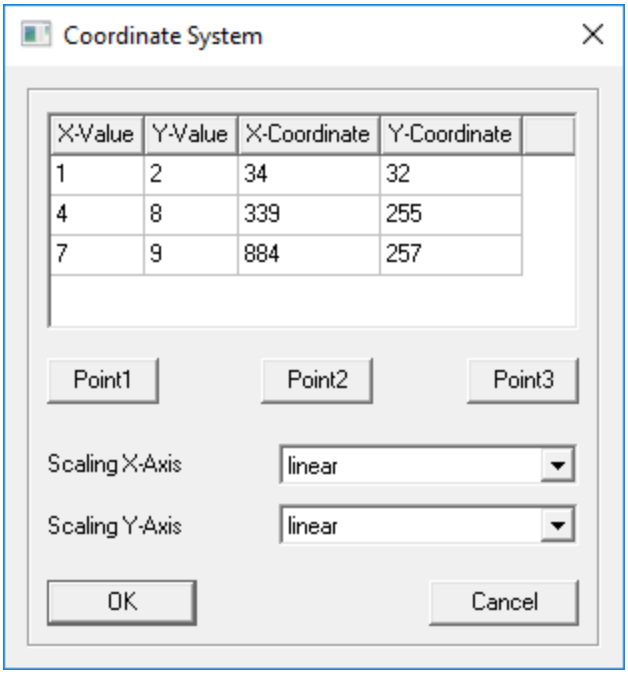
After loading a datasheet picture into the SheetScan editor, the next step is defining a coordinate system for the imported picture.

Defining a SheetScan Coordinate System

You can define a coordinate system for a graph on a datasheet picture that you have previously loaded into the SheetScan editor.

To define the coordinate system:

1. Select **Coordinate System** > **New** to open the **Coordinate System** dialog box.



   You can enter points manually or select them using a crosshair.

2. To activate the crosshair cursor, click **Point1**.

   The cursor transforms into a crosshair.

3. Position the cursor over a corner of the datasheet graph and click the left mouse button.

   The **Coordinate System** dialog box reappears, displaying the X- and Y-Coordinate values for the chosen point.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Enter the X-Value and Y-Value for this point. Typically, these values will correspond to the values taken from the axis scale values on the datasheet.
5. Select the desired scaling (linear, logarithmic, or decibel) for both the X and Y axes.
6. Repeat the above steps for **Point2** and **Point3**.
7. Click **OK**.

   The grid is placed over the graphic.

   > **Note:**
   >
   > - You can edit the grid after placement by selecting **Coordinate System** > **Properties**, by clicking the coordinate system icon on the Curve toolbar ( ), or by right-clicking in the SheetScan editing window and selecting **Coordinate System**.
   > - You can hide the grid by selecting **View** > **Grid**.

8. You can now define a Characteristic Curve.

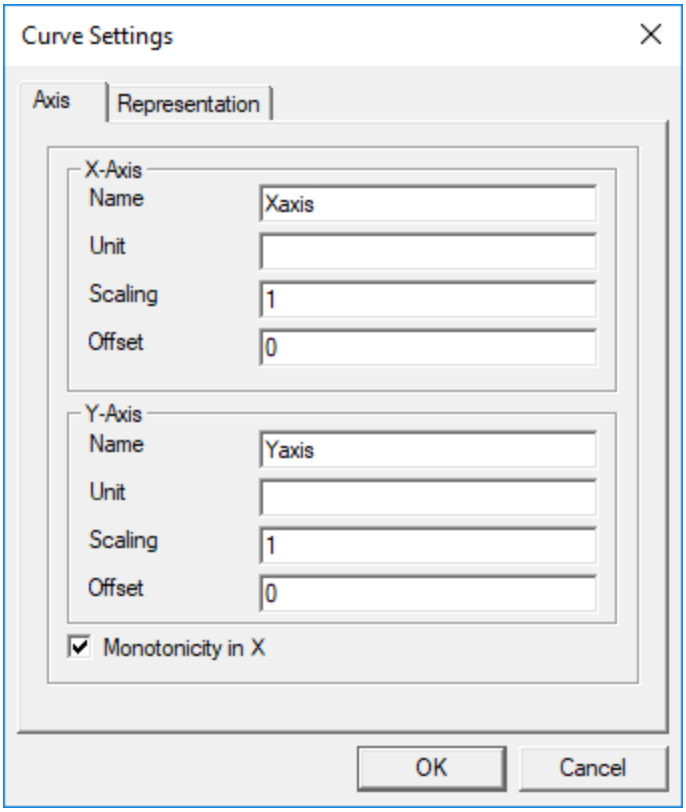
Defining a Characteristic Curve in SheetScan

Once you have loaded a datasheet picture in the editor and have defined a coordinate system, you can define one or more characteristic curves as follows:

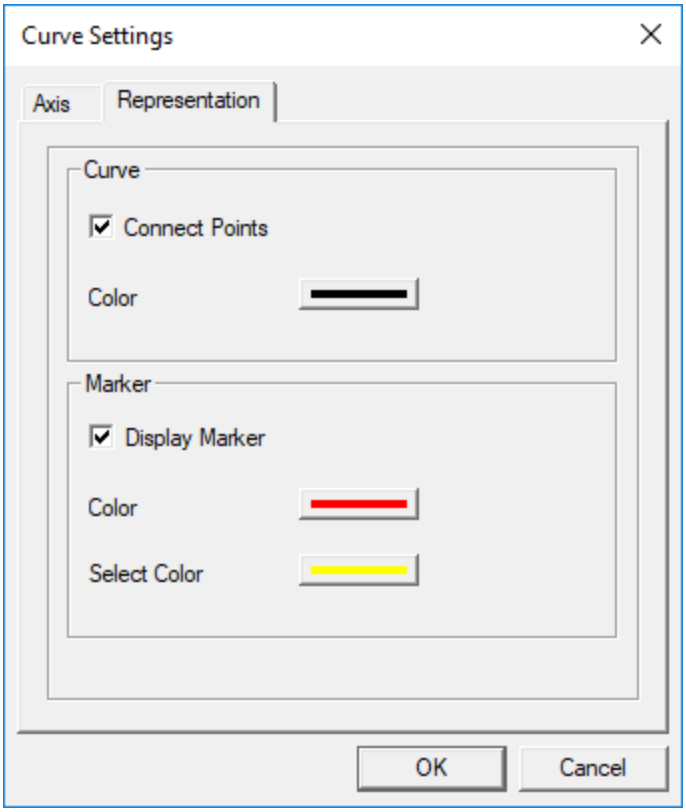
1. Choose **Curve** > **New**.

   The **Curve Settings** dialog box opens, containing the **Axis** and **Representation** tabs.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Use the fields on the **Axis** tab to define the curve's properties. Use the buttons on the **Representation** tab to change the curve's appearance.
3. Click **OK**.

   The cursor changes to a crosshair.

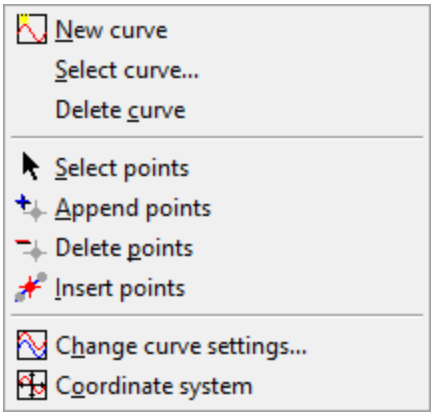
4. Click the points of the characteristic which you want to capture for the dataset. The points are connected automatically.
5. Repeat steps 1 through 4 for each additional characteristic curve you wish to define.

After characteristic curves have been defined, you can perform various operations on them. See: *Performing Operations on SheetScan Curves.*

Performing Operations on SheetScan Curves

Use the **Curve** menu (from the toolbar or by right-clicking in the sheet) to perform actions on SheetScan curves.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can perform the following actions:

- **New Curve** – allows you to define a new characteristic curve.
- **Select Curve** – opens the **Select Curve** dialog box, listing all active curves. Select the desired curve and click **OK**.
- **Delete Curve** – deletes the selected curve.

  > **Warning:**
  >
  > You cannot undo this action. If you delete a curve and its data points from the SheetScan editor, you must reconstruct it manually.

- **Select Points** – select this option, then click a point to select it. **Ctrl+click** allows you to select multiple points.
- **Append Points** – select this option, then click to add data points to the end of a curve.
- **Delete Points** – select this option, then click a data point to remove it from the curve.
- **Insert Points** – select this option, then click to insert new data points between existing data points.
- **Change Curve Settings** – opens the **Curve Settings** dialog box, where you can edit the initial curve settings.
- **Coordinate System** – opens the Coordinate System dialog box, where you can edit the coordinate system settings.

Checking for Monotonicity in X

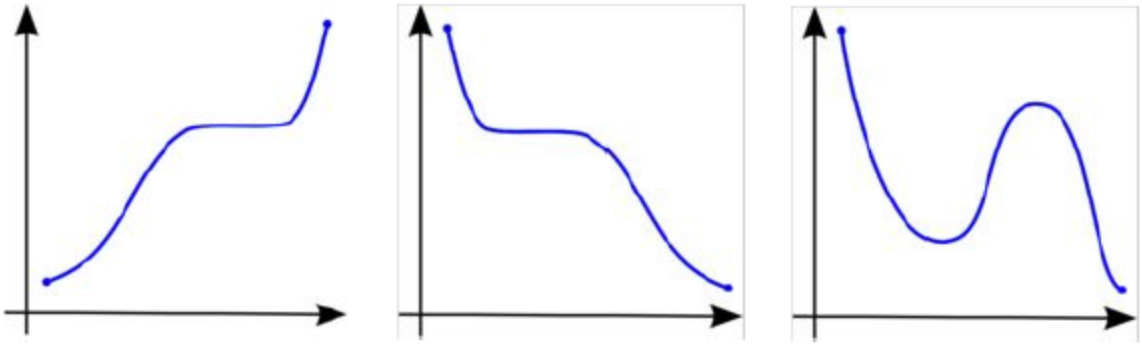
Maxwell requires that characteristics be monotonically increasing along the X-axis. In other words, successive data points must have increasing X-values, while Y-values may both increase and decrease.

To check for monotonicity in X from SheetScan:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select **Curve** > **Check Monotonicity**.

   If the characteristic curve is monotonically increasing in X-value, the check completes without notice. Typical examples of curves that meet monotonicity criteria are shown below.



2. If the characteristic curve is not monotonically increasing in X-value, a dialog box displays informing you that errors were found. Click **Yes** to have SheetScan automatically correct the errors.

Importing Characteristic Data into SheetScan

SheetScan supports data import from the following file types:

- Twin Builder Characteristic (*.mdx, *.mda)
- Microsoft Excel (*.xls, *.xlsx)
- Text (*.txt)
- Comma Separated Value (*.csv)
- Spice (*.out)
- Comtrade (*.cfg)
- TEK Oscilloscope (*.dat)

> **Note:**
>
> Before importing characteristic data, you must define a coordinate system.

To import characteristic curve data into SheetScan:

1. Click **File** > **Import**.
2. In the file **Open** dialog box, select the desired data file and click **Open**.
   a. Selecting an **.xls** or **.xlsx** file containing multiple sheets opens a **Table Properties** dialog box where you can choose the desired sheet from a drop-down menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

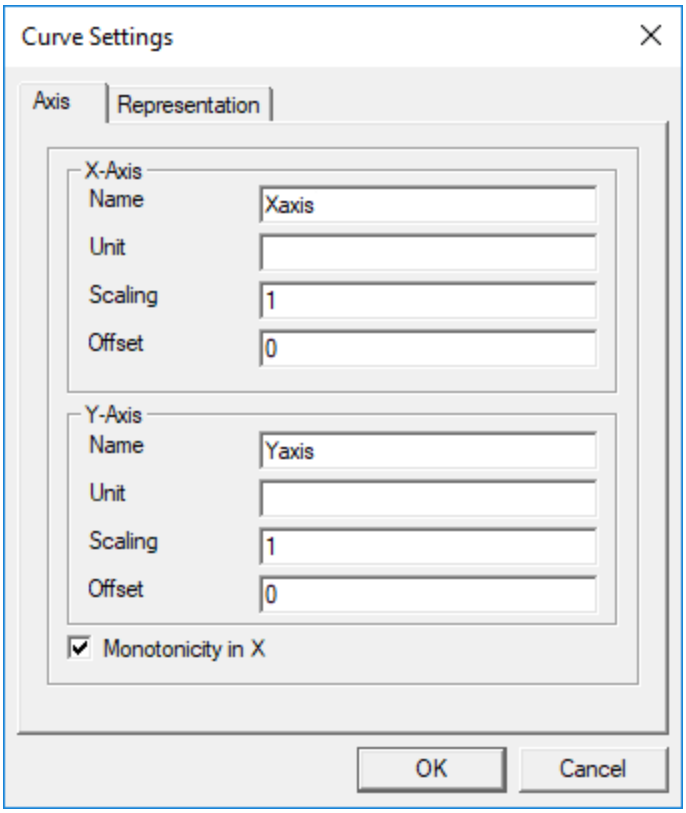
Otherwise, selecting an **.xls** or **.xlsx** file imports the data immediately into the **Add Dataset** dialog box.

> **Note:**
>
> - Only the first two columns of data are imported, the left-most column containing the X-coordinate values. The x-coordinate values for successive data points must increase within ten significant digits. Non-numeric entries are assigned a value of zero.
> - The first row of data is assumed to contain column headings and is ignored.

b. Selecting a **.txt** or **.csv** file opens an **Import** dialog box in which you can specify how to settings for reading the data in the file for import. You can choose the separator(s) and decimal symbol, as well as the line at which to begin the import. The dialog box shows both the original text and the text as it would appear when imported based on the current import settings.

When satisfied with the import settings, click **OK** to import the data.

The **Curve Settings** dialog box opens.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Change **Curve Settings** as needed and click **OK** to complete the data import.

   The new characteristic curve is added to the current SheetScan sheet.

Exporting SheetScan Data

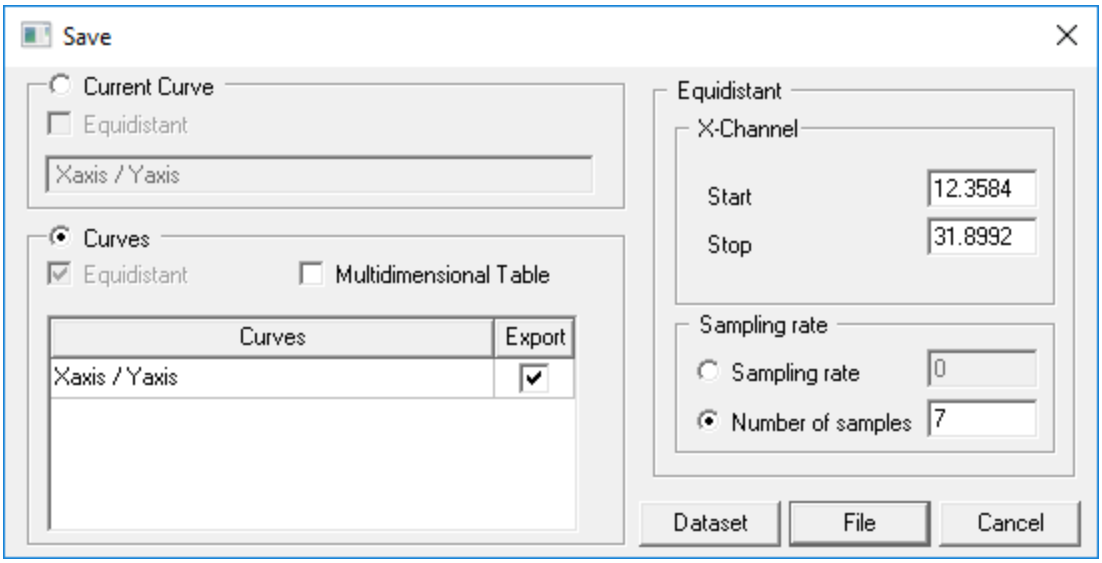
After defining a coordinate system and adding a curve, you can export SheetScan curve data directly to:

- Electronics Desktop Dataset
- Twin Builder Characteristic (*.mdx) file
- Comma Separated Value (*.csv) file
- Comtrade (*.cfg) file

To export curve data:

1. Click **File** > **Export**.

   The **Save** dialog box appears.



2. In the **Save** dialog box, choose **Current Curve** (default) to export current curve data, or **Curves** to choose the curve(s) whose data you wish to export. Choosing **Curves** reveals a list box showing all of the curves available for export.

> **Note:**
>
> The **Multidimensional Table** option is not currently supported.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Choose **Equidistant** if you want to set the **Start** and **Stop X-Channel** values and a **Sample Rate** or **Number of samples** for the exported dataset(s).
4. Click **Dataset** to export curve data directly to the project's dataset file, or click **File** to export as one of the supported file types.

Defining an Expression

Expressions are mathematical descriptions that typically contain Intrinsic Functions, such as sin (x), and arithmetic operators, such as +, -, *, and /, as well as defined variables. For example, you could define: x_size = 1mm, y_size = x_size + sin(x_size). Defining one variable in terms of another makes it a dependent variable. Dependent variables, though useful in many situations, cannot be the subject of optimization, sensitivity analysis, tuning, or statistical analysis.

The Constants tab of the **Project Variables** window lists all available pre-defined constants. These may not be reassigned a new value.

Numerical values may be entered in Ansys shorthand for scientific notation. For example, $5x10^7$ could be entered as **5e7**.

Valid Operators for Expressions

The operators that can be used to define an expression or function are performed in a sequence.

The following table lists valid operators and the sequence in which they are accepted (listed in decreasing precedence):

()	Parenthesis	1
!	Not	2
^ (or **)	Exponentiation (If you use "**" for exponentiation, as in previous software versions, it is automatically changed to "^".)	3
-	Unary minus	4
*****	Multiplication	5
/	Division	5
+	Addition	6
-	Subtraction	6
==	Equals	7
!=	Not equals	7
>	Greater than	7
<	Less than	7
>=	Greater than or equal to	7
<=	Less than or equal to	7

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

&&	Logical and	8
||	Logical or	8

Using Intrinsic Functions in Expressions

Maxwell recognizes a set of intrinsic trigonometric and mathematical functions that can be used to define expressions. Intrinsic function names are reserved and may not be used as variable names.

> **Note:**
>
> If units are not specified, all trigonometric functions interpret their arguments as radians. Likewise, return values of inverse trigonometric functions are output in radians. When the argument to a trigonometric expression is a variable, the units of the variable are assumed to be radians. To have values interpreted in degrees, supply the argument with the unit name **deg**.

The following intrinsic functions may be used to define expressions:

Function	Description	Syntax
abs	Absolute value (|x|)	abs(x)
sin	Sine	sin(x)
cos	Cosine	cos(x)
tan	Tangent	tan(x)
asin	Arcsine	asin(x)
acos	Arccosine	acos(x)
atan	Arctangent (in range of -90 to 90 degrees)	atan(x)
atan2	Arctangent (in range of -180 to 180 degrees)	atan2(y,x)
asinh	Hyperbolic Arcsine	asinh(x)
atanh	Hyperbolic Arctangent	atanh(x)
sinh	Hyperbolic Sine	sinh(x)
cosh	Hyperbolic Cosine	cosh(x)
tanh	Hyperbolic Tangent	tanh(x)
even	Returns 1 if integer part of the number is even; returns 0 otherwise.	even(x)
odd	Returns 1 if integer part of the number is odd; returns 0 otherwise.	odd(x)
sgn	Sign extraction	sgn(x)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

exp	Exponential (e^x)	exp(x)
pow	Raise to power (x^y)	pow(x,y)
if	If	if(cond_exp,true_exp,false_exp)
pwl	Piecewise Linear. (**pwl** can be used with datasets for Design Variables but not for Project variables).	pwl(dataset_exp,variable)
pwl_periodic	Piecewise Linear for periodic extrapolation on x.	pwl_periodic (dataset_exp,variable)
sqrt	Square Root	sqrt(x)
ln	Natural Logarithm (The "log" function has been discontinued. If you use "log(x)" in an expression, the software automatically changes it to "ln (x)".)	ln(x)
log10	Logarithm base 10	log10(x)
int	Truncated integer function	int(x)
nint	Nearest integer	nint(x)
max	Maximum value of two parameters	max(x,y)
min	Minimum value of two parameters	min(x,y)
mod	Modulus	mod(x,y)
rem	Returns the fractional part of a decimal number such that rem (x) = x-int(x)	rem(x)
clp	Implements smooth interpolation employing weighted impact of all points of the dataset (not just the closest one). See formula below. **Note:** If used with a large 3D dataset, clp function will degrade.	clp (*datasetName* , X,Y,Z)

clp Formula

$$clp\left(Dataset,X,Y,Z\right)=\begin{cases} DatasetValue_i \text{ if } r_i\left(X,Y,Z\right) < r_{rounding} \\ \Sigma_i \frac{DatasetValue_i}{r_i^4} \Big/ \Sigma_i \frac{1}{r_i^4} \quad otherwise \end{cases}$$

$$r_i\left(X,Y,Z\right)=\sqrt{\left(X-DatasetX_i\right)^2+\left(Y-DatasetY_i\right)^2+\left(Z-DatasetZ_i\right)^2}$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using Piecewise Linear Functions in Expressions

The following piecewise linear intrinsic functions are accepted in expressions:

- **`pwl (dataset_expression, variable)`** – piecewise linear along the X-axis and returns a corresponding Y value.
- **`pwlx (dataset_expression, variable)`** – piecewise linear with linear extrapolation on x returns a corresponding Y value.
- **`pwl_periodic (dataset_expression, variable)`** – also interpolates along the X-axis, but periodically.

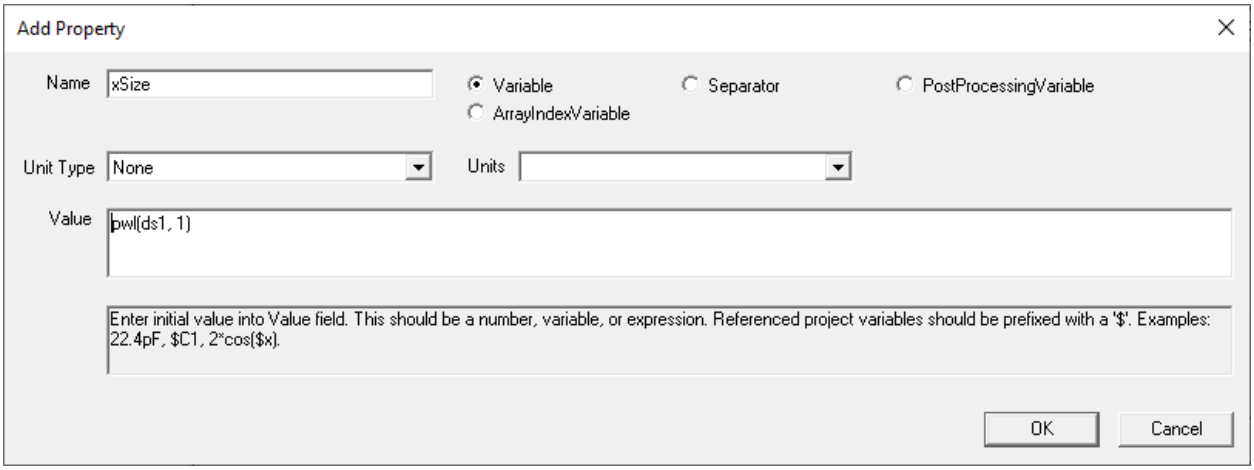
You can use **pwl** in an expression that uses datasets for such things as a frequency dependent material property (See: Adding Datasets).

For example, you can specify BulkConductivity as:

```
pwlx($ds1, Freq)
```

You can create a design variable representing a dimension `xSize` as `pwl(ds1, 1)` where `ds1` is a dataset.

Doing so looks like this:

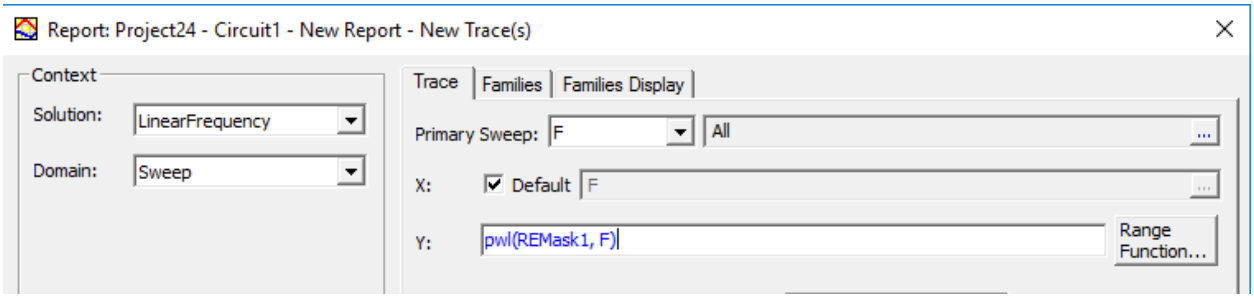


After the dataset is configured, the **Properties** window shows the new variable.

To use datasets in reports, create a report using **pwl(dataset_expression,**`variable`**)** where `variable` is the primary sweep in the report.

For example:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

pwl can be used with datasets for Design Variables but not for Project variables.

Using Dataset Expressions

A dataset is a collection of data. It can take the following form:

$ds1((x0,y0), ..., (xn,yn))

Once created, a dataset (such as $ds1) may be used as the first parameter to piecewise linear (**pwl**, **pwlx**, and **pwl_periodic**) functions, and may also be assigned to variables, in which case the variable may be used as the second parameter to **pwl**, **pwlx**, and **pwl_periodic** functions.

You can generate a dataset using a series of points in a plot on the **Datasets** dialog box (See: Adding Datasets). Each plot consists of straight line segments whose vertices represent their end points. A curve is fitted to the segments of the plot. This curve, which best fits the segmented plot, consists of the co-ordinates used in the creation of the dataset. The dataset may then be used in the piecewise linear intrinsic functions.

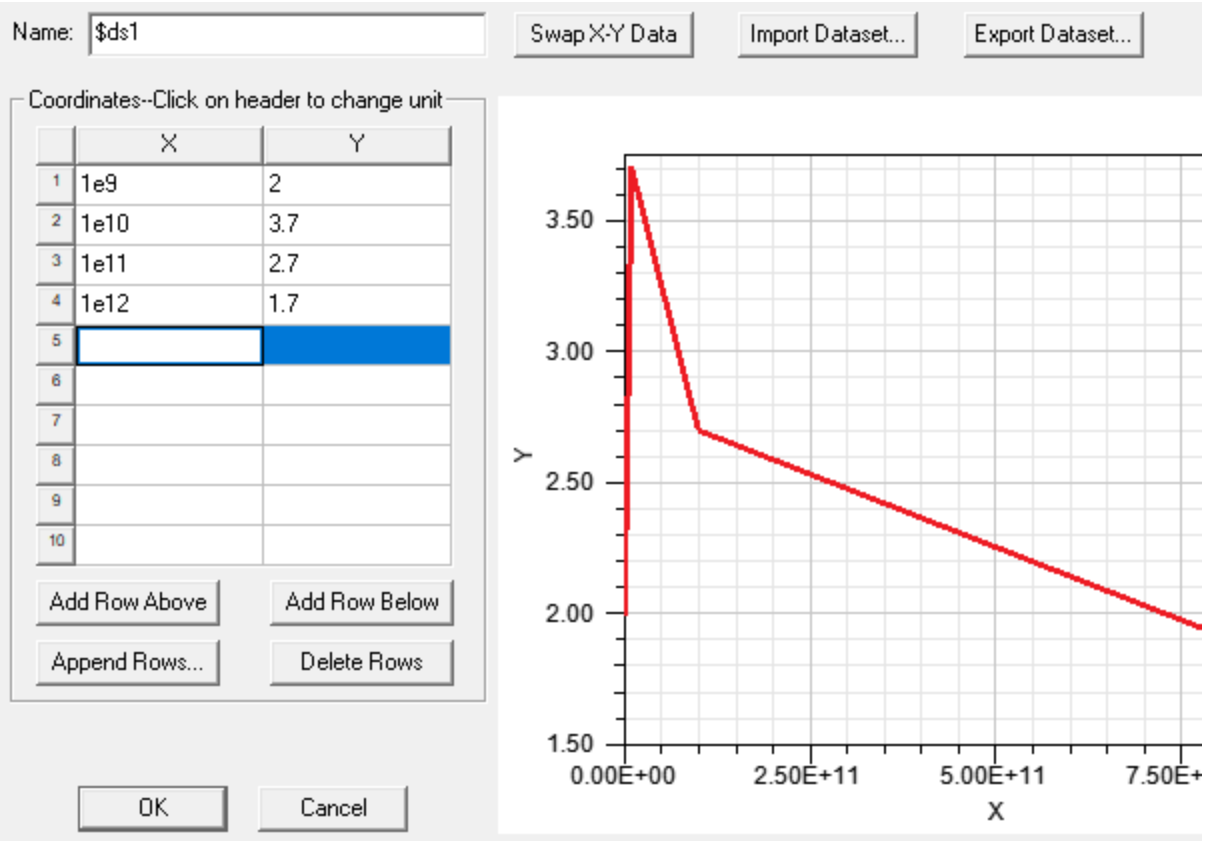
Note:

The following is an example showing how to make a material property frequency dependent using a dataset as the first parameter to a **pwl** function. The values used are arbitrary.

1. For a project-level dataset, click **Project** > **Datasets**. For a design-level dataset, click **Maxwell 3D**, **Maxwell 2D**, or **RMxprt** > **Design Datasets**.
2. Click **Add**.
3. Set the **Name** field and **Coordinates** as desired and click **OK**.

   The dataset is created.

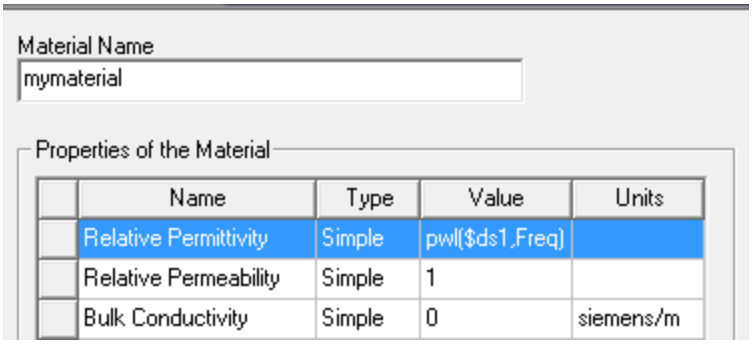
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> By default, the dollar sign ($) is assigned to a project dataset even if you do not use one while naming it.

4. Go to **Tools** > **Edit Libraries** > **Materials** to open the **Edit Libraries** dialog box.
5. Click **Add Material**.
6. Type in the piecewise linear function and use the dataset $ds1, as shown in the following figure.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



7. Click **OK**.

Handling Delta Temperature Units in Expressions

When Temperature and Delta Temperature quantities are used as operands to plus or minus operations in an expression, they are handled specially. The biggest difference is an automatic unit change of the resulting values based on the units of the operands.

- When two Temperature quantities are being subtracted (either the first or both operands is/are temperature quantities), the result value has Delta Temperature units.
- When a Delta Temperature quantity is added or subtracted from a Temperature quantity, the result value has Temperature units.
- Adding or subtracting two Delta Temperature quantities results in a quantity with Delta Temperature units.
- Subtracting two temperature quantities is an observable behavior change when compared to earlier releases of Ansys Electronics Desktop.

Temperature Units

Celsius Family

- cel, delta_cel (Legacy name for delta_cel, celdiff continues to be supported)

Kelvin Family

- mkel, delta_mkel (milli Kelvin) (Legacy name for delta_mkel, mkeldiff continues to be supported)
- ckel, delta_ckel (centi Kelvin)
- dkel, delta_dkel (deci Kelvin)
- kel, delta_kel (Legacy name for delta_kel, keldiff continues to be supported)

Farenheit Family

- fah, delta_fah

Temperature-related Use Cases

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The high-level legal use cases (in unit types) are:

- Temperature - Temperature = DeltaTemperature
- Temperature +/- DeltaTemperature = Temperature
- DeltaTemperature +/- DeltaTemperature = DeltaTemperature

The minor variations in these use cases specify how the actual units (Celsius is a unit of the Temperature unit type) are handled.

Temperature - Temperature

If they have the same unit, then the Delta Temperature unit will be a matching one.

For example:

- 10 cel - 5 cel = 5 delta_cel
- 100 kel - 90kel = 10 delta_kel
- 10 fah - 1 fah = 9 delta_fah

If they have different units, they are converted to the default units for temperature difference.

For example:

- 10cel - 1 kel = 9 delta_kel

Temperature +/- DeltaTemperature

The resulting temperature quantity will retain the units of the Temperature quantity (first operand)

DeltaTemperature +/- DeltaTemperature

- If they have the same unit, the resulting Delta Temperature quantity will retain that.
- If they have different units, the resulting Delta Temperature quantity will have the default Delta Temperature units
- 5 delta_cel + 10 delta_cel = 15 delta_cel
- 10 delta_cel + 1 delta_kel = 11 delta_kel

All other use cases in a plus or minus arithmetic operation are physically meaningless and revert to previous behavior, where Electronics Desktop converts quantities to their SI values and then operates on the plain numbers.

- Temperature + Temperature
- Temperature +/- non-Temperature/non-Delta Temperature
- Delta Temperature +/- non-Temperature/non-Delta Temperature
- non-Temperature/non-Delta Temperature +/- Delta Temperature
- non-Temperature/non-Delta Temperature +/- Temperature

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining Mathematical Functions

A mathematical function is an expression that references another defined variable. A function's definition can include both expressions and variables.

The following mathematical functions may be used to define expressions:

Basic Functions	**/**, **+**, **-**, *, mod (modulus), ****** (exponentiation), **-** (Unary minus), **==** (equals), **!** (not), **!=** (not equals), **>** (greater than), **<** (less than), **>=** (greater than equals), **<=** (less than equals), **&&** (logical and), **||** (logical or)
Intrinsic Functions	**if**, **abs**, **exp**, **pow**, **ln** (natural log), **log10** (log to the base 10), **sqrt**
Trigonometric Expressions	**sin**, **cos**, **tan**, **asin**, **acos**, **atan**, **sinh**, **cosh**, **tanh**

The predefined variables X, Y, Z, Phi, Theta, R, and Rho must be entered as such. X, Y, and Z are the rectangular (Cartesian) coordinates. Phi, Theta, and Rho are the spherical coordinates. R is the cylindrical radius, and Rho is the spherical radius.

If you do not specify units, all trigonometric expressions expect their arguments to be in radians, and the inverse trigonometric functions' return values are in radians. If you want to use degrees, you must supply the unit name **deg**. When the argument to a trigonometric expression is a variable, the units are assumed to be radians. These function names are reserved and may not be used as variable names.

As far as expression evaluation is concerned: units are conversion factors (that is, from the given unit to SI). Note also that the evaluated value of an expression) is always interpreted as in SI units.

Maxwell ACT Extensions Wizards

The following Maxwell extension Wizards implemented in Ansys ACT can be accessed through the **View** > **ACT Extensions** user interface.

- Maxwell Eccentricity
- Machine Toolkit

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> The **ACT Extensions** window and the design wizards it contains (*5G Wizard, HFSS Antenna Design Toolkit, HFSS-EMA3D Link,* and *Maxwell Eccentricities*) are only available for the Windows version of the Ansys Electronics Desktop software. These items are not available when the software is installed on a Linux platform.

Using the Maxwell Machine Toolkit Wizard (Windows only)

The Machine Toolkit wizard (Windows only) requires a valid 2D or 3D transient design with motion setup. The machine toolkit uses the original design to create a new design that is then used to perform the analysis. The machine toolkit supports five machine types:

- PM (Permanent Magnet) Synchronous Machines
- Induction Machines
- Synchronous Reluctance Machines
- Switched Reluctance Machines
- Wound-rotor Synchronous Machines

The machine toolkit can be used in the following four scenarios. Choose the one appropriate for your design needs.

Interactive Solve

1. For interactive solve, define the machine parameters, then click **Finish**. The toolkit will solve the parametric table sweep and calculate efficiency maps immediately on the same machine. Make sure you have set the analysis setting you wish to use (DSO/HPC).

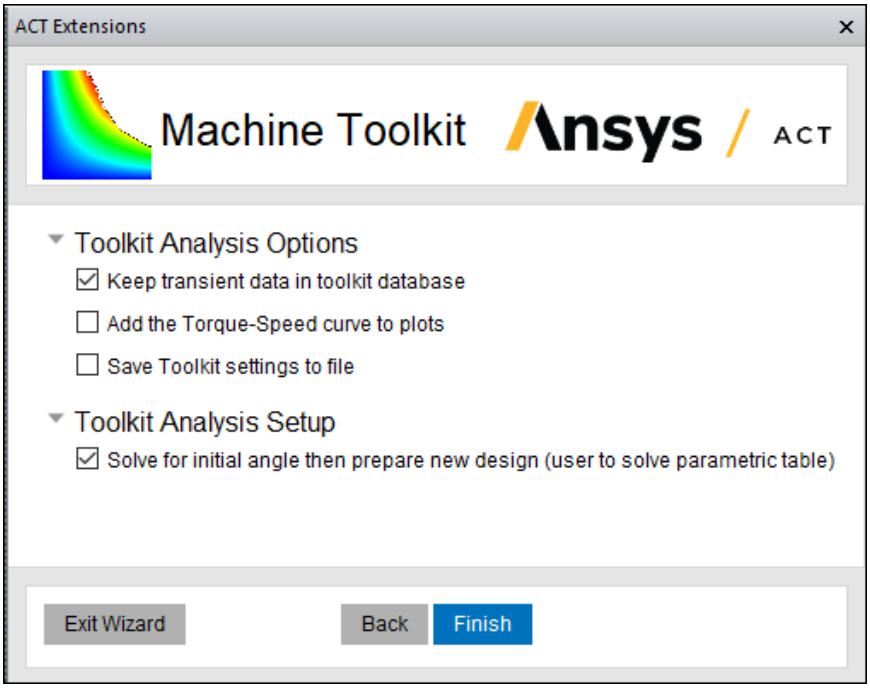
   > **Note:** The user does not have to solve the parametric sweep, it is solved automatically.

- The toolkit creates a new design.
- Parametric sweep is solved.
- For PM Synchronous machines, Synchronous Reluctance machines, and Wound-rotor Synchronous machines: If not provided by the user, the DQ initial alignment angle is calculated by solving an independent simulation.
- Results are extracted.
- Postprocessing is performed, and reports (contour plots) are created in the design.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

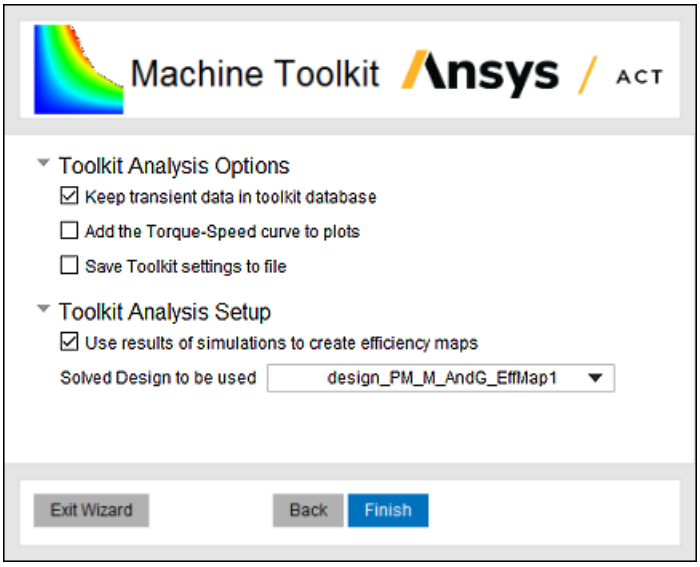
Delayed Solve

1. For delayed solve, define the machine parameters, and check **Solve for initial angle then prepare new design (user to solve parametric table)**.



2. Click **Finish**.
   - A new design is created.
   - For PM Synchronous machines, Synchronous Reluctance machines, and Wound-rotor Synchronous machines: If not provided by the user, the DQ initial alignment angle is calculated by solving an independent simulation.
   - The parametric sweep is then solved separately by the user.
3. After solving, reopen the toolkit. The parameter entries will be prefilled.
4. Navigate to the last wizard page and select **Use results of simulations to create efficiency maps** and select the **Solved Design to be used**.

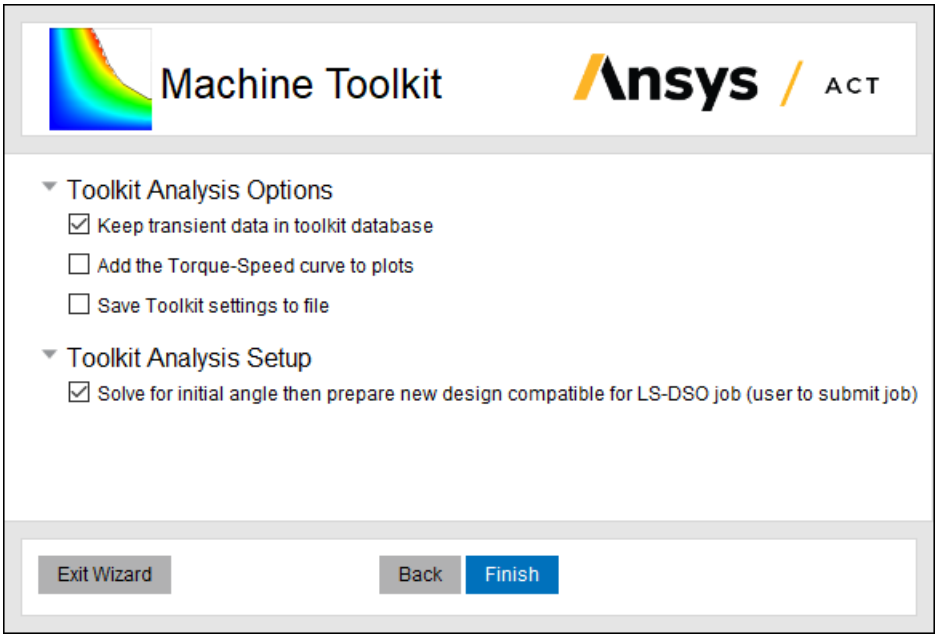
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Click **Finish**.
    - Results are extracted.
    - Postprocessing is performed, and reports (contour plots) are created in the design.

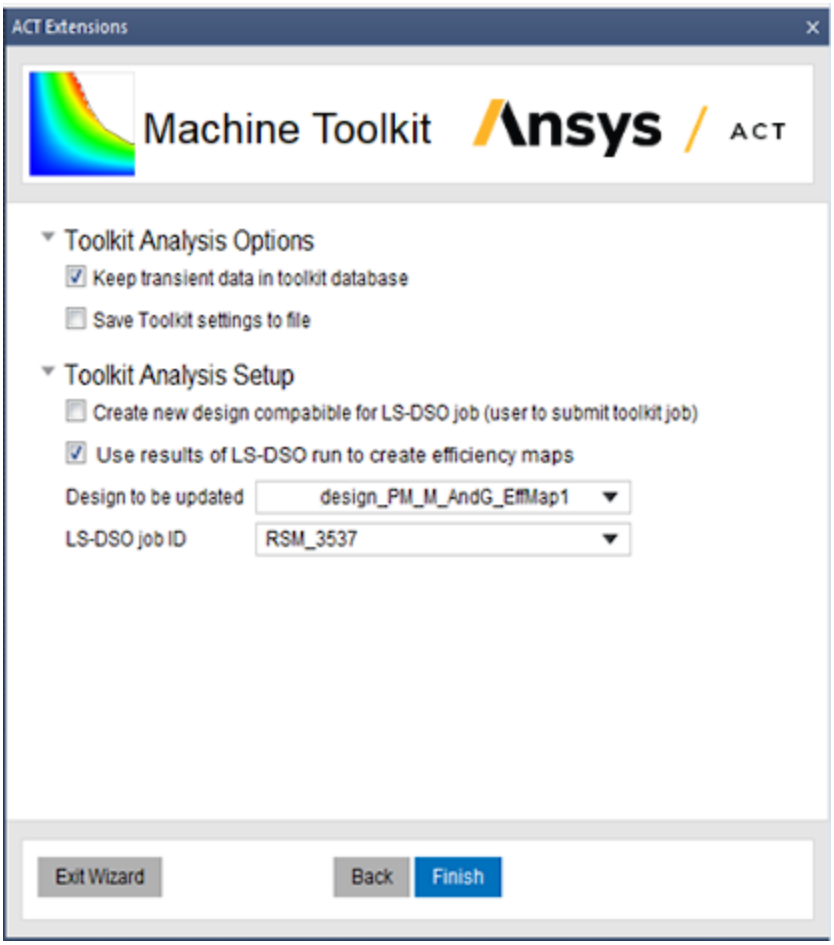
Solve Using LS-DSO

1. For solve using LS-DSO, define the machine parameters, and check **Solve for initial angle then prepare new design compatible for LS-DSO job (user to submit job)**.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **Finish**.
    - A new design is created.
    - For PM Synchronous machines, Synchronous Reluctance machines, and Wound-rotor Synchronous machines: If not provided by the user, the DQ initial alignment angle is calculated by solving an independent simulation.
    - The new design contains a parametric sweep that needs to be solved.
    - The toolkit stops to let user solve the LS-DSO job separately.
3. Upon completion of the LS-DSO job, restart the toolkit. The parameter entries will be prefilled.
4. Navigate to the last wizard page and select **Use results of LS-DSO run to create efficiency maps**, then select the **Design to be updated** and also the **LS-DSO job ID** that was used for the job.

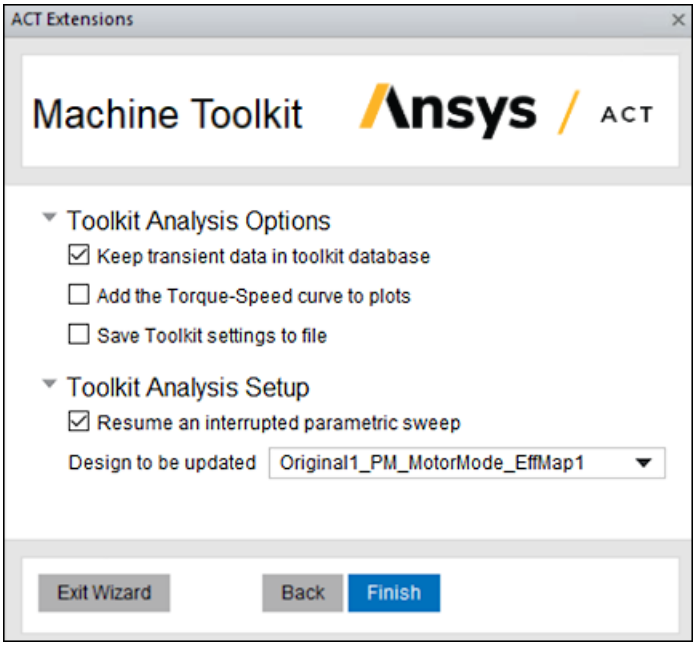


5. Click **Finish**.
    - Results are extracted.
    - Post-processing is performed, and reports (contour plots) are created in the design.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Resume an Interrupted Parametric Sweep Run

1. For this scenario, first select an existing original design. The parameters of the interrupted efficiency map design are automatically prefilled in the UI.
2. Navigate to the last wizard page and select **Resume an interrupted parametric sweep** and select the **Design to be updated**.



The parametric sweep of this design will start from where it was interrupted, and efficiency maps will be generated in post processing.

Generate Efficiency Map Only

1. For this scenario, first select an existing design. The parameter entries will be prefilled.
2. Navigate to the last wizard page and select **Use results of simulations to create efficiency maps** and select the **Solved Design to be used**.
3. Both general transient and periodic (or half-periodic) TDM can be used to conduct parametric analysis and create performance maps. Define the TDM option in the analysis setup of the original design. When conventional transient method or general transient TDM is used to extract performance data, it is necessary to define Number of electric periods simulated and Number of periods used for average in DOE settings. There is an option for the user to define two time intervals, and also specify the number of electric periods and number of time steps per period for each interval. When periodic or half-periodic TDM is enabled in the original design, such options are hidden in DOE settings, and the Machine Toolkit automatically defines the time duration for each simulation.

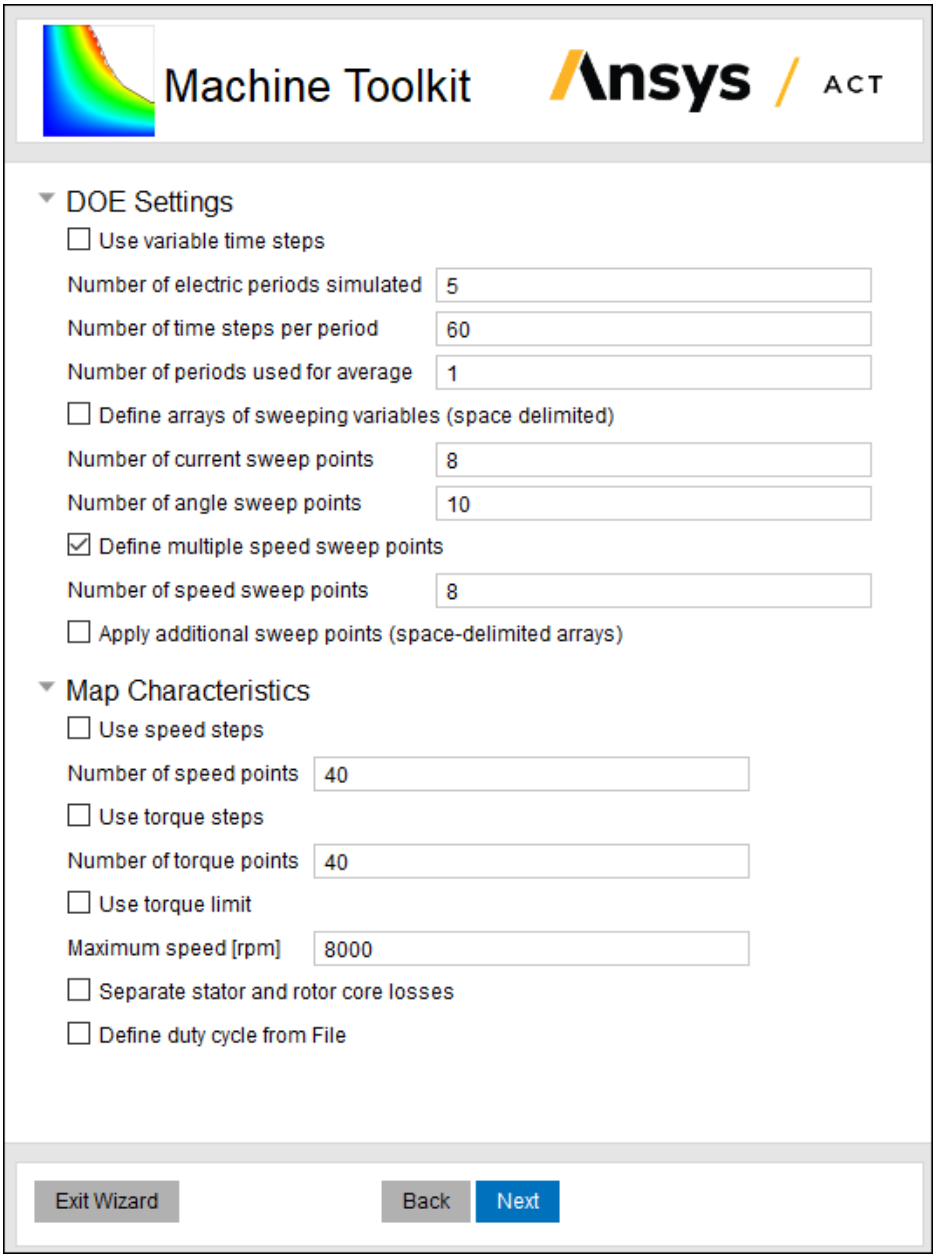
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Machine Toolkit provides two options for the user to determine sweep points in the parametric analysis. The first option is to define the number of sweep points for each independent variable, and then Machine Toolkit assigns the sweep points that are uniformly distributed in the solution domain. The second option is to define the numerical arrays of the sweeping variables, where each value is separated from its adjacent values by a single space character.

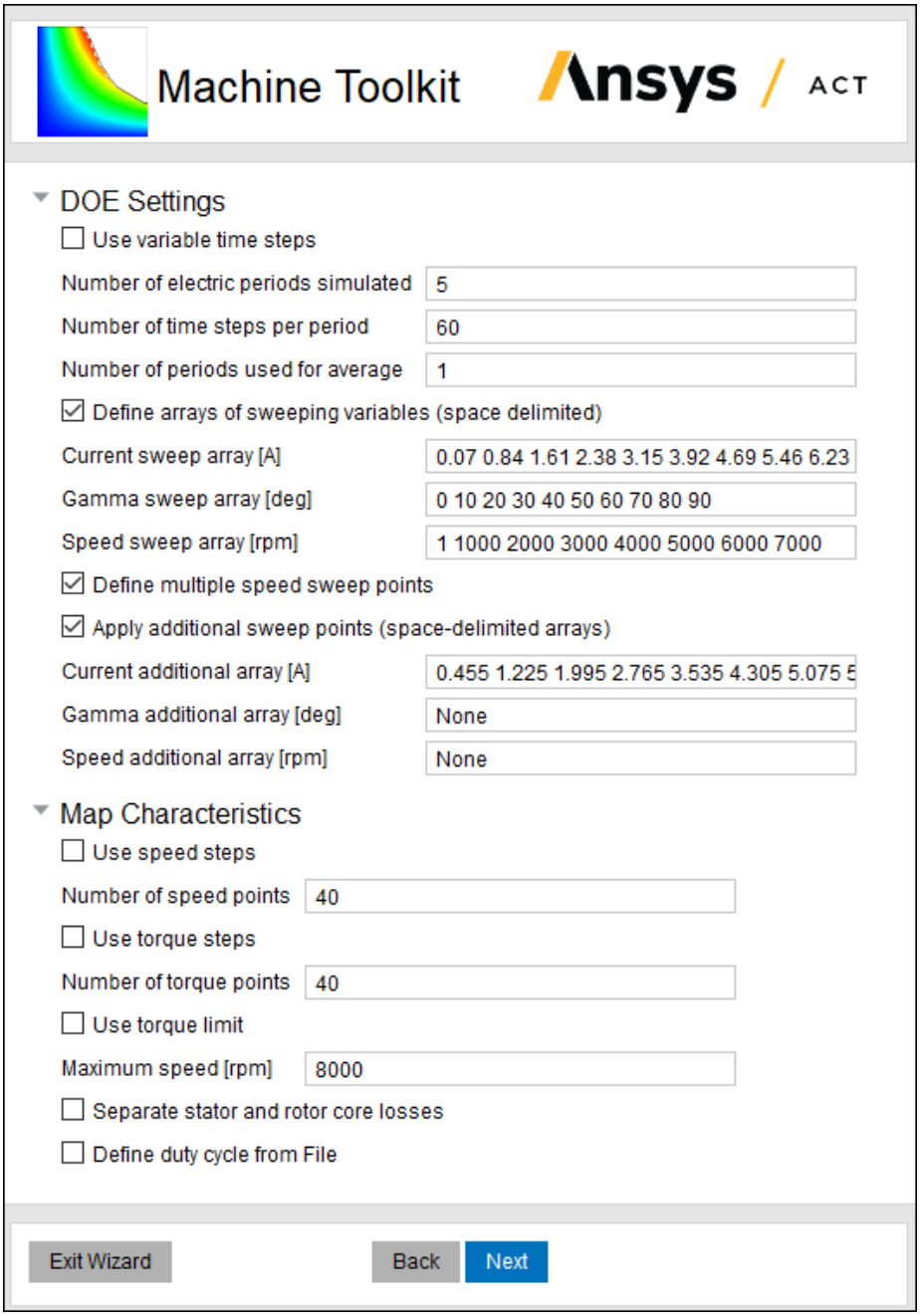
Once the parametric analysis of the prior efficiency map simulation is completed, there is an option for the user to apply additional sweep points to the parametric analysis. The operating points that are already computed in the prior parametric analysis are automatically skipped, and Machine Toolkit only simulates these additional points.

> **Note:** For an induction machine model with general transient analysis, the simulations of some operating points in the prior parametric sweep may not be skipped.
>
> TDM options cannot be enabled for Switched Reluctance machines.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For induction machines, to use periodic TDM or half-periodic TDM, all sources including induced eddy current have to be in the same frequency. This means periodic TDM or half-periodic TDM is only applicable when rotor is locked at zero speed. To this end, we can introduce equivalent conductivity σ_eq associated with locked rotor to approximate the performance at original speed (σ_eq = σ * slip). For this kind of equivalence, fundamental component will be correctly modeled. This approximation is normally good enough to the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

simulated performances including torque, current, voltage, etc with one exception that computed loss in the rotor needs to be rescaled in terms of slip.

The following steps must be done before proceeding to use periodic TDM solver in the toolkit:

a. Add a project parameter called **$slip**. The **Value** and **Evaluated Value** can be any number since the parametric sweep will override it.
b. Modify the rotor bar's material, and change the conductivity from **X** to **X*$slip**.
c. Modify the end connection resistance from **X** to **X/$slip**.
d. Select **Enable two level for HPC** setting.

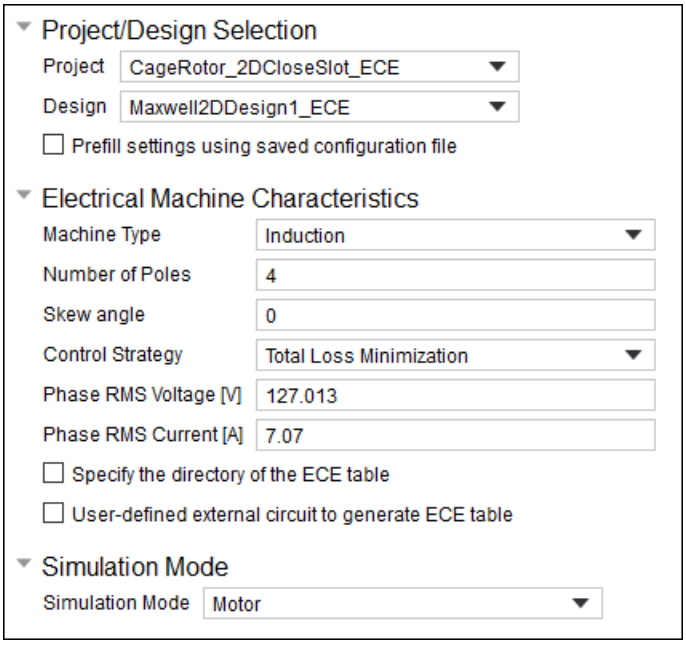
4. Click **Finish**.
   - Results are extracted.
   - Post-processing is performed, and reports (contour plots) are created in the design.

Create Efficiency Maps Based on Reduced-Order Model

Machine Toolkit supports efficiency map generation based on the reduced-order model (ROM) for induction machines. This feature must be implemented in a design with AC Magnetic solution type. The following parameters must be defined through the Machine Toolkit UI about the characteristics of the efficiency maps, including phase RMS voltage, phase RMS current, maximum torque and speed, core loss correction factor, control strategy, simulation mode, friction loss, windage loss and their reference speed.

Refer to the item ECEIM_Model for details about its input parameters and the mathematical representations of the ECE ROM for induction machines. All identified parameters of the ECE ROM are saved in the file *ece_table.txt*, and the Machine Toolkit needs this file to generate efficiency maps. There are three options for the user to obtain this file and define its directory in the Machine Toolkit. The first option is that the user directly specifies the file by checking **Specify the directory of the ECE table**. The second option is that the user first defines an external circuit that includes an ECEIM_Model module, then the Machine Toolkit launches the analysis and automatically obtains the file *ece_table.txt*. The third option is that the Machine Toolkit creates an external circuit with the ECEIM_Model module and imports it to the definition of Excitations, then runs the simulation to get this table. Additional parameters must be input by the user if the third option is selected, including the skew angle, phase RMS current, number of current sweep points, and the resistance and end inductance of the windings. The user can also define the current sweep array for the ECEIM_Model:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Defining Machine Toolkit Parameters

> **Note:**
>
> The **ACT Extensions** window and the design wizards it contains (*5G Wizard, HFSS Antenna Design Toolkit, HFSS-EMA3D Link,* and *Maxwell Eccentricities*) are only available for the Windows version of the Ansys Electronics Desktop software. These items are not available when the software is installed on a Linux platform.

The machine toolkit supports five machine types:

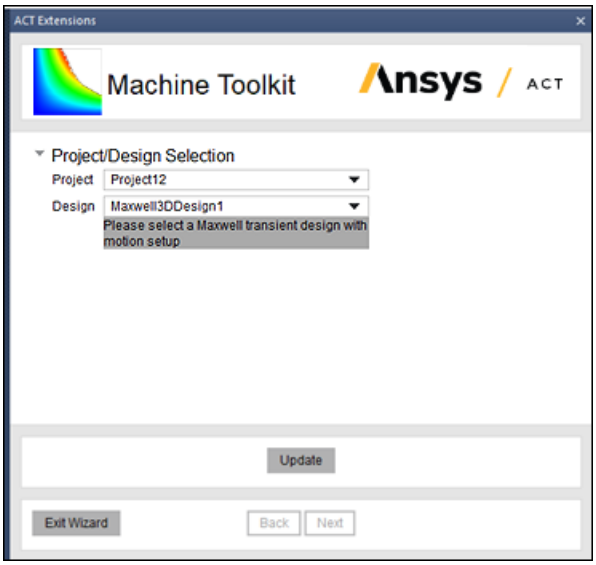
- PM (Permanent Magnet) Synchronous Machines
- Induction Machines
- Synchronous Reluctance Machines
- Switched Reluctance Machines
- Wound-rotor Synchronous Machines

> **Note:** The Machine Toolkit requires a valid 2D or 3D transient design with motion setup. The machine toolkit uses the original design to create a new design which is then used to perform the analysis.

The general workflow for using the Machine Toolkit to define machine parameters is as follows:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the ACT Extensions Wizards window, click on the Machine Toolkit icon to launch the wizard, and select a valid transient design with motion.
2. You can open a project/design in the project manager if one is not already open.

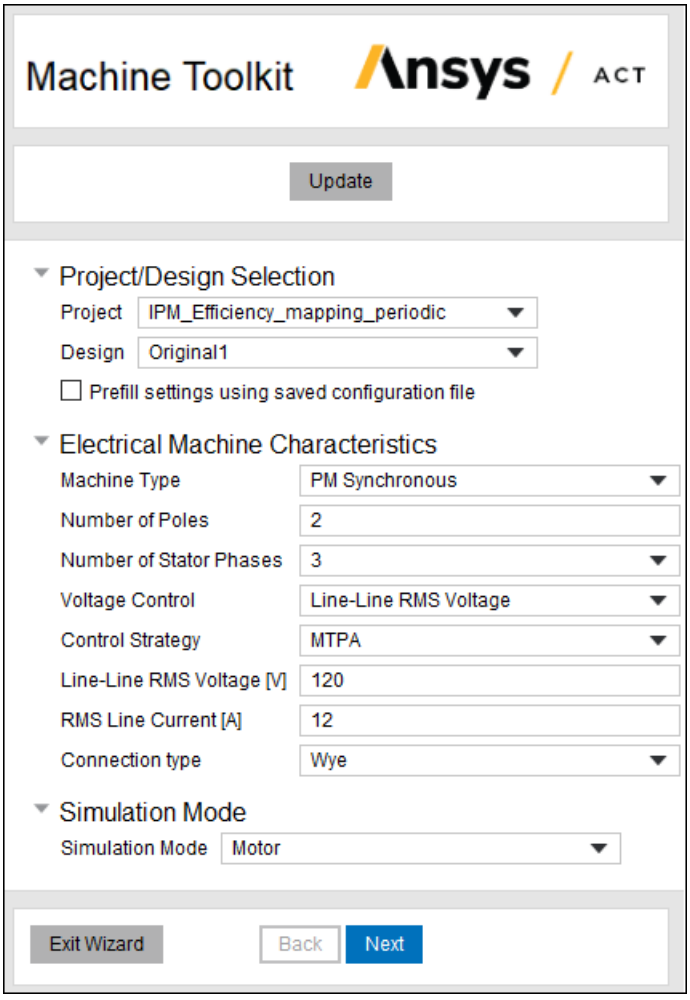


After loading the project, select **Update** to refresh the Project/Design selections.

3. Select the desired project and design. Only designs that do not use reserved local variables are shown in the drop-down menu.

   Reserved local variables are Speed_TSC, Imax_TSC, Gamma_TSC, Vmax_TSC, Slip_TSC, Freq_TSC, and Field_TSC.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Enter all characteristics of the machine: **Machine Type** (**PM Synchronous**, **Induction**, **Synchronous Reluctance**, **Switched Reluctance**, or **Wound-rotor Synchronous**), **Number of Poles**, *etc.*, in the various fields. Also select the desired **Simulation Mode** (**Motor** or **Generator**).

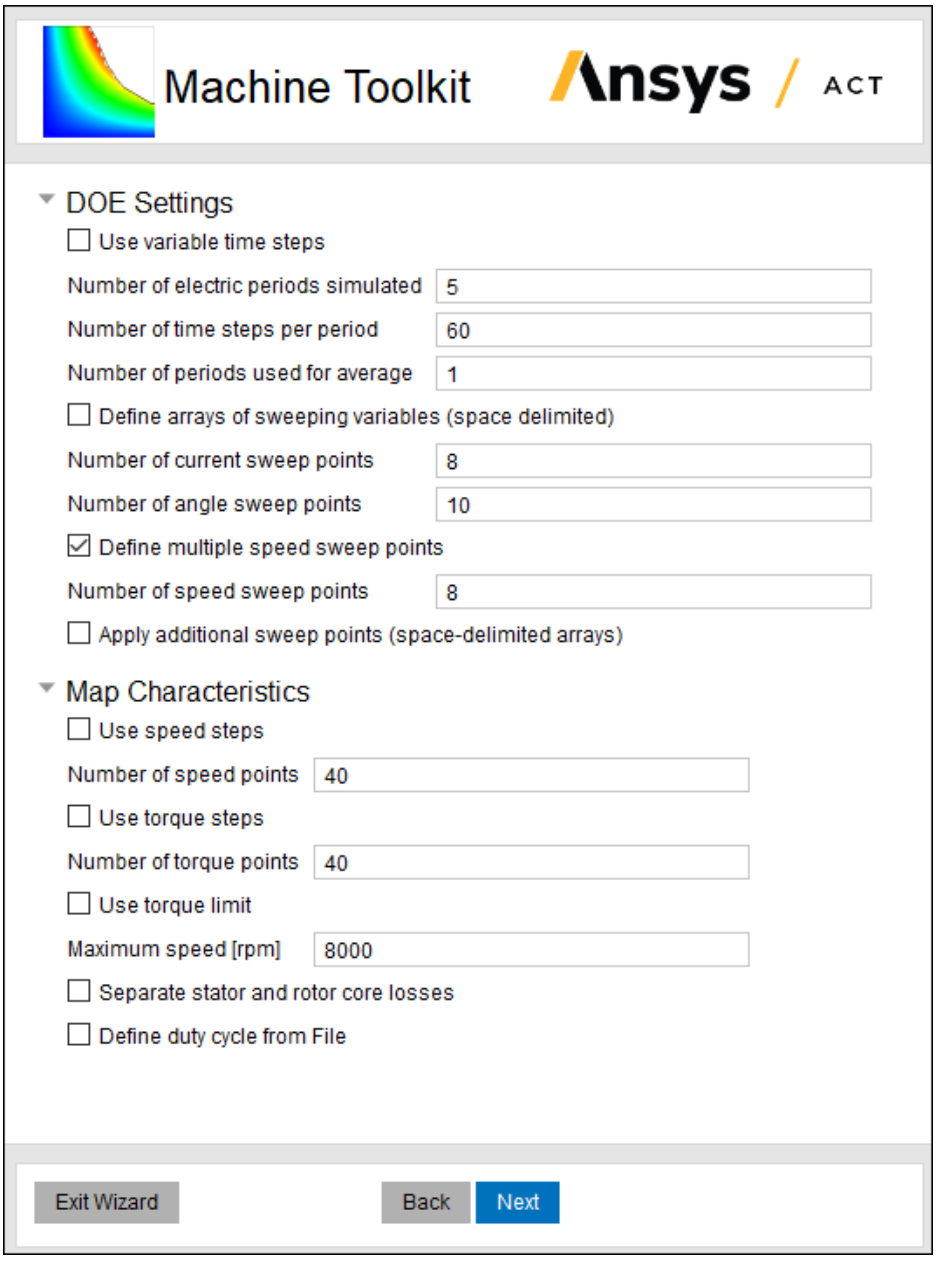
   > **Note:** You can prefill all settings using a previously saved configuration .sav file by selecting **Prefill settings using saved configuration file**.
   >
   > If you have previously used the toolkit with the current project, the toolkit prefills entries automatically.

5. Click **Next** to go to the next page of the wizard. Here you enter the desired **DOE** (design of experiments) **Settings**, and the **Map Characteristics** used to generate the efficiency map when the design is solved.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

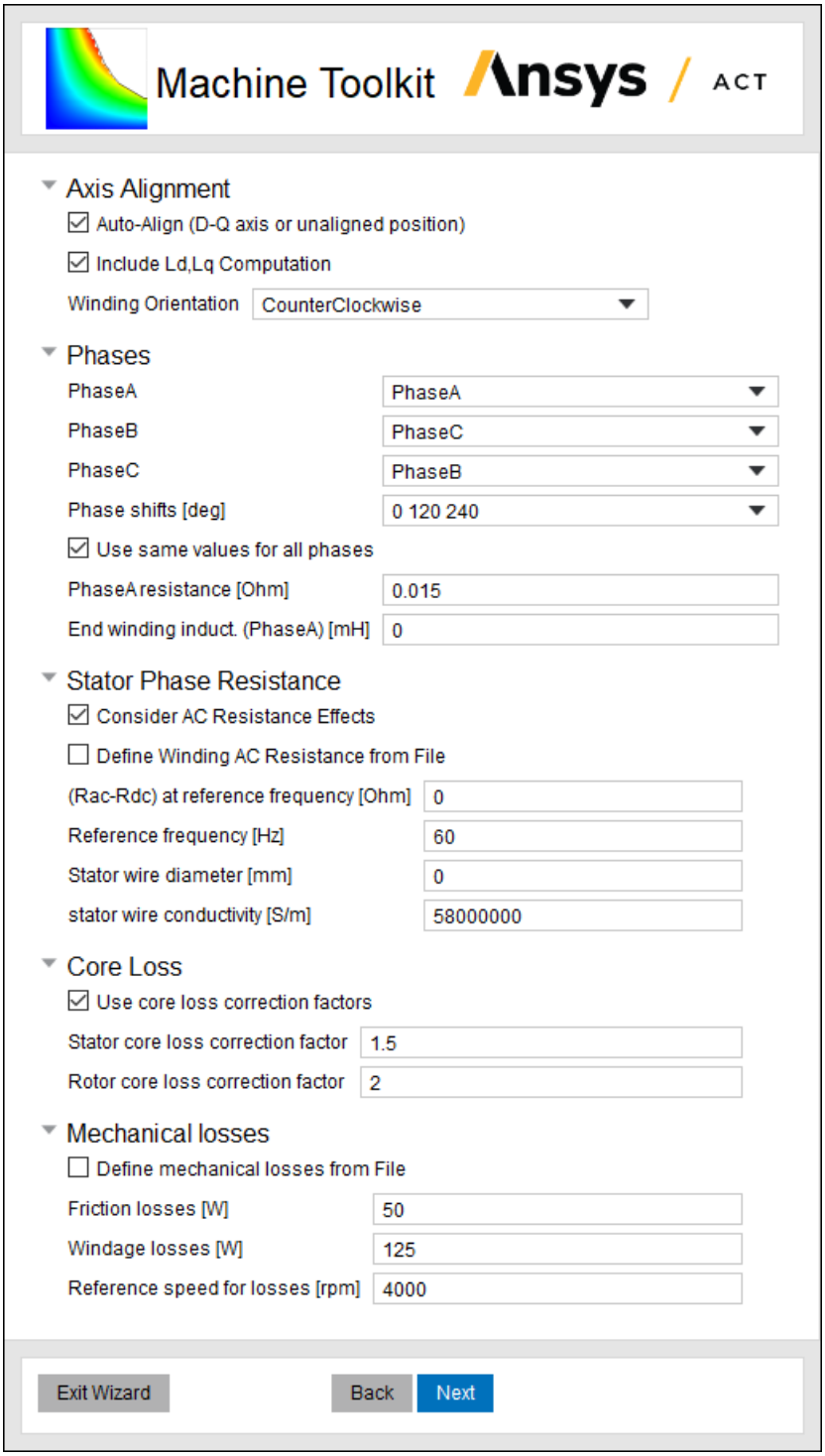
> **Note:** The content of the DOE Settings and Map Characteristics change based on which machine type you selected on the previous page.

Note that check boxes allow you to **Use speed steps** and **Use torque steps** instead of using the default speed and torque points. A parametric sweep will be created to account for the number of points (or steps).



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

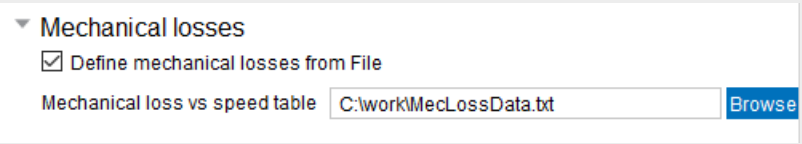
6. When finished, click **Next** to move to the next page of the wizard.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For PM Synchronous machines, Synchronous Reluctance machines, and Wound-rotor Synchronous machines, you can provide the alignment angle or have the toolkit calculate it. No matter which simulation mode is chosen, the initial angle must be determined first.
- For Phases, provide phase names, phase shifts, and other phase information.
- Optionally, provide custom loss information (Core Loss, Stator Phase Resistance winding losses, and Mechanical Losses).

> **Note:** There are three options for AC winding loss calculation:
>
> - Apply the analytical AC winding loss model, and the user is required to input the stator wire diameter and conductivity, as well as AC resistance at the reference frequency.
> - Use the Litz wire model.
> - Input a .txt file that provides a frequency vs. AC resistance table; in this file, the first line should include the names and units of frequency and resistance, for example, "Freq[Hz] Rac[ohm]", followed by combinations of frequency vs. resistance data in each line, for example, "60 1.5".

> **Note:** Mechanical losses include friction and windage losses at a reference speed.
>
> You can optionally "Define mechanical losses from File", then specify the location of a user-defined “Mechanical loss vs speed table” text file.
>
> 
>

7. When finished, click **Next** to move to the next page of the wizard.
   - It is recommended to have the **Keep transient data in toolkit database** option enabled to retain the data because the extraction process can be lengthy.
   - Check **Save Toolkit settings to file** if you want to save all settings to a `.sav` file for use with another project. (For the current project, the settings are saved automatically.)
   - Check **Disable smoothing in post processing** if you want to bypass the smoothing algorithm, which may improve the fidelity but increase the noise level of the data in the performance maps. You can also define the **Number of processes in post processing** depending on the CPU specifications, to enable parallel

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

computing and reduce the simulation time for the optimization that searches for the optimal operating condition on each point in the performance maps.

- You can click **Back** to review or edit settings on previous pages.
- If you choose to **Exit Wizard**, all settings are automatically saved and loaded the next time the current project is loaded.
- **Toolkit Analysis Setup** options are used for the various scenarios described in Using the Maxwell Machine Toolkit Wizard.

8. Click **Finish** to complete the process for the various scenarios as described in Using the Maxwell Machine Toolkit Wizard.

Using the Maxwell Eccentricity Wizard

> **Note:**
>
> The **ACT Extensions** window and the design wizards it contains (*5G Wizard, HFSS Antenna Design Toolkit, HFSS-EMA3D Link,* and *Maxwell Eccentricities*) are only available for the Windows version of the Ansys Electronics Desktop software. These items are not available when the software is installed on a Linux platform.

You can use the Maxwell Eccentricity wizard to model both 2D and 3D transient designs that have cylindrical rotational motion setup.

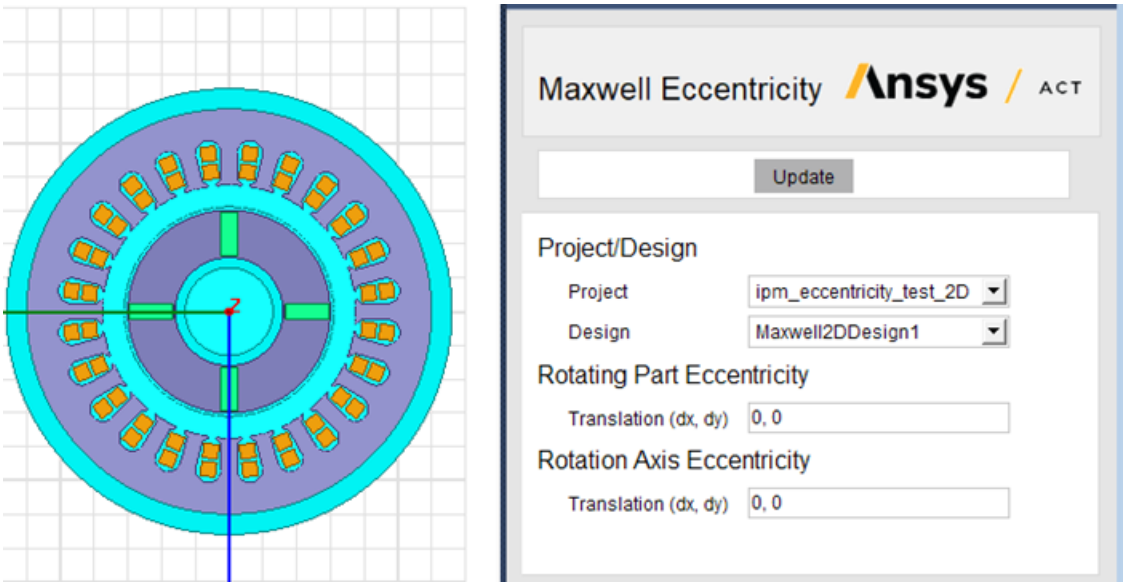
> **Note:**
>
> If you want to generate a 3D rotational eccentricity motor design from an RMxprt-generated Maxwell 3D design, you must enter **Eccentricity 1** in the RMxprt **Design Setting** window's **User Defined Data tab** for the RMxprt design *before* generating the Maxwell 3D design.

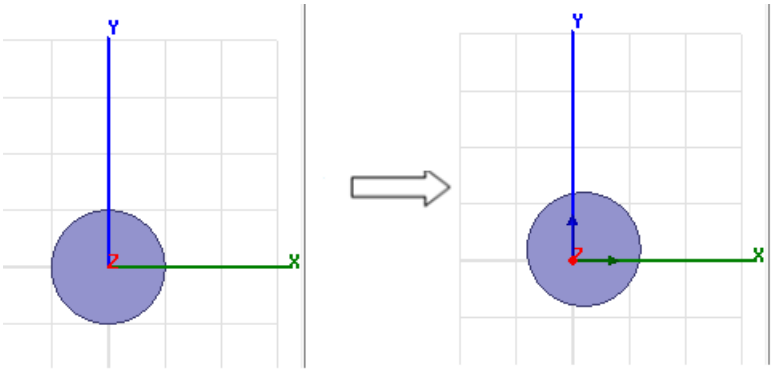
1. Select **View** > **ACT Extensions** to open the ACT Extensions window.
2. Click the **Wizards** button to open the Wizards page, then click **Maxwell Eccentricity** to load the Maxwell Eccentricity setup page.
3. Select the project and design you want to use.

> **Note:**
>
> - The **Update** button synchronizes the project and design selection lists with those currently loaded in the Desktop.
> - You can click **Exit Wizard** to close the eccentricity wizard and return to the wizard start page without executing any changes.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

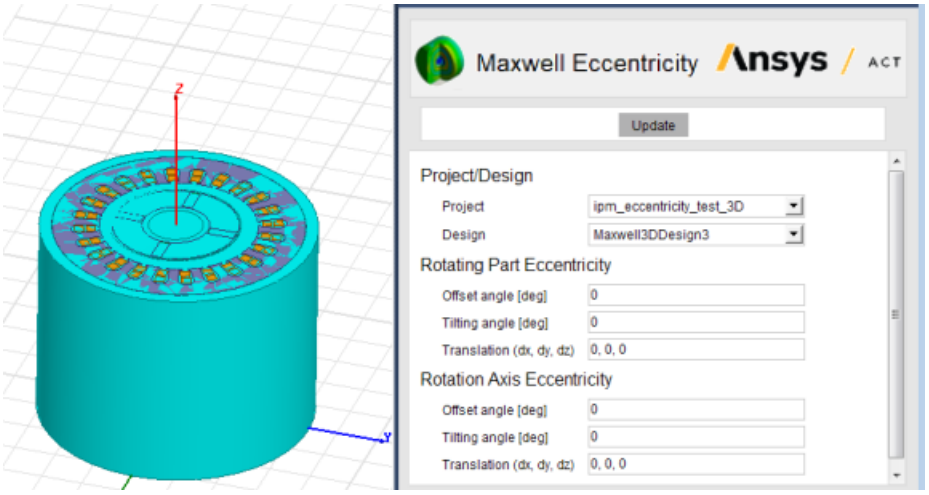
4. Depending on whether your design is 2D or 3D, do one of the following:
    - For a 2D Maxwell project, enter the desired dx and dy translation values to set up the Rotating Part Eccentricity and Rotation Axis Eccentricity. The unit of measure for translation values is the model's unit, which is set using **Modeler** > **Units**.



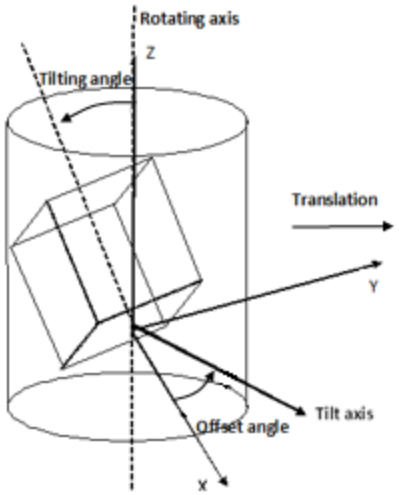


    - For a 3D Maxwell project, enter the desired dx, dy , and dz translation values, the offset angle, and the tilting angle for the rotating part eccentricity axis and rotation axis. The unit of measure for translation values is the model's unit, which is set using **Modeler** > **Units**.

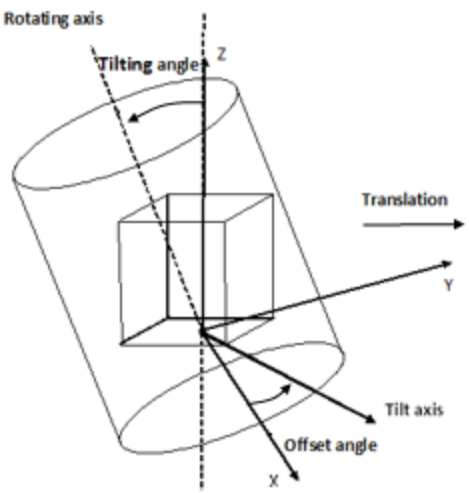
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Rotation Part Eccentricity



Rotation Axis Eccentricity



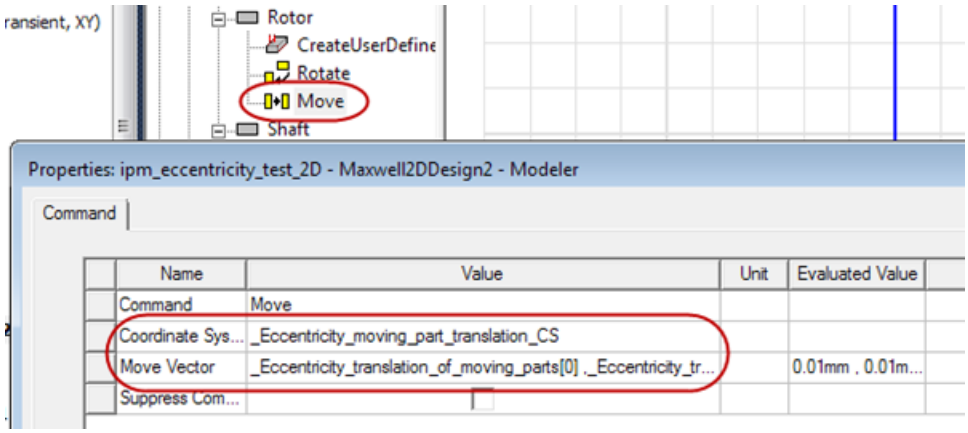
5. Click the **Finish** button at the bottom of the page. The page closes and the wizard adds a new design in the same project, in which the Maxwell geometry has been updated in accordance with the eccentricity setup values you entered, as explained in the sections that follow.

For 2D Designs

- Design variables are created for each of the data entered in the wizard.
  - For rotating part eccentricity: _Eccentricity_translation_of_moving_parts
  - For rotation axis eccentricity:_Eccentricity_translation_of_rotating_axis

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The moving parts in the design motion setup are shifted according to the data defined for **Rotating Part Eccentricity** in the Wizard. New coordinate systems are generated as the object coordinate systems for moving parts, according to the same operations.
- The band is translated according to the data defined for **Rotation Axis Eccentricity** in the wizard.
- The **Moving Vector** entry in the design's motion setup is replaced with the new direction (represented by an axis of a new coordinate system) generated according to the data defined for **Rotation Axis Eccentricity** in the Wizard.



- New design coordinate systems are created.

  _Eccentricity_rotating_axis_translation_CS

  _Eccentricity_moving_part_translation_CS

  _Eccentricity_moving_part_CS_<*part name*> – one for each moving part

Note:

- All the above operations of translation, as well as the creation of new coordinate systems, are dependent on the eccentricity design variables.
- You can modify these design variables to adjust the eccentricity.
- The newly added coordinate systems should not be changed manually by the user.
- Using the wizard to modify a design that has already been set up for eccentricity is not recommended.

For 3D Designs

- Design variables are created for each of the data entered in the wizard.

  For rotating part eccentricity:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- _Eccentricity_offset_angle_of_moving_parts
- _Eccentricity_translation_of_moving_parts
- _Eccentricity_tilting_angle_of_moving_parts

For rotation axis eccentricity:

- _Eccentricity_offset_angle_of_motion_axis
- _Eccentricity_translation_of_motion_axis
- _Eccentricity_tilting_angle_of_motion_axis

- The moving parts in the design motion setup are rotated and shifted according to the data defined for **Rotating Part Eccentricity** in the Wizard. New coordinate systems are generated as the object coordinate systems for moving parts, according to the same operations.
- The band is rotated and translated according to the data defined for **Rotation Axis Eccentricity** in the wizard.
- The **Moving Vector** entry in the design's motion setup is replaced with the new direction (represented by an axis of a new coordinate system) generated according to the data defined for **Rotation Axis Eccentricity** in the Wizard.
- New design coordinate systems are created.

  For the motion moving vector:

  - _Eccentricity_motion_axis_CS
  - _Eccentricity_motion_axis_tilting_axis_CS

  For the moving parts:

  - _Eccentricity_moving_part_tilting_axis_CS
  - _Eccentricity_moving_part_translation_CS
  - _Eccentricity_moving_part_CS_<*part name*> – one for each moving part

Note:

- All the above operations of rotation and translation, as well as the creation of new coordinate systems, are dependent on the eccentricity design variables.
- You can modify these design variables to adjust the eccentricity.
- The newly added coordinate systems should not be changed manually by the user.
- Using the wizard to modify a design that has already been set up for eccentricity is not recommended.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

When modeling motion eccentricity, the position and direction of the moving objects and/or band are adjusted based on the eccentricity setup. In such cases, all the adjustments are referenced to the original position and direction of the geometry of each object, As a result, after you finish the eccentricity setup and click the **Finish** button, all the geometries in the design created by the wizard are updated to new positions based on the eccentricity setup. If you want to edit geometry, you should do so in the original design. After editing the geometry, you can then redo the eccentricity setup to create a new design.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4 - Setting Up a Maxwell Design

After you insert a design, you do not need to perform the basic steps sequentially, but they all must be completed before a solution can be generated.

To set up a Maxwell design, follow this general procedure:

1. Insert a Maxwell design into a project. After you insert the design, you can then set the model's units of measurement and the background material.
2. Draw the model geometry.
3. Specify the solver type.
4. Assign material characteristics to objects.

   > **Important:** Always make sure the material properties assigned to an object correspond to the real properties of the materials in the electromagnetic device that is being simulated. Material properties supplied in the default library are generic properties and may not always be substituted for actual properties.

5. Assign boundaries and excitations.
6. (*For Transient Solutions Only*) Set up motion.
7. Add parameters for which you want to solve.
8. Specify mesh settings.
9. Specify how Maxwell will compute the solution.
10. (Optional) Set up any optimetrics you want to run.
11. Run the simulation.
12. View solution results, postprocess results, view reports, and create field overlays.
13. (Optional) Export the circuit to generate a circuit equivalent of the model.

The basic modeling entity is the design (model). The next level up is the project. A project is a collection of one or more designs (models) that is saved in a single *.aedt file. A new project is automatically created when Ansys Electronics Desktop is launched. As many Maxwell designs as needed can be added to a single project.

When setting up a Maxwell design, specific commands are available via the menu bar or ribbons using the mouse. You can also work directly in the Project Manager tree of the desktop.

Inserting a Maxwell Design

The first step in setting up a Maxwell project is to add a design to the active project.

To insert a design:

1. Click **Project** and select one of the following:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Insert Maxwell 3D Design**
- **Insert Maxwell 2D Design**
- **Insert RMxprt Design**

2. The new design is listed in the project tree. It is named Maxwell3DDesign*n*, Maxwell2DDesign*n*, or RMxprtDesign*n* by default, where *n* is a number signifying the order in which the design was added to the project.

The **Modeler** window appears to the right of the Project Manager. You can now create the model geometry.

> **Note:** Click the plus sign to the left of the design icon in the project tree to expand the project tree and view specific data about the model.

Creating New 2D Designs from 3D Designs

The Maxwell system contains both 2D and 3D projects. Often, you may want to take a cross-section of a 3D model to perform a 2D analysis. The following cases represent the scenarios available for automatically converting models:

- 3D model to 2D XY model
- 3D model to 2D RZ model

You must keep the following points in mind before you convert any project:

- A 2D design is created in the same project.
- All 3D geometry is used, you cannot select which geometry to include in the conversion.
- If there are any sheet objects in 3D, only those sheets that lie in the same plane as the section plane are converted.
- 1D objects are not converted from 3D to 2D.

Converting a 3D Model to 2D

To create a 2D design from an existing 3D design:

1. Click **Maxwell 3D** > **Create 2D Design**.
   The **Create 2D Design** dialog box appears.
2. Select the **Coordinate System** to be used for creating sections in the 3D model.
3. Select the **Section Plane** within the chosen coordinate system. You can select XY, YZ, or ZX.
4. Select the **Geometry Mode** to use for the target 2D design. You can select either XY or RZ.
5. Click **OK** to create the 2D design.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The modeler creates a new design within the current project and generates the 2D model within the new design. All material properties and material assignments are copied to the new design.

Creating New 3D Designs from 2D Designs

The Maxwell system contains both 2D and 3D projects. Often, you may want to sweep or rotate a 2D model to create a full 3D design. The following cases represent the scenarios available for automatically converting models from 2D to 3D:

- 2D XY model to 3D model
- 2D RZ model to 3D model

You must keep the following points in mind before you convert any project:

- A 3D design is created in the same project.
- All 2D geometry is used, you cannot select which geometry to include in the conversion.
- 1D objects are converted from 2D into 3D sheet objects.

Converting 2D XY Model to 3D

To create a 3D design from an existing 2D XY design:

1. Click **Maxwell 2D** > **Create 3D Design**.

   The **Sweep Along Vector** dialog box appears.

2. Enter a value in the **Length along z-axis** box to obtain a swept length of the model.
3. Choose the desired **Dimension** of the sweep distance entered from the drop-down menu.

4. Click **OK** to create the 3D design.

The modeler creates a new design within the current project and generates the 3D model within the new design. All material properties and material assignments are copied to the new design.

Converting 2D RZ Model to 3D

To create a 3D design from an existing 2D RZ design:

1. Click **Maxwell 2D** > **Create 3D Design**.

   The **Sweep Around Z Axis** dialog box appears.

2. Enter the **Angle of Sweep** to obtain a full 360° 3D model or a wedge model to a selectable degree.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Enter the desired **Draft Angle** and **Draft Type**.

   > **Warning:** Entering a draft angle for a model where objects touch the z axis results in a error due to an invalid object being requested. Draft angles may only be used when no objects are touching the z axis.

4. Enter the **Number of Segments** to use in creating the 3D model. If you do not change the default value (=0), the project will be created with true surfaces.
5. Click **OK** to create the 3D design.

The modeler creates a new design within the current project and generates the 3D model within the new design. All material properties and material assignments are copied to the new design.

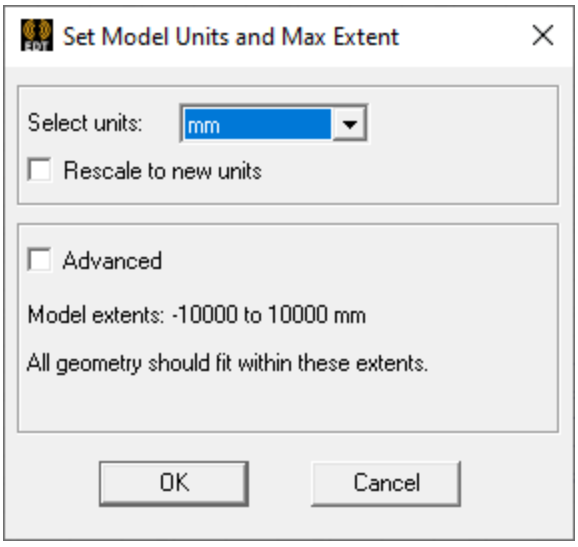
Setting Units of Measurement for the Model

You can specify the units of measurement for drawing geometric models. After the units of measurement have been specified, they are assigned to the objects in the **3D Modeler** window. You can then choose to display the model's dimensions in the new units, or rescale the model's dimensions to the new units.

To set the model's units of measurement:

1. Click **Modeler** > **Units**.

   The **Set Model Units** dialog box appears.

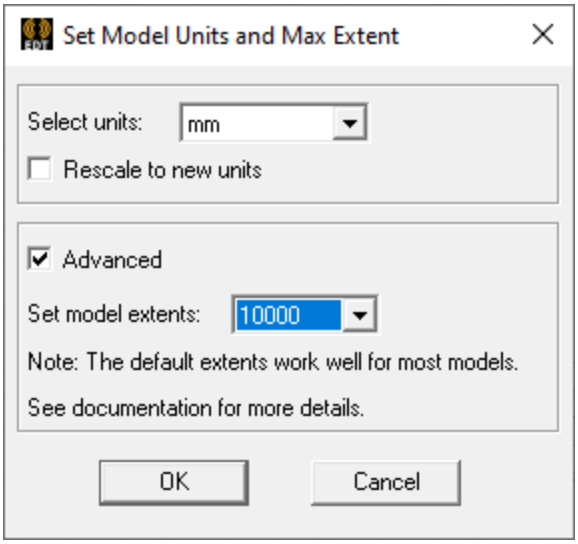


2. Select the new units for the model from the **Select units** drop-down menu.
3. Specify how the change in units affects the model:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select the **Rescale to new units** option to rescale the dimensions to the new units. For example, selecting centimeters (cm) as the new unit of measurement results in a dimension of 10 millimeters (mm) becoming 10 cm.
- Clear the **Rescale to new units** option (the default) to convert the dimensions to the new units without changing their scale. For example, selecting cm as the new unit of measurement results in a dimension of 10 mm becoming 1 cm.

4. While most models should fit the default, you can check Advanced to enable Max model extent (E) to specify model extents in the 3D modeler.



Parasolid kernel has a strict size limit of +/- 500 units. As many AEDT models go beyond this limit in the desired units, we have implemented internal scaling to allow creation of models with a larger extent. With our default model extent of 10,000, all geometry will be internally scaled down by a factor of 100 to fit within parasolid size limit. We recommend using the default extent of 10,000 as much as possible. Modeling operations will output an error if model goes beyond 50,000 units.

A second extent option of 100 is available in the advanced options.

If model extent is set to 100, we do not scale geometry internally. Modeling operations will output an error if model goes beyond +/- 500 units. Extent of 100 should be only used in rare situations where it is determined that scaling is causing a failure, and there is no workaround.

5. Click **OK** to apply the new units to the model.

Design Settings for the Maxwell Application

The Design Settings window allows you to specify how the simulator will deal with some aspects of the design. It is accessed from **Maxwell** > **Design Settings**.The tabs present in this window

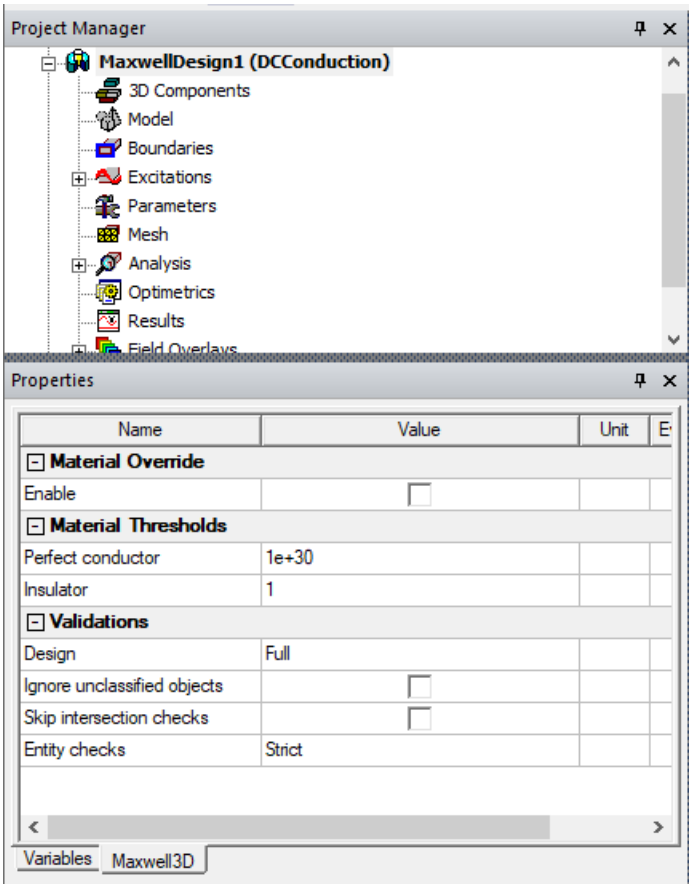
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and the options available on each tab vary with the solution type. Here are some of the typical options:

- Set the Material Thresholds for treating materials as conductors/insulators
- Set the Model Depth Settings
- Set the Material Override (Maxwell 3D only) to allow metals to automatically override dielectrics
- Set the Matrix/Permeability Computation method for 3D Magnetostatic Solutions
- Set the Inductance Computation method for 2D Magnetostatic Solutions
- Set the Matrix Computation to compute the inductance matrix for Transient solutions
- Adjust model and design Validations settings
- Set the Symmetry Multiplier for Transient and AC Magnetic solutions
- Set Preserve Transient Solution options (Transient solutions only)
- Set transient coupling with Twin Builder on the Advanced Product Coupling tab (Transient solutions only)
- Set the Model Settings (Maxwell 2D XY Transient, and 2D AC Magnetic solutions only)
- Set Skew Model options (Maxwell 2D XY Transient solutions with cylindrical rotational motion only)
- Set the default Background material (Maxwell 2D solutions only)
- Set Export Options (RMxprt only)

You can also view and set design settings by selecting the current design, and then in the **Properties** window, select the tab for the solver:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Set Material Threshold Tab

The material thresholds tell the **Maxwell 2D** and **Maxwell 3D** solvers how to deal with conductors and insulators. Materials with conductivities above the **Perfect Conductor** threshold are treated as having infinite conductivity and surface current only. Materials with conductivity values below the **Insulator/Conductor** threshold are treated as insulators with no conductivity and no current carrying capability. Materials that fall between the two thresholds are treated as normal conductors that can carry current throughout the volume of the material.

1. Click **Maxwell** > **Design Settings**. The **Design Settings** window appears.
   The material thresholds are set in the **Set Material Thresholds** tab. You can also view and set design settings by selecting the current design, and then in the **Properties** window, select the tab for the solver.
2. Type a value in the **Perfect Conductor** text box.
3. Type a value in the **Insulator/Conductor** text box.
4. Click **OK**.

RMxprt Material Thresholds:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **RMxprt** > **Design Settings**.
   The **Design Settings** window appears with the **Set Material Threshold** tab selected. You can also view and set design settings by selecting the current design, and then in the **Properties** window, select the tab for the solver.
2. Type a value in the **Conductivity Threshold** text box (Default=10,000).
3. Type a value in the **Permeability** text box (Default=100).

   > **Note:** RMxprt will treat materials with conductivity greater than 10,000 as conductors, and materials with Permeability greater than 100 as steels.

4. If you want these values to be the default, change the values by clicking the **Tools** > **Options** > **RMxprt Options** menu and setting the material thresholds in the **RMxprt Options** window.
5. Click **OK**.

Model Depth Settings

Maxwell 2D allows you to set the model depth for 2D magnetostatic, transient, and AC magnetic solutions that have the **Cartesian, XY** Geometry Mode. The model depth setting is retained when switching between magnetostatic and transient and eddy solutions.

To set the model depth:

1. Click **Maxwell** > **Design Settings**.

   Alternate method: In the Project Manager tree, right-click **<***Design_name***>**, and select **Design Settings**.

   The **Design Settings** window appears.

   > **Note:** You can also view and set design settings by selecting the current design, and then in the **Properties** window, select the tab for the solver.

2. On the **Model Depth** tab, enter the model depth value and select the desired unit of measure. The model depth value must be greater than zero. The default setting is 1 meter.

   > **Note:** Changing the model depth invalidates the solution.

3. Optionally, if you check **Save as default**, newly inserted designs will be set to use the specified model depth setting.
4. When finished changing settings, click **OK**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Set Material Override Tab

For a 3D Maxwell design, the Design Settings window includes a **Set Material Override** tab, which has the **Enable material override** check box. Normally, the modeler considers any intersection between 3D objects to be an error. But, if you check this option, the modeler allows a metal object to intersect a dielectric, and just gives a warning. Intersections between two metals or two dielectrics will still be errors.

In the mesher, the dielectrics are "subtracted" from the metals in the intersecting region. That is, the part of the dielectric that is inside the metal is removed, and if the dielectric is completely inside, the whole object disappears.

The purpose of this feature is to allow you to avoid doing explicit subtraction in the modeler. One example application is a via that passes through many dielectric layers – with the option turned on, the via does not have to be subtracted from the layers.

> **Warning:** Users must be careful: this setting changes the "ground rules" of the modeler and may have unexpected results.

Inductance Type Settings for 3D Magnetostatic Solutions

Maxwell 3D provides two computation methods for calculating the inductance of current-carrying coils for magnetostatic solutions: **Apparent** and **Incremental**.

- **Apparent** inductance is defined as the ratio of flux to current at a certain operating condition. Alternatively, this can be expressed as the slope of a line from the origin to the operating point on the B-H curve. Apparent inductance is commonly used in nonlinear magnetic applications and in circuit analysis, and is the default method used by Maxwell.
- **Incremental** inductance is defined as the ratio of delta flux to delta current at a certain operating condition. This is equivalent to the slope of the tangent to the B-H curve at the operating point. Incremental inductance is commonly used in physical design because it is easy to measure.

To set the computation method:

1. Click **Maxwell 3D** > **Design Settings**.

   Alternate method: In the Project Manager tree, right-click **<***Design_name***>**, and select **Design Settings**.

   The **Design Settings** window appears. You can also view and set design settings by selecting the current design, and then in the **Properties** window, select the tab for the solver.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. On the **Inductance Type** tab, select the desired method: **Apparent** or **Incremental**.

   > **Note:** If you change the computation method after solving, all matrix results of both AdaptivePass and LastAdaptive solutions will be invalidated. Then, on the next simulation, the matrix of the last solved pass will be recomputed with the new method. If there are matrix expression caches for the AdaptivePass solution, then the cached values of previous passes will not be available because only the last pass is resimulated.

3. Optionally, if you check **Save as default**, newly inserted designs will be set to use the specified computation method.
4. When finished changing settings, click **OK**.

Inductance Computation Tab Settings for 2D Magnetostatic Solutions

Maxwell 2D provides two **Inductance Computation** methods for calculating the inductance of current-carrying coils for magnetostatic solutions: **Apparent** and **Incremental**.

- **Apparent** inductance is defined as the ratio of flux to current at a certain operating condition. Alternatively, this can be expressed as the slope of a line from the origin to the operating point on the BH curve. Apparent inductance is commonly used in nonlinear magnetic applications and in circuit analysis, and is the default method used by Maxwell.
- **Incremental** inductance is defined as the ratio of delta flux to delta current at a certain operating condition. This is equivalent to the slope of the tangent to the BH curve at the operating point. Incremental inductance is commonly used in physical design because it is easy to measure.

To set the matrix computation method:

1. Click **Maxwell 2D** > **Design Settings**.

   Alternate method: In the Project Manager tree, right-click **<***Design_name***>**, and select **Design Settings**.

   The **Design Settings** window appears. You can also view and set design settings by selecting the current design, and then in the **Properties** window, select the tab for the solver.

2. On the **Inductance Computation** tab, select the desired method: **Apparent** or **Incremental**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** If you change the computation method after solving, all matrix results of both AdaptivePass and LastAdaptive solutions will be invalidated. Then, on the next simulation, the matrix of the last solved pass will be re-computed with the new method. If there are matrix expression caches for the AdaptivePass solution, then the cached values of previous passes will not be available because only the last pass is re-simulated.

3. Optionally, if you check Save as default, newly inserted designs will be set to use the specified computation method.
4. When finished changing settings, click **OK**.

Matrix Computation Tab Settings for Transient Solutions

Maxwell 2D/3D provides an option on the **Matrix Computation** tab for computing the inductance matrix for transient solutions. When the **Compute inductance matrix** option is selected, the computed matrix quantities will be available at all timesteps. The matrix quantities are listed in the reporter and thus are available for postprocessing, output variables creation, and optimetrics studies.

> **Note:**
> During the inductance calculation in the transient solver:
> - Eddy effects for all objects are turned off even if **Eddy Effect** is checked in the Set Eddy Effects window. However, the field solution and all other results will consider the eddy effects as set in the interface.
> - For a field system with multiple stranded windings, after all nonlinear materials are frozen, one winding is supplied with 1 A DC current while the other winding is open (all other sources are turned off).

The size of the computed inductance matrix is equal to the number of winding setups, that is, the matrix will be N x N when there are N winding setups; and the total number of inductance-related quantities will be N^2. Components of the full matrix are available for plots and optimetrics calculation even though the matrix will always be symmetric. You can choose to access only the upper right half of the matrix by selecting the corresponding quantities as listed in the reporter dialog box. For example, the upper half of a 2 x 2 matrix will be the L11, L12, and L22 quantities.

Maxwell 2D/3D provides two **Matrix Computation** methods for calculating the inductance of windings for transient solutions: **Apparent** and **Incremental**.

- **Apparent** inductance is defined as the ratio of flux to current at a certain operating condition. Alternatively, this can be expressed as the slope of a line from the origin to the operating point on the BH curve. Apparent inductance is commonly used in nonlinear magnetic applications and in circuit analysis, and is the default method used by Maxwell.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Incremental** inductance is defined as the ratio of delta flux to delta current at a certain operating condition. This is equivalent to the slope of the tangent to the BH curve at the operating point. Incremental inductance is commonly used in physical design because it is easy to measure.

To enable the matrix computation for transient solutions:

1. Click **Maxwell** > **Design Settings**.
   - Alternate method: In the Project Manager tree, right-click **<***Design_name***>**, and select **Design Settings**.

   The **Design Settings** window appears. You can also view and set design settings by selecting the current design, and then in the **Properties** window, select the tab for the solver.

2. On the **Matrix Computation** tab, select **Enable**, and select the desired method: **Apparent** or **Incremental**.

   Note:
   - The default is NOT to compute the inductance matrix. Changing the setting for an existing design does not affect the solution for the previous simulation. For example, if a transient solve setup has 1 second set as the end time and is solved with the **Enable** option off (unchecked) and a plot of matrix quantities is created vs. time, the plot will not show any values because no matrix quantities were computed. Subsequently, if the compute option is enabled (checked) and the end time extended to 2 seconds, after simulation, matrix quantities will be available from > 1 second to 2 seconds, at all solved timesteps.
   - If you change the computation method after solving, all matrix results of both AdaptivePass and LastAdaptive solutions will be invalidated. Then, on the next simulation, the matrix of the last solved pass will be recomputed with the new method. If there are matrix expression caches for the AdaptivePass solution, then the cached values of previous passes will not be available because only the last pass is re-simulated.

3. Optionally, if you check **Save as default**, newly inserted designs will use the specified setting.
4. When finished changing settings, click **OK**.

Validations Tab

Model Settings

You can adjust the degree to which the software checks a model for faults that could jeopardize mesh accuracy. There are four levels of model validation that a user can specify for a given

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

design: **Warning Only**, **Basic**, **Strict**, and **None**. Note that this setting affects only the "3D Model" stage of a design validation. The default Entity Check Level is **Strict**.

- The **Warning Only** entity check setting allows all models to pass 3D Model validation regardless of any faults that are found. These faults are posted in the message window as warnings.
- The **Basic** entity check setting allows most models to pass 3D Model validation. Some faults are flagged as model errors, thereby prohibiting a design from proceeding to the meshing stage of an analysis. You must correct such errors before attempting to analyze the design under the Basic setting, or you must change the entity check level to Warning Only.
- The **Strict** entity check setting enforces a tighter tolerance for model faults than the Warning Only and Basic settings. All model faults that are found during 3D Model validation are posted to the message window. These errors must be corrected before attempting to analyze the design under the Strict setting, or you must change the entity check level to Basic or Warning Only.
- The **None** setting disables model validation.

In addition, check boxes allow you to adjust model validation to

- **Ignore Unclassified Objects**
- **Skip Intersection Checks**

These settings are unchecked by default.

Maxwell 2D and Maxwell 3D Design Settings

For both 2D and 3D designs, you can choose either:

- **Perform full validations** (the default setting) – All design validations are performed.
- **Perform minimal validations** – All design validations except boundary overlap validation are performed.

For 3D designs only, you can select **Skip mesh checks** to prevent the solver from failing validation when the mesh has a few elements of poor quality.

Adjusting Settings

To adjust settings on the **Validations** tab:

1. Select **Maxwell** > **Design Settings** to open the **Design Settings** dialog box; then select the **Validations** tab.
2. Choose the desired level of Model validation from the **Entity Check Level** drop-down menu.
3. Optionally, adjust Model validation to **Ignore Unclassified Objects**, or to **Skip Intersection Checks**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Optionally, select **Perform minimal validations** if you want to omit boundary overlap validation checking.
5. Optionally, **Skip mesh checks** to prevent the solver from failing validation when the mesh has a few elements of poor quality.
6. Optionally, click the **Save as Default** button to make the current selections the default settings.
7. Click **OK** to accept the selections and close the dialog.

> **Note:** You can also view and set design settings by selecting the current design, and then in the **Properties** window, select the tab for the solver.

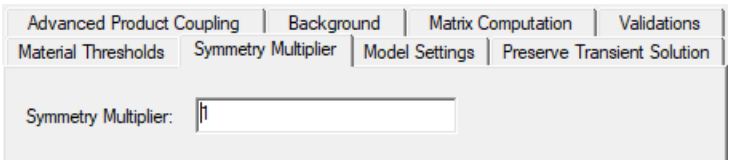
Setting a Symmetry Multiplier

(Transient and AC Magnetic Solutions Only)

If you are using symmetry and/or independent/dependent boundary conditions in your solution, you may need to specify a multiplier (so that the other part of the model, which is not included in the solution domain, can be properly taken into account). This symmetry multiplier will be automatically applied to all input quantities, including input voltage, inductance, resistance, load torque, mass, damping, external circuit. The symmetry multiplier will be automatically applied to the following output quantities: induced voltages, flux linkages in every winding, stranded loss, solid loss, core loss, and torque. For the transient solver, the symmetry multiplier will also be applied to the force output.

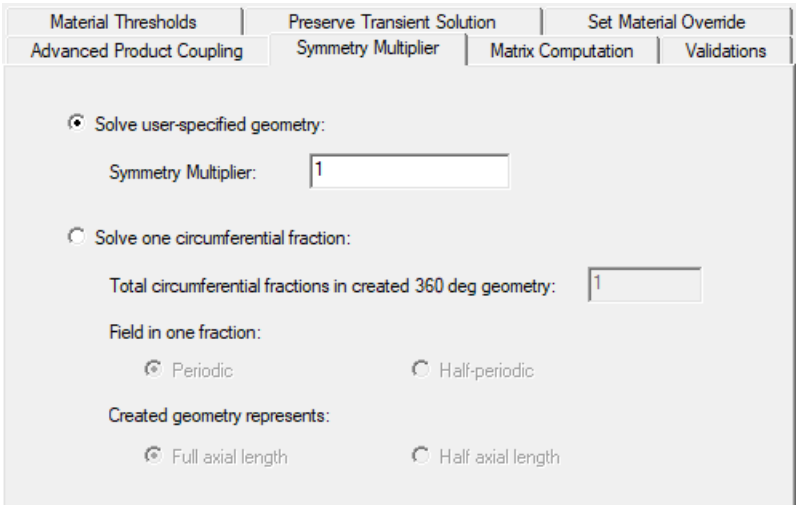
To specify a multiplier:

1. Select **Maxwell** > **Design Settings** from the main menu to open the Design Settings dialog box, then select the **Symmetry Multiplier** tab. You can also right-click **Model** in the Project Manager tree, and select **Set Symmetry Multiplier**; or select **Maxwell** > **Model** > **Set Symmetry Multiplier**, to open the Design Settings dialog to the **Symmetry Multiplier** tab. You can also view and set design settings by selecting the current design, and then in the **Properties** window, select the tab for the solver.
2. Depending on the design type, do one of the following:
   - For 2D and 3D AC Magnetic, and 2D Transient designs, enter an integer value (or a variable that evaluates to an integer value) in the **Symmetry Multiplier** text box, and click **OK**.

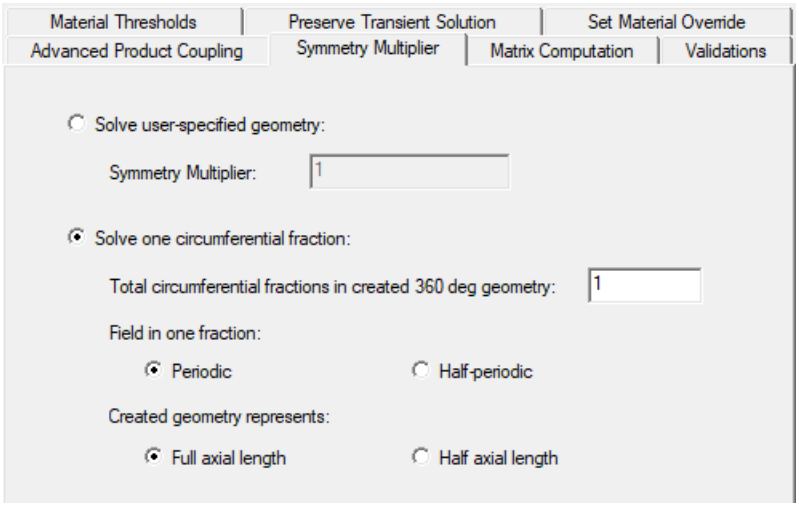


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For 3D Transient designs, you can select either:
  - **Solve user-specified geometry** and enter an integer value (or a variable that evaluates to an integer value) in the **Symmetry Multiplier** text box, then click **OK**.



  - **Solve one circumferential fraction**, then enter an integer value for the **Total circumferential fractions in created 360 deg geometry**.



    Select either **Full axial length** or **Half axial length** for **Created geometry represents**.

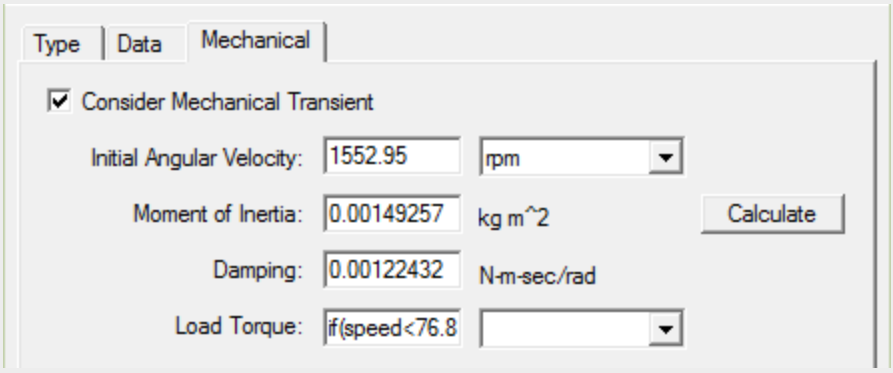
    Select either **Periodic** or **Half-periodic** for **Field in one fraction**.

  - Click **OK**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

When a symmetry multiplier has been set, the parameter values (moment of inertia, damping, load torque) specified on the **Motion Setup** window's **Mechanical** tab are related to the full model.



Export Options Tab

You can also view and set design settings by selecting the current design, and then in the **Properties** window, select the tab for the solver.

To set the Export options for RMxprt, choose which options to use when exporting a design from RMxprt to Maxwell 3D:

- **Periodic**
- **Difference**
- **Band Arc**
- **Teeth-Teeth**
- **Segmented Arc**

Note:

These options may also be set on the **Export Options** tab of the RMxprt Options window. Using the **Tools** > **Options** > **RMxprt Options** command changes the default for the current design and all future designs.

You may also set the default **Design Sheet** for use with RMxprt by entering the path and filename or by browsing to the Excel file using the **ellipsis** button.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

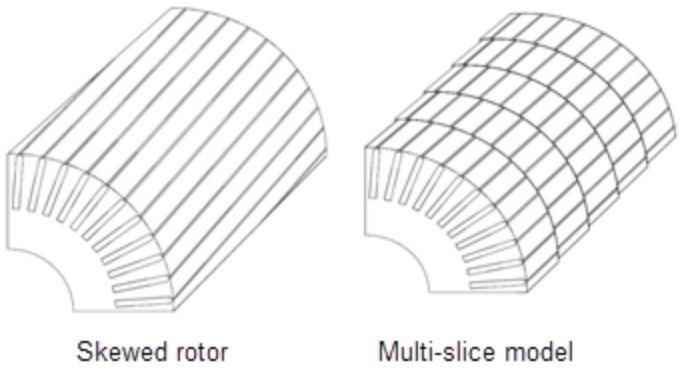
Model Settings Tab

(Maxwell 2D XY Transient and 2D AC Magnetic Designs Only)

For Maxwell 2D XY transient and 2D AC magnetic designs, torque, flux linkage, back EMF, *etc.* are scaled by the length provided in the **Model Depth** field on the **Model Settings** tab. The scaled results are used to calculate output quantities.

(Maxwell 2D XY Transient solutions with cylindrical rotational motion only)

For Maxwell 2D transient designs with cylindrical rotational motion setup, the transient solver can take into account the effects of skewing by using a multi-slice model. It is based on creating multiple 2D models, the model depth of which represents a fraction of the total model length. These 2D models are often referred to as *slices*.



When the skew model is active, the initial angle of each slice is internally offset in such a way that the defined skew angle results.

The same mathematical approach is used to consider rotor and stator skewing. This will only affect the direction of the angle offset in field plots.

The electric degrees of freedom of the multiple 2D models are coupled during the calculation, ensuring that the currents flowing through the conductors of each slice are consistent.

In postprocessing, the integral results of each slice (*e.g.*, torque, losses, flux linkage, induced voltages) are calculated by summation of each slice's result. The torque, loss, and flux linkage results are calculated using the following equations:

$$Torque: \quad T = \sum_{m=0}^{sn} \frac{L^m}{L} T^m$$

$$Loss: \quad Loss = \sum_{m=0}^{sn} \frac{L^m}{L} Loss^m$$

$$Flux\ Linkage: \quad \phi = \frac{1}{S} \sum_{m=0}^{sn} \frac{L^m}{L} \phi^m$$

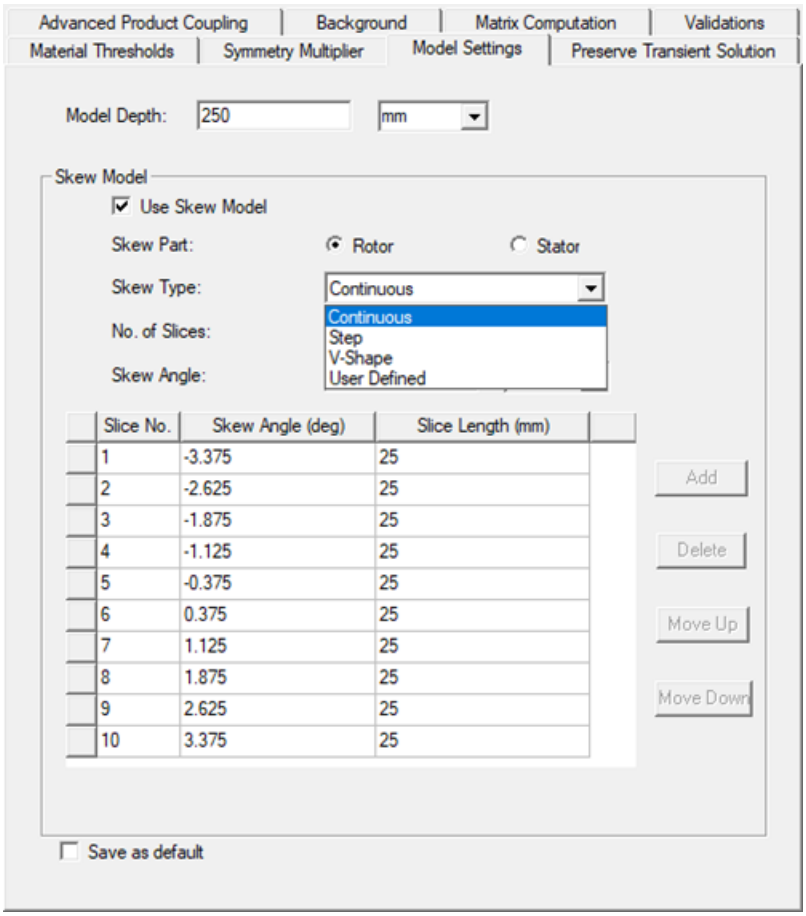
where *sn* is the total slice number, *L* is the model depth, and L^m is the slice length.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Maxwell 2D transient designs that use the skew model can use MPI HPC for distributed computing of multiple slices.

To enable use of the multi-slice skew model:

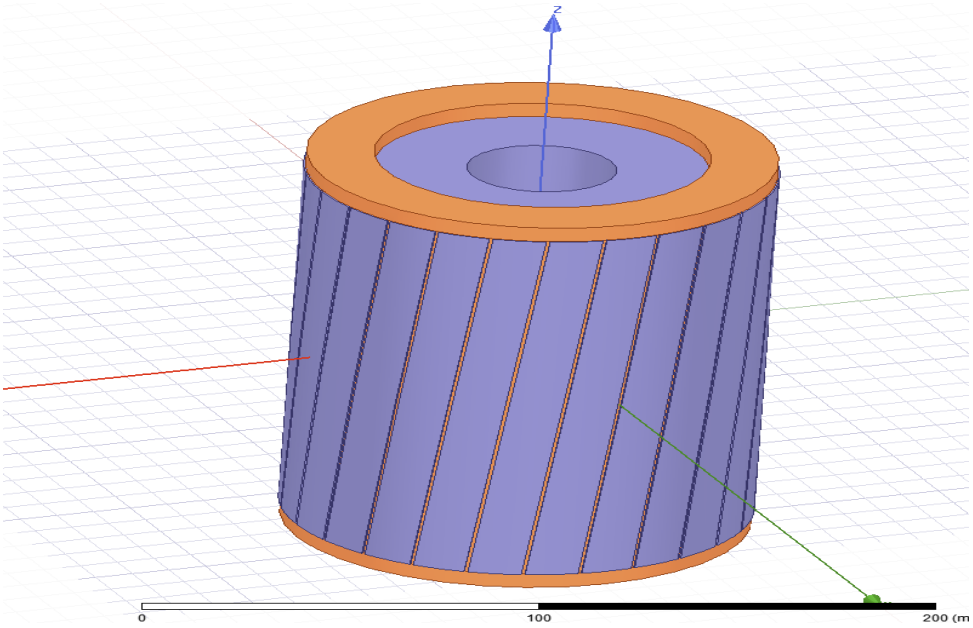
1. Right-click on **Model** in the Project Manager tree and select **Set Skew Model** on the context menu to open the **Design Settings** window to the **Model Settings** tab. You can also click **Maxwell 2D** > **Design Settings** to open the **Design Settings** window, then select the **Model Settings** tab. You can also view and set design settings by selecting the current design, and then in the **Properties** window, select the tab for the solver.
2. Check the **Use Skew Model** check box to enable the parameter fields below it.



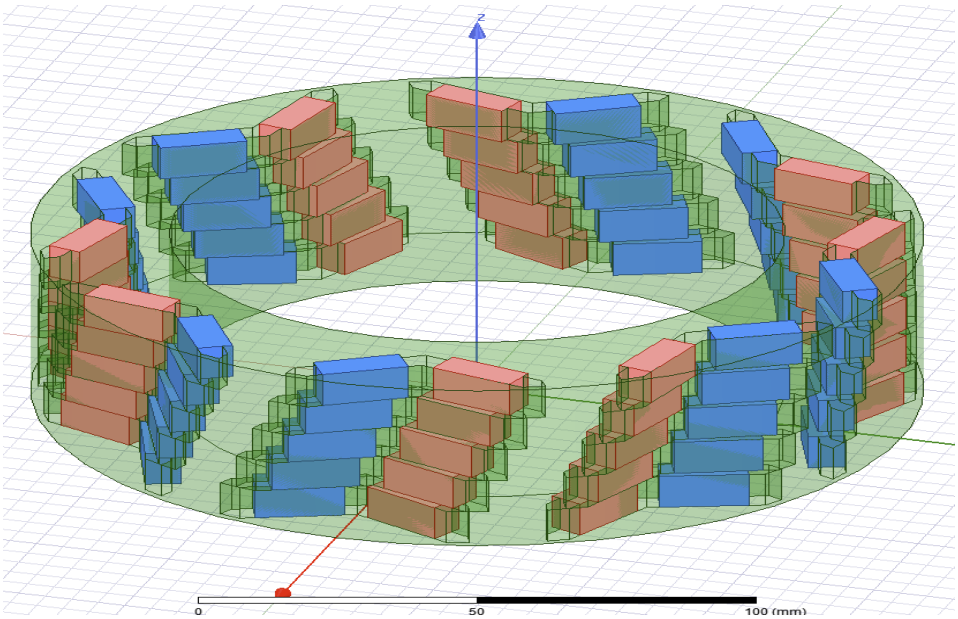
3. Select either **Rotor** or **Stator** as the **Skew Part** being modeled. Default is **Rotor**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Select a **Skew Type**.
    - **Continuous** (the default) - usually used in induction machines

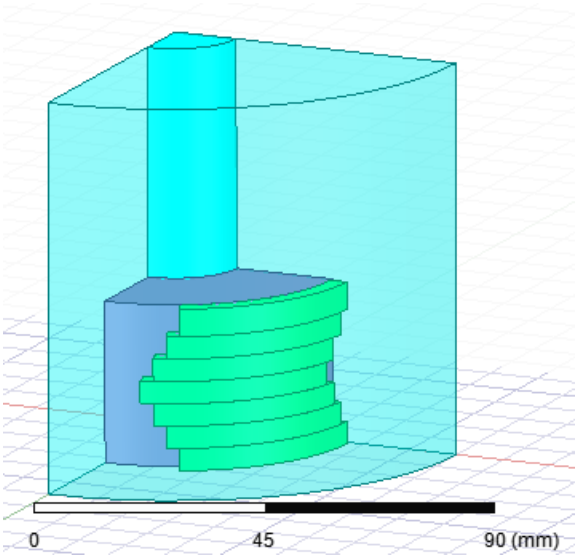


    - **Step** - usually used in PM machines

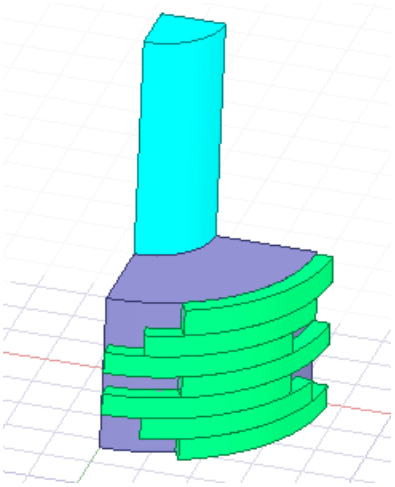


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **V-Shape**



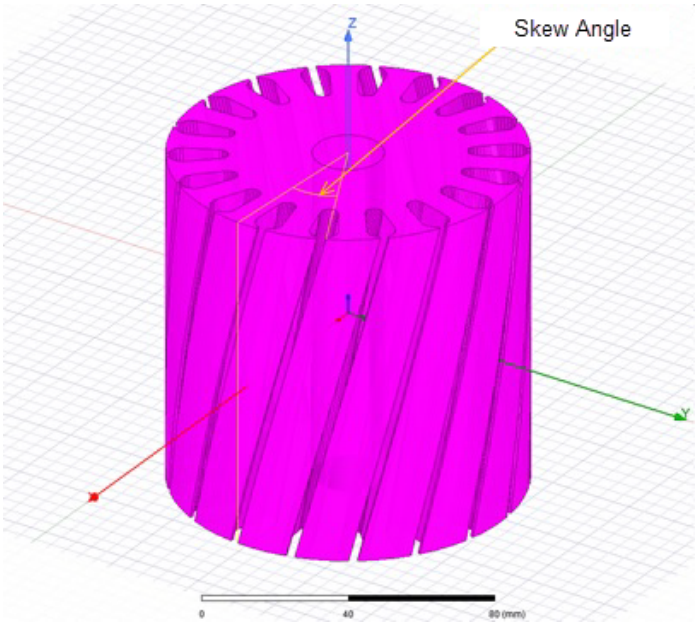
- **User Defined**



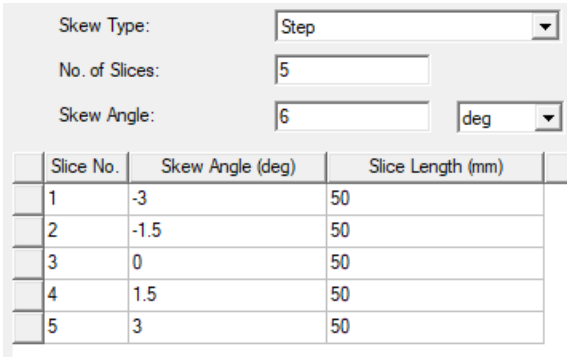
5. For **Continuous**, **Step**, and **V-Shape** skew types:
    a. Enter the **No. of Slices** that you want the model divided into in the Z-direction. The input must be a positive integer value greater than zero. The default is 5. You can also enter a variable name. (**No. of Slices** is not used for the **User Defined** skew type and is grayed out.)
    b. Enter the **Skew Angle** and its unit. The Skew Angle represents the mechanical angle between the two axial ends of the stator/rotor core's active length (not between two adjacent slices). Valid entries are non-negative real numbers in the range of 0 to 180 degrees inclusive (or equivalent values in units of measure such

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

as radians). The default value is 5 degrees. You can also enter a variable name.



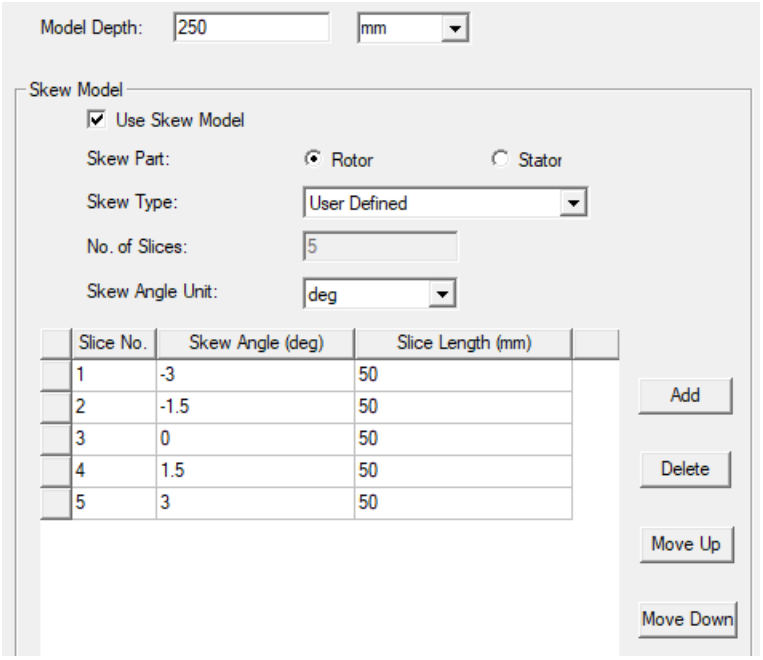
The Skew Angle and Slice Length values for each slice are calculated by the modeler and are listed in the (non-editable) table.



Slice No.	Skew Angle (deg)	Slice Length (mm)
1	-3	50
2	-1.5	50
3	0	50
4	1.5	50
5	3	50

6. For the **User Defined** skew type:
    a. Select a **Skew Angle Unit**, which will be applied to the Skew Angle values you enter in the table.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



b. Enter the desired **Skew Angle** and **Slice Length** values for the selected skew part. If the total of the **Slice Length** values differs from the **Model Depth** value, a message informing you of the difference will display in the Message Manager. **Add**, **Delete**, **Move Up**, and **Move Down** buttons allow you to control the table row entries.

7. When finished, click **OK** to close the window.

User-Defined Skew Type Computations

The phase angle of flux-linkage in a user-defined skew part will change according to the slice skew angle.

For a user-defined skew type, the solver will rotate all the slices at an additional angle β in order to keep flux-linkage with the same initial phase angle as that of the unskewed case.

The additional angle β can be calculated by

$$\beta = -\frac{1}{L}\sum_{i=1}^{i=N} \alpha_i * L_i$$

where

- β is the additional rotate angle for all the slices
- N is the slice number
- α_i is the ith slice skew angle which user defines

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- L_i is the ith slice length which user defines
- $L = \sum_{i=1}^{i=N} L_i$ is the model depth.

Thus, the slice skew angle for each slice will be $\alpha_i + \beta$.

Background Material Tab

(Maxwell 2D Designs Only)

The area of a model not enclosed within specific objects is termed the background. The material assigned to this area is vacuum by default.

To change the default setting:

1. Click **Maxwell 2D** > **Design Settings**.

   The Design Settings window is displayed. You can also view and set design settings by selecting the current design, and then in the **Properties** window, select the tab for the solver.

2. Select the **Background** tab.
3. Click the **Background** button listing the current material setting.

   The **Select Definition** window is displayed with the Material Database.

4. Locate the material desired and click on the **Name** in the table.
5. Click **OK** to return to the **Design Settings** window.
6. The **Background** button will now list the newly selected material indicating that it will be used as the background material.
7. Click **OK** to finalize the change and dismiss the window.

Preserve Transient Solution Tab

For **Transient** solutions, you have the option of enabling the **Preserve solution after dataset edits** function on this tab.

Datasets may be used to describe material and source behavior in Maxwell. Enabling the **Preserve solution after dataset edits** function allows source, material, or other parameters that depend upon datasets to be modified without deleting the transient solution data. The transient solver may then be restarted from the final timestep of the previous simulation.

You can also view and set design settings by selecting the current design, and then in the **Properties** window, select the tab for the solver.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Advanced Product Coupling Tab

For Transient solutions, this tab provides the option to **Enable transient-transient link with Twin Builder** components. You can also view and set design settings by selecting the current design, and then in the **Properties** window, select the tab for the solver.

Maxwell designs may be dynamically coupled to Twin Builder components through the Twin Builder user interface. The Twin Builder interface allows 2D, 3D, and RMxprt designs to be linked as dynamic components. Source windings set to **External** are available as conservative pins in Twin Builder.

Dynamically linked components provide the ability to have Twin Builder vary a variable in a Maxwell design and have Maxwell solve the design at the new design point and provide the solution data back to Twin Builder for use in the circuit simulation.

For more information, refer to the Twin Builder Help.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5 - Specifying the Solver Type

Before you begin to draw the models in your project, choose which field solver to use for it. Each solver requires a different type of problem setup. If you later change the solver, all problem setups become invalid, and all solutions are deleted. Because of this, it is a good idea to decide on a solver before starting.

To specify a solver:

1. Click **Maxwell 2D** > **Solution Type** or **Maxwell 3D** > **Solution Type**.
   The **Solution Type** window appears with the solvers generally split between **Magnetic** and **Electric** solver types.
2. For Maxwell 3D designs, select one of the following solver types:
   - **Magnetostatic**
   - **AC Magnetic (Eddy Current)** (T-Omega or A-Phi)
     - **Include DC Fields** (T-Omega only)
   - **Transient** (T-Omega or A-Phi)
   - **Electrostatic**
   - **3D AC Conduction**
   - **DC Conduction** either with or without Insulator Field
   - **Electric Transient**
3. For Maxwell 2D designs, select one of the following solver types:
   - **Magnetostatic**
   - **AC Magnetic (Eddy Current)**
   - **Transient**
   - **Electrostatic**
   - **2D AC Conduction**
   - **DC Conduction**

   > **Note:** For Maxwell 2D designs, you must also specify the geometry mode: **Cartesian, XY** or **Cylindrical about Z**.

4. Click **OK**.

> **Note:** For information on the various material property types supported by each solver, refer to Maxwell Solvers Material Properties.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Magnetostatic Solver

The magnetostatic field simulator computes static magnetic fields in 2D or 3D. The source of the static magnetic field can be

- DC currents in conductors
- static external magnetic fields represented by boundary conditions
- permanent magnets

The quantities for which the magnetostatic field simulator solves are the magnetic field, **H**, and the current distribution, **J**; the magnetic flux density, **B**, is automatically calculated from the **H**-field. Derived quantities such as forces, torques, energy, and inductance may be calculated from these basic field quantities.

AC Magnetic (Eddy Current) Solver

The AC Magnetic (Eddy Current) field simulator computes electromagnetic fields in the frequency domain in 2D or 3D. The sources of the electromagnetic field can be

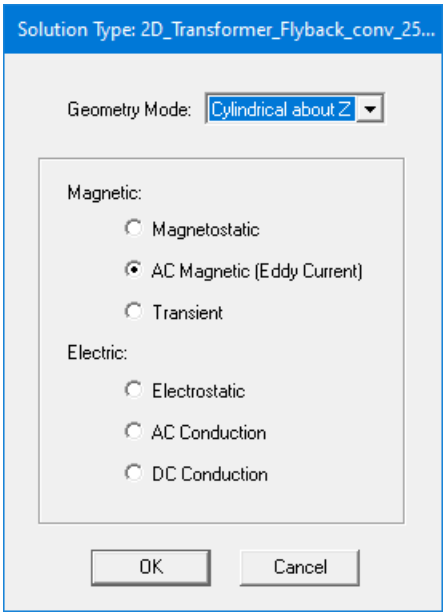
- AC currents in conductors.
- Time-harmonic external magnetic fields represented by boundary conditions.

The quantity for which the AC magnetic field simulator solves is the magnetic field, **H**. The magnetic flux density, **B**, is automatically calculated from the **H**-field.

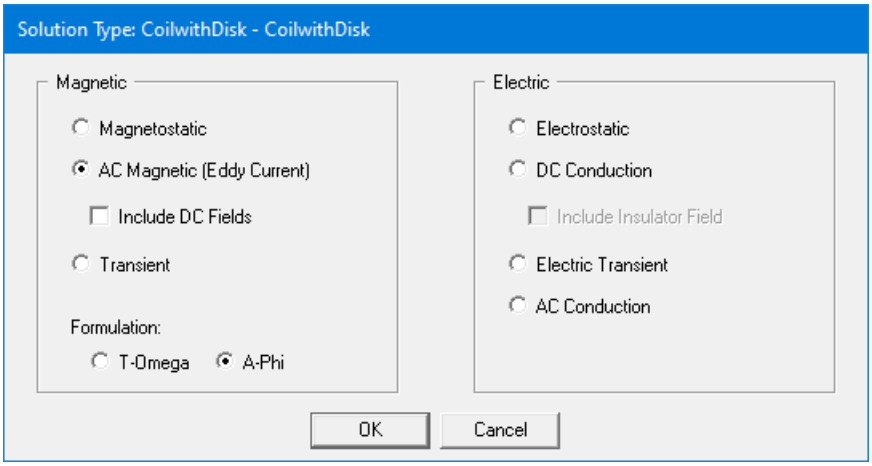
Optionally, the AC Magnetic solver can incorporate displacement currents into the solution even in non-conducting domains, thereby allowing the solution of the full system of Maxwell's equations and the recovery of the electric field **E** everywhere including the non-conducting regions. Derived quantities such as forces, torques, energy, losses, and impedances may be calculated from these basic field quantities at the solution frequencies.

In 2D, only the A-Phi formulation is available, so even though the 2D **AC Magnetic (Eddy Current)** option is not labeled so, it uses the A-Phi formulation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



However, in 3D, both T-Omega and A-Phi formulations are available and can be selected from the Solver Type window:



> **Note:** The AC Magnetic 3D A-Phi solver is only accessible if it has been activated from **Tools** > **Options** > **General Options** > **Desktop Configuration** > **Beta Options** window. Select the **Maxwell AC Magnetic A-Phi Solution** feature.

For 3D AC Magnetic T-Omega solutions, user can **include DC fields** in the calculation; see Including DC Fields in a 3D AC Magnetic T-Omega Solution.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

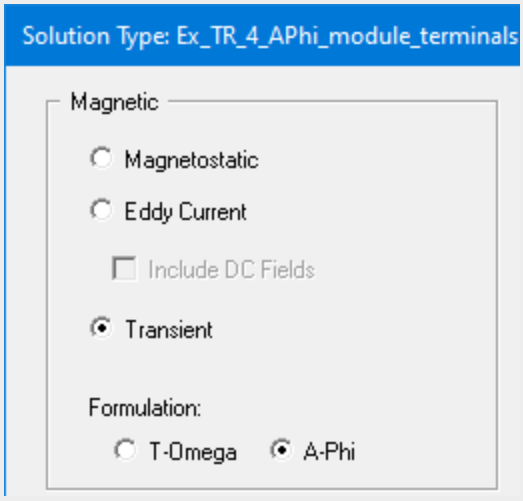
Transient Solver

The transient field simulator computes the time-domain magnetic fields in 2D or 3D. The source of the magnetic fields can be

- moving or non-moving time-varying currents and voltages.
- moving or non-moving permanent magnets and/or coils.
- moving or non-moving external circuit coupling.

The quantities for which the transient field simulator solves are the magnetic field, **H**, and the current distribution, **J**; the magnetic flux density, **B**, is automatically calculated from the H-field. Derived quantities such as forces, torques, energy, speed, position, winding flux linkage, and winding induced voltage may be calculated from these basic field quantities.

Note: If **A-Phi Formulation** is selected as the 3D Transient solver, refer to **Maxwell 3D Transient Solution Based on A-Phi Formulation** for details.



Maxwell 3D Transient A-Phi Solution

By default, the Maxwell 3D transient solver is based on T-Omega (Ω) formulation, which is a powerful method for solving a wide range of low-frequency EM problems. However, there are some areas where the applicability of the method is limited – such as multiple (mixed) source excitations on a single conduction path and capacitive effect (displacement current). For simulation of such problems, the Maxwell 3D Transient A-Phi (Φ) formulation is a more suitable solution type.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Comparison of T-Omega and A-Phi Solvers

T-Omega	A-Phi
Solves second-order elements for magnetic B field	Solves second-order Φ for electrical E field, and solves first-order A for magnetic B field. (In order to account the difference in the order of elements solved, increasing the mesh density in A-Phi should help achieve the same B field results as T-Omega.)
Computational efficient for electric machines applications	Computational efficient and flexible for ECAD PCB and electronics applications
Does not support multi-terminals with mixed excitation types on the same conduction path	Supports multi-terminals with mixed excitation types on the same conduction path
Ignores displacement current	Can consider capacitive effects (displacement current)
Supports all advanced material modeling	Limited advanced material modeling capabilities
Easy handling of motion due to only scalar potential for the motion coupling	Does not support motion

For more information on the Transient A-Phi formulation, refer to the following sections:

- Limitations
- Boundaries and Excitations
- Rules/Limitations of Excitations on Conduction paths
- Matrix Setup
- Postprocessing Support
- 3D Layout Component with an A-Phi Solver

Limitations

The following 3D Transient features is not supported/available for this release:

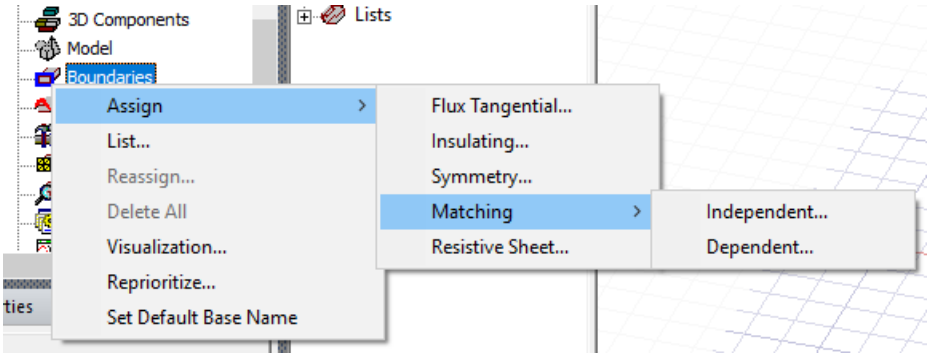
- Motion-related features

Boundaries and Excitations

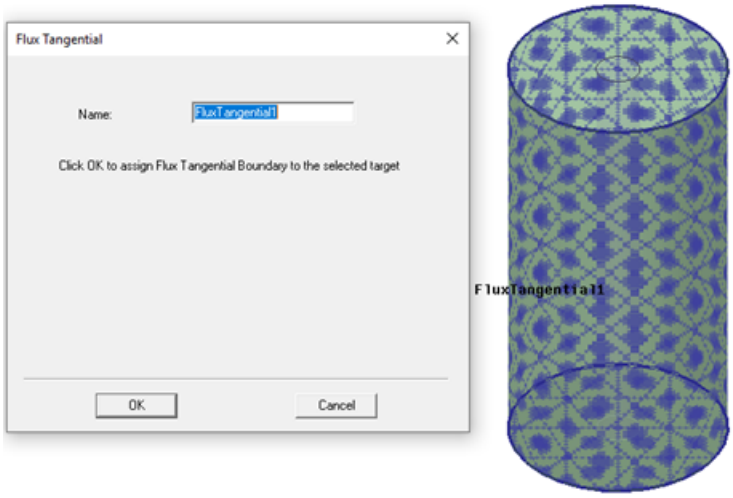
Boundaries and Excitations supported are as follows:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Boundaries supported are **Flux Tangential**, **Insulating**, **Symmetry**, **Independent** and **Dependent Matching**, and **Resistive Sheet**.

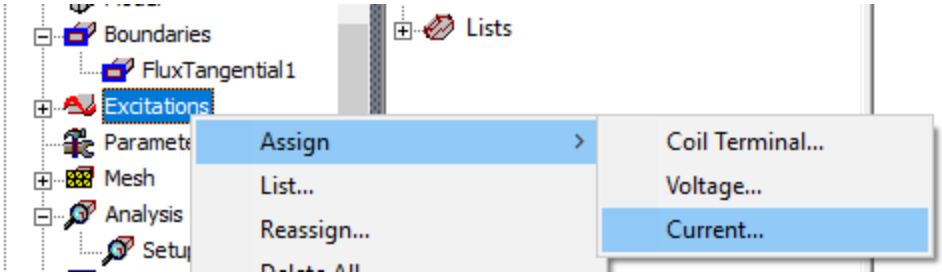


Flux Tangential boundaries can only be applied to face objects. Unlike the T-Omega formulation, the Neumann boundary condition on external faces is flux normal for A-Phi formulation. That is why a Flux Tangential boundary condition must be assigned when the magnetic field is tangential to the face.



- Excitations supported are **Winding** (via **Coil Terminal**), **Voltage**, and **Current**. Winding excitation is assigned as in the Transient solver (Refer to *Coil Terminals* for more information on winding assignments). **Voltage** and **Current** excitations are also included on the **Excitations** > **Assign** menu:

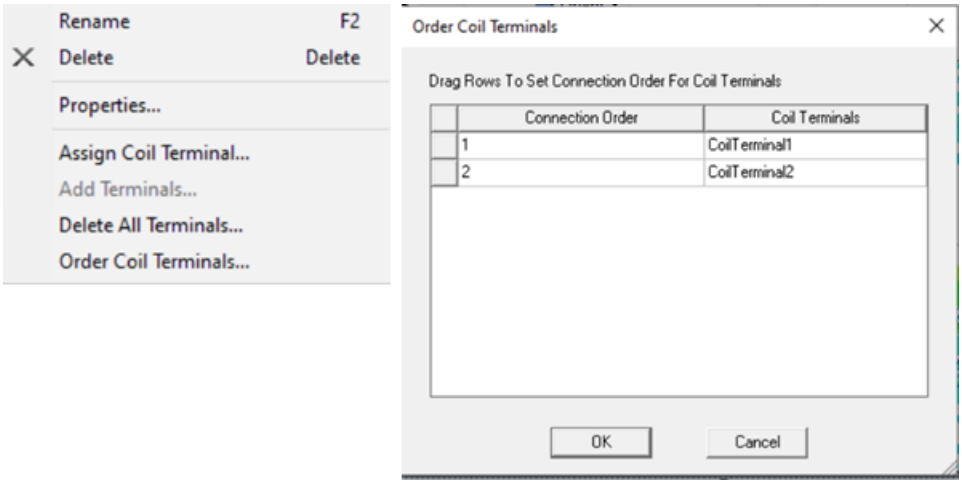
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Winding Excitations for A-Phi Formulation

Winding excitation is assigned as in the Transient solver (Refer to Assigning a Winding Setup for a Transient Solver for more information on winding assignments). For winding excitations, the coil terminals must be ordered according to the physical potential distribution. To do this:

1. Right-click on the winding to open the context menu.
2. Click on **Order Coil Terminals** to open the **Order Coil Terminals** dialog box.



3. Order the terminals by dragging them in the **Order Coil Terminals** dialog box.

Coil Order

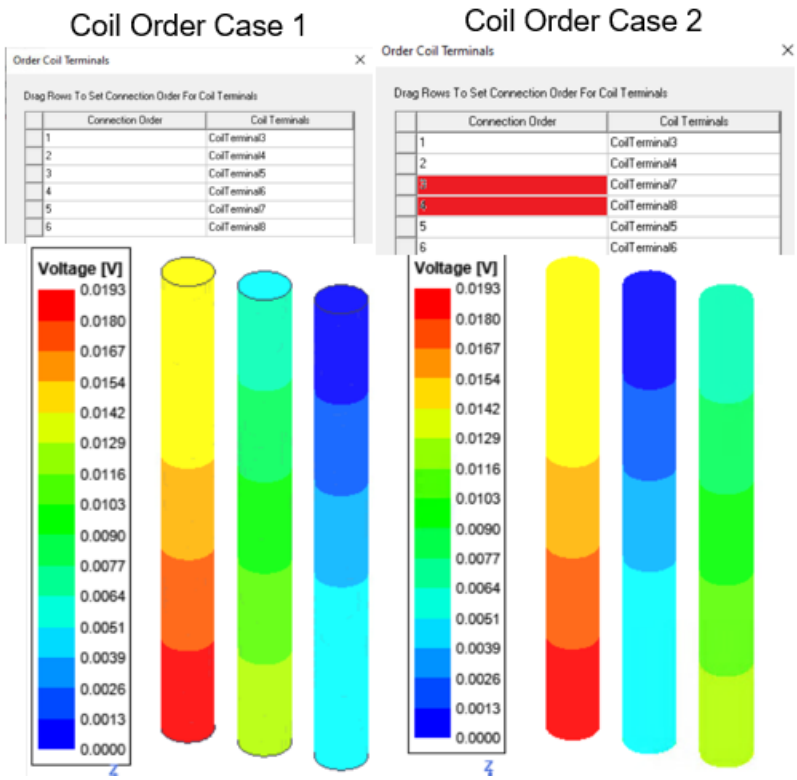
The solver connects the coils according to the coil order specified via **Order Coil Terminals**. Voltage/current values are assigned to the first terminal, and the last terminal is grounded. To generate the correct potential plot, the terminals should be ordered according to the actual connection of the conductors.

A winding with coil terminals assigned to the external surfaces of the conductors are shown below. The coil terminals are labeled with numbers.

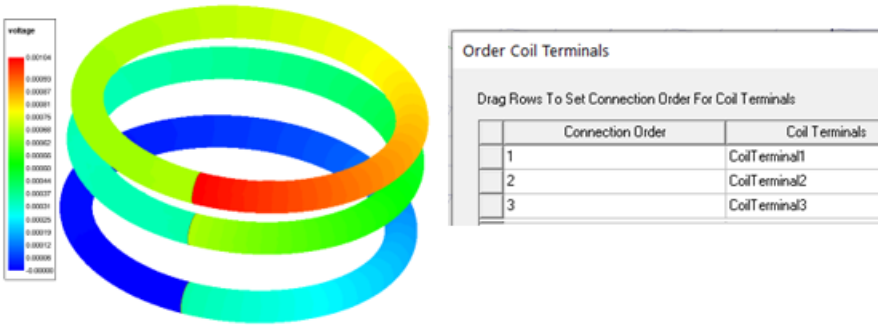
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If a winding has more than three external terminals, as in this example, the solver makes the consecutive terminals (except first and last terminal) equipotential (or connected). For example, in Coil Order Case 1, terminals (4,5) and (6,7) have the same potentials . Coil Order Case 2 shows the effect of reordering the coil terminals. Even though Coil Order 1 and 2 have the same terminals, the different order of the terminals gives different potentials, but still the same B, H, and J fields.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For coil terminals defined inside the conductors, the solver defines two potentials for each terminal, and the same connection rule described above applies considering this double potential definition. See "Rules/Limitations of Excitations on Conduction Paths" on page 5-14 for further information about winding excitations.



Voltage Excitations for A-Phi Formulation

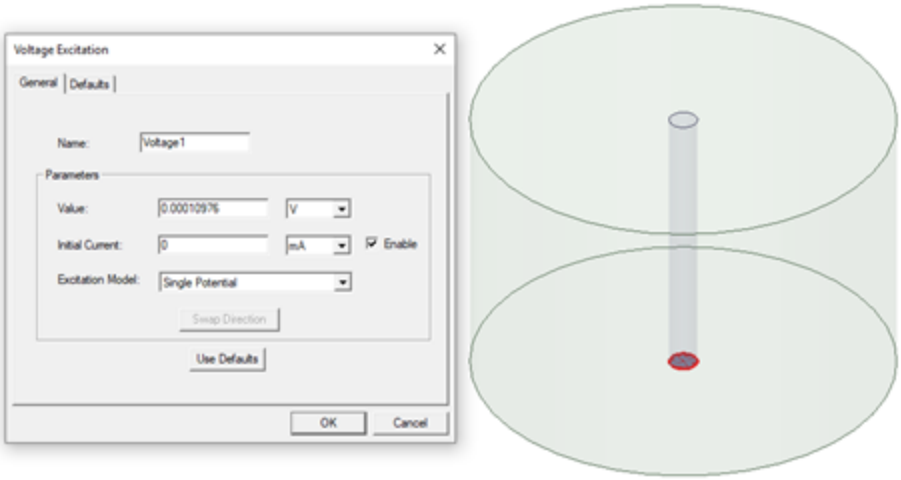
For designs with known potentials, select and assign a voltage excitation on each face.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also define initial currents for Voltage Excitation by checking the **Initial Current** box. When initial current is defined, the solver treats it as the current excitation. The default value is 0 A. You should select at least one voltage excitation without enabling the initial currents on a solid conduction path to set a reference potential. (If the path is touching the odd symmetry boundary, where the potential is set to 0 V, or if displacement effect is considered in the problem domain, then this rule does not apply on this path.)

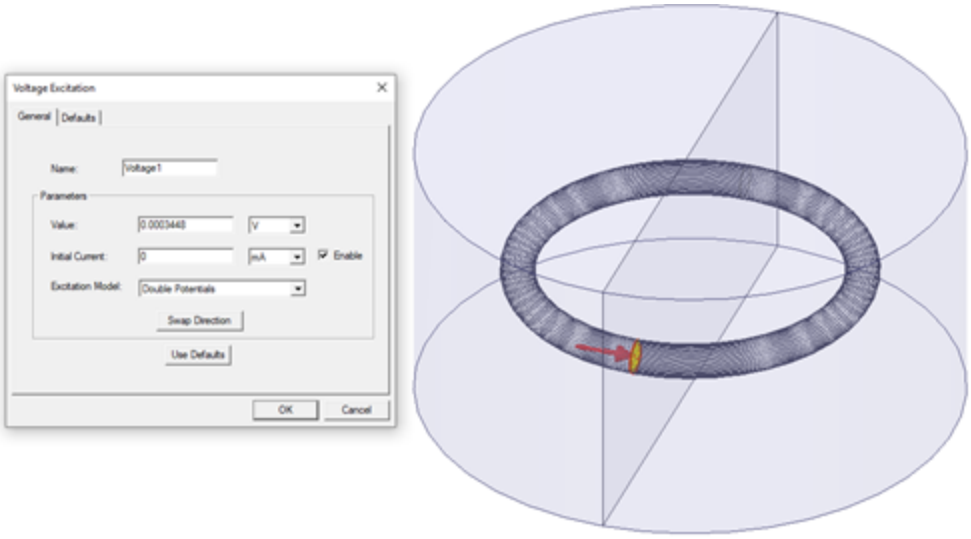
Voltage excitation supports three different excitation models:

- **Single Potential**: The solver treats the terminal as an equipotential terminal. Only one scalar electric potential DOF (Degree of Freedom) is assigned to the surface, and the value of the DOF is set to the value that you input. *This model must be used only at the external boundaries of the problem domain.*

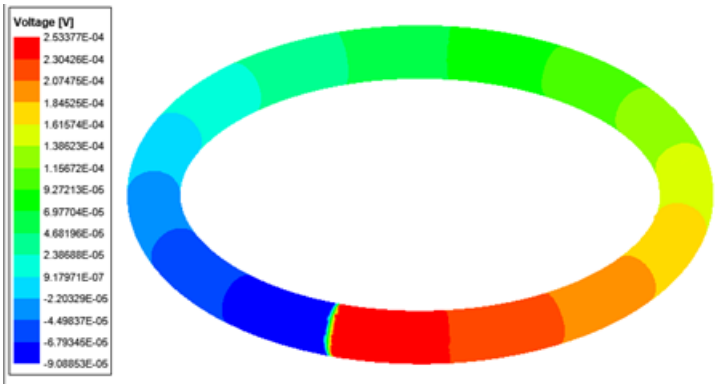


- **Double Potentials**: The solver treats the terminal as a voltage (potential difference) terminal. At the terminal face, two scalar electric potential DOFs are defined and the DOFs are unknown. The difference between those potentials equals the voltage value that you specified. If the voltage value is a positive quantity, higher potential is in the specified direction. *This model must be used on faces inside the problem domain.*

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

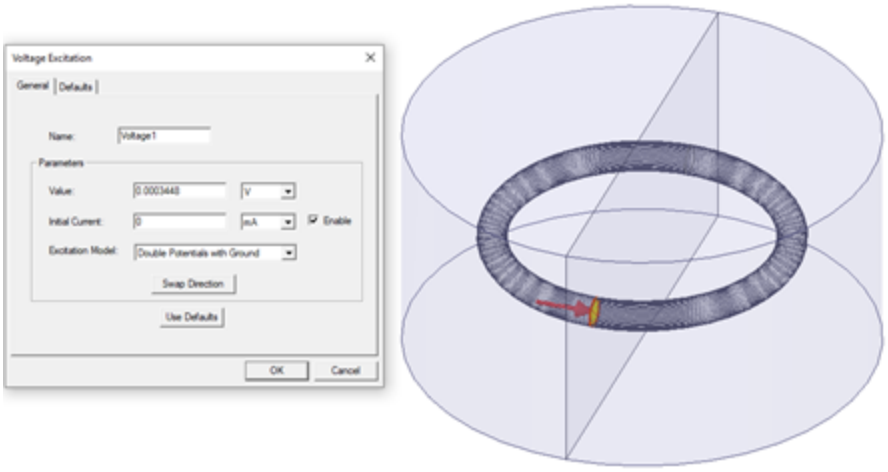


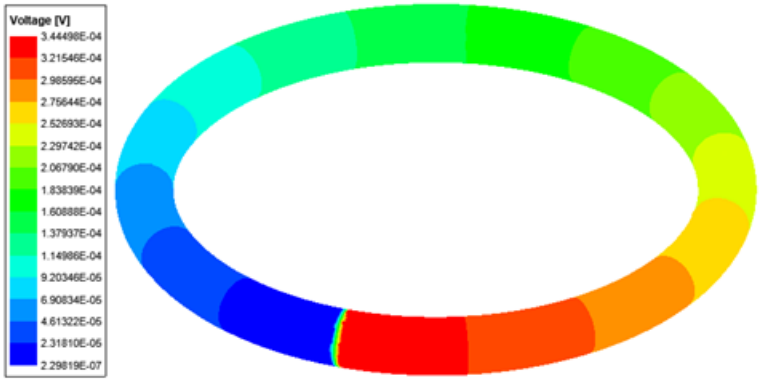
If you do not define a reference potential on the same conduction path, the potentials at the terminal will be floating, so they are not unique. However, B, J, H, and all other quantities will be unique because they depend on the derivative of the potential.



- **Double Potentials with Ground**: The solver treats the terminal as voltage (potential difference) terminal. At the terminal face, two scalar electric potential DOFs are defined and both DOFs are known. The difference between those potentials equals to the voltage value that you specified. If the voltage value is a positive quantity, the DOF in the specified direction is set to the specified voltage value, and the DOF in the opposite direction is set to 0. *This model must be used on faces inside the problem domain.*

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



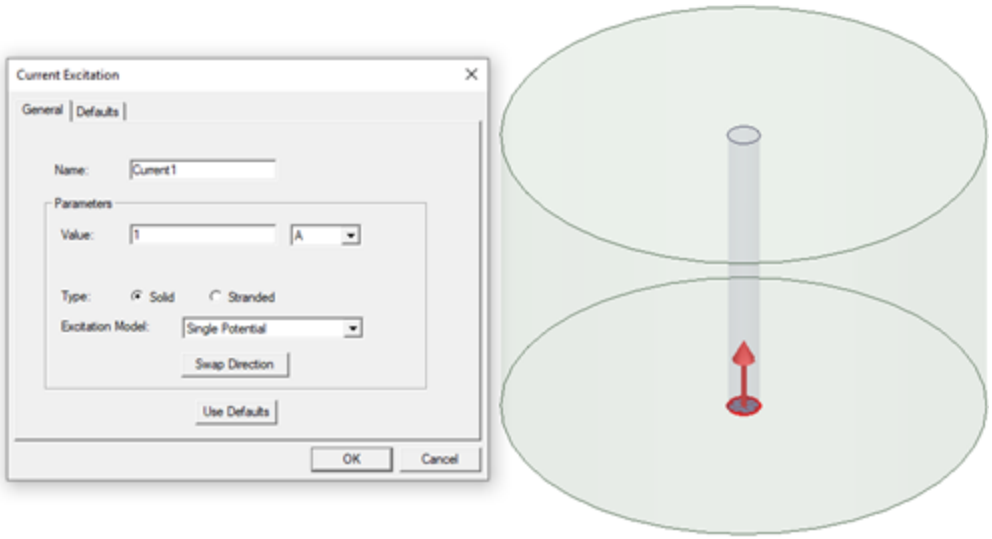


Current Excitations for A-Phi Formulation

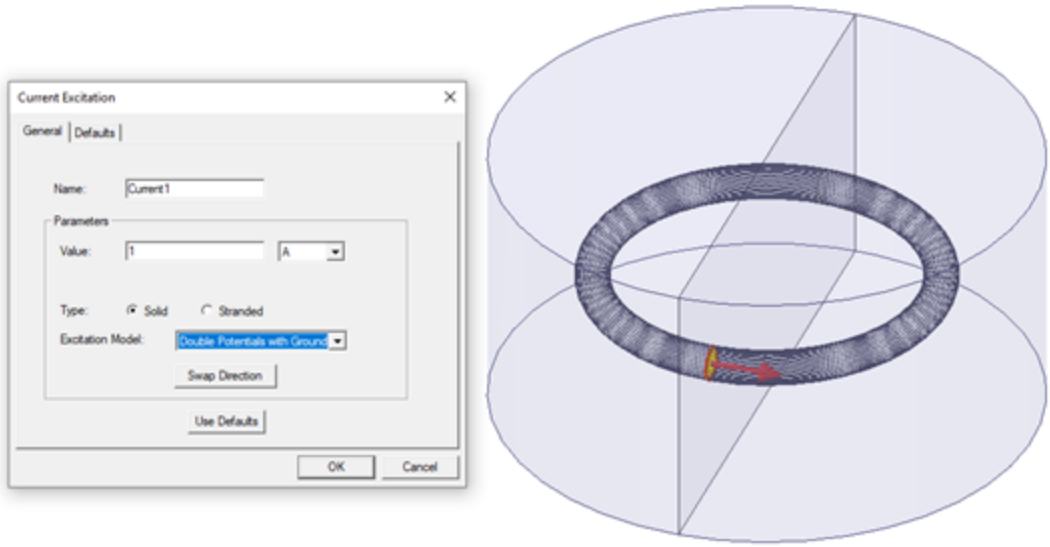
For designs with known current values, the user selects and assigns a current excitation on each face and also selects the type of conductor: solid or stranded. Current excitation supports three different excitation models for a **Solid** conductor type based on how the potential DOF is assigned on the face:

- **Single Potential**: Only one scalar electric potential DOF is assigned to the surface and it is unknown. *This model must be used only at the external boundaries of the problem domain.*

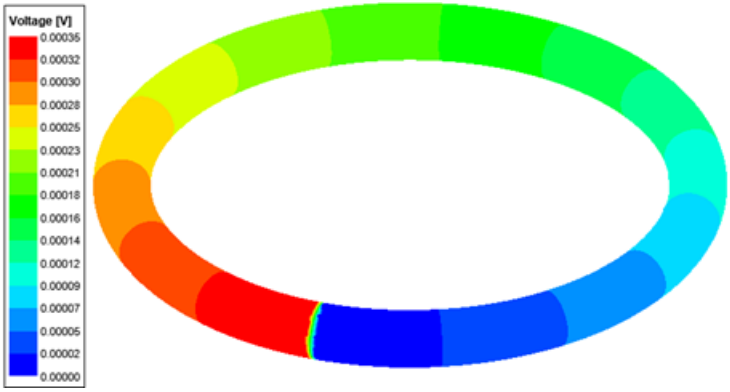
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Double Potentials**: At the terminal face, two electric scalar potential DOFs are defined and the DOFs are unknown. There is a potential difference at the terminal. The current will be unidirectional, and it will flow in the specified direction if the current value entered is a positive quantity. *This model must be used on faces inside the problem domain.*
- **Double Potentials with Ground**: At the terminal face, two scalar electric potential DOFs are defined. The potential on the opposite side of the specified direction is set to 0. The current will be unidirectional, and it will flow in the specified direction if the current value entered is a positive quantity. *This model must be used on faces inside the problem domain.*



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Rules/Limitations of Excitations on Conduction Paths

Winding Excitations

For the eddy current simulations with displacement effect **off** for all objects:

- One or two terminals can be defined on a conduction path: if the path is a loop, then only one coil terminal per coil must be defined, else only two coil terminals per coil must be defined (for both solid and stranded cases).
- More than two coil terminals are not allowed on a conduction path.
- Coil terminals must be ordered according to the physical potential distribution. One potential degree of freedom (DOF) is assigned to each external coil terminal. Two DOFs are assigned to each internal coil terminal. For windings with external coil terminals, there are two coil terminals defined for each conduction path. The potential DOF of the second terminal on a conduction path is linked to the potential DOF of the first terminal on the next conduction path, which means the coils are serially connected. That is why it is essential to create the coil terminals in the order of the actual potential distribution on serially connected coils (conduction paths). The potential on the last Coil Terminal is set to zero. The same rule for linking the DOFs applies to the windings with internal coil terminals.

When the displacement effect is **on**, the restrictions above **will not** apply. You can define one external coil terminal per conduction path and needs to ensure that there is a reference potential defined in the design.

Current Excitations

- For stranded current excitation: More than two terminals are not allowed on a conduction path. For a loop conduction path, one terminal is defined. For other conduction paths, you must define two terminals (in and out). A voltage terminal and a stranded current terminal cannot be defined on the same conduction path.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For solid current excitations: At least one voltage excitation is to be defined as a reference potential on the same conduction path of the current excitation. If a current excitation with **Double Potentials with Ground** exists in the path, or displacement current is enabled, this requirement does not apply. Multiple current excitations on a conduction path are allowed.

Voltage Excitations

- There must be at least one voltage excitation on a conduction path with current/voltage excitations if displacement effect is not enabled (if the path is not touching the odd symmetry boundary, or there is no current/voltage excitation with Double Potentials with Ground excitation, where the potential is set to 0 V).
- Winding excitation cannot be defined together with voltage or current excitations on the same path.
- On a conduction path there can be multiple solid voltage and current terminals defined.

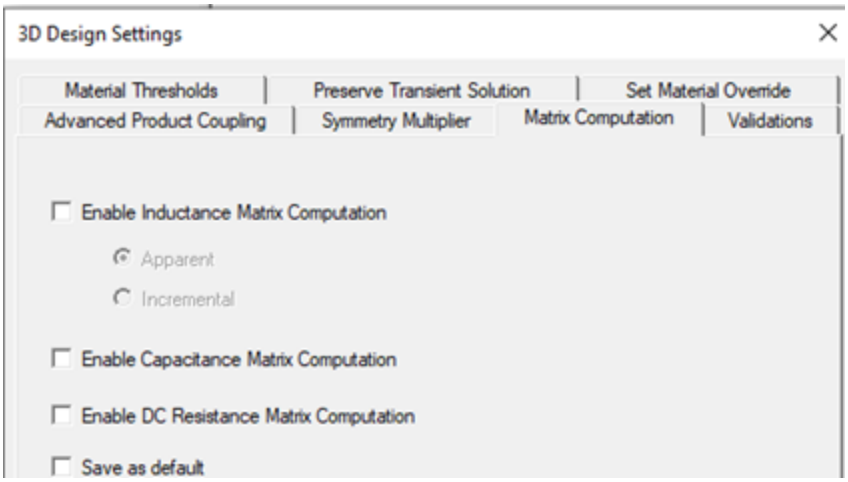
Independent/Dependent and Odd Symmetry Boundaries

- Excitations cannot be defined on **Independent**/**Dependent** boundaries.
- If a conduction path crosses an **Odd Symmetry** boundary, excitations must be defined at **External** boundaries because the default electric potential value is zero on the Odd Symmetry boundary. Voltage excitation can be used to set a different potential value on the face intersecting with an Odd Symmetry boundary. Also, any excitation (Coil Terminal, Voltage, or Current Excitation) assignment is allowed on the face intersecting with an Odd Symmetry boundary.
- If a conduction path crosses both Odd Symmetry and Independent/Dependent boundaries, the software's validation process does not allow one external terminal definition on the path. You can select **Perform minimal validations** under Design Settings to simulate such designs at the your own risk. Full model simulation is highly recommended for such designs.

Matrix Setup

Parametric matrix setup is supported for the A-Phi transient solver. Only one matrix setup can be defined for inductance, capacitance, or DC resistance calculations. For computation of the defined matrix, select **Enable Inductance Matrix Computation**, **Enable Capacitance Matrix Computation,** or **Enable DC Resistance Matrix Computation** in the 3D Design Settings window:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

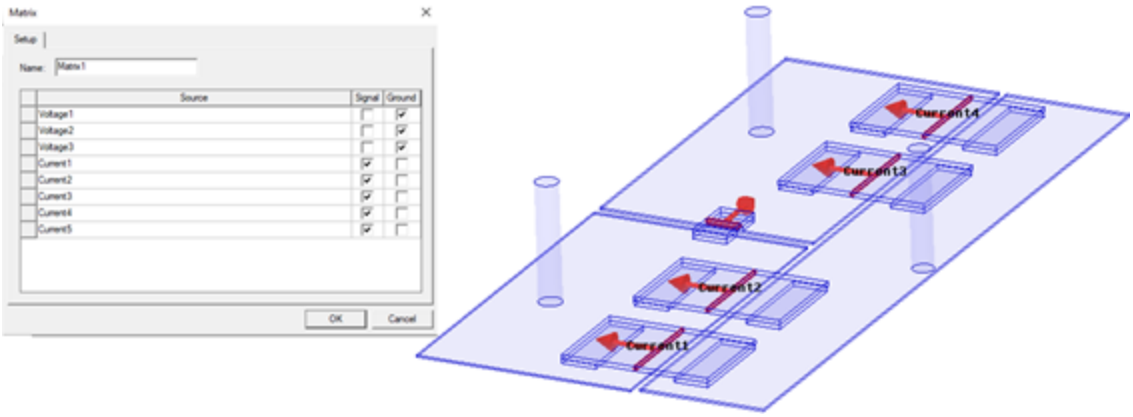


Next, select the signals and grounds for the matrix of interest. For the inductance matrix, the following are restrictions on ground selections:

- Ground selection is not allowed for windings because they are internally grounded.
- Ground selection is not allowed for stranded current excitations because no potential is defined on the conduction path.
- Ground selection is not allowed for current excitations with double potentials and double potentials with ground.
- Ground selection is only available for Voltage excitation with Single Potential model. Ground selection is not allowed for voltage excitations with double potentials and double potentials with ground.

It is highly recommended that ground selection be as in the nominal problem setup, especially for designs with multi-terminal excitations on a conduction path. Other solver types do the matrix computation after the solution of nominal problem so it is fully postprocessed, and you can select different signal/ground/return path configurations. However, the transient solver is computing the matrix at each timestep and the computation is highly dependent on how the current flows on a path, especially for designs with multiple excitations on the path.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The capacitance matrix is computed using the electrostatic energy stored in a system under 1 V excitation for one selected signal and 0 V for all others. Based on the selection of signals and grounds, the solver computes the capacitance matrix of the system. Stranded conductors and floating conductors (no signal or ground assigned) are treated as equipotential surfaces. For the capacitance matrix, the following rule applies for the ground selection:

- There must be at least one ground selection in the Matrix setup if there is no internal zero reference potential defined in the design. Odd Symmetry, winding excitation, and double potentials with ground excitation model are examples of internal grounds.

The resistance matrix is computed with the assumption of a DC conduction solution using the material values at time zero, so it is time independent. For stranded conductors, the solver internally calculates the stranded resistance values. The resistance value (the user typically inputs the stranded resistance value in addition to the source resistance) in the winding excitation setup is not included in the DC resistance matrix.

> **Note:** When there is a layout component with RLC components, the inductance, capacitance, and resistance matrices are calculated with the assumption that R is connected, L is shorted, and C is open. Thus, the L matrix values are the total inductances of the conduction paths connected with R and L circuit elements. Also, the resistance matrix values include the lumped R circuit component values. Therefore, you should be careful while defining signal(s) and ground(s) on a conduction path.

Postprocessing Support

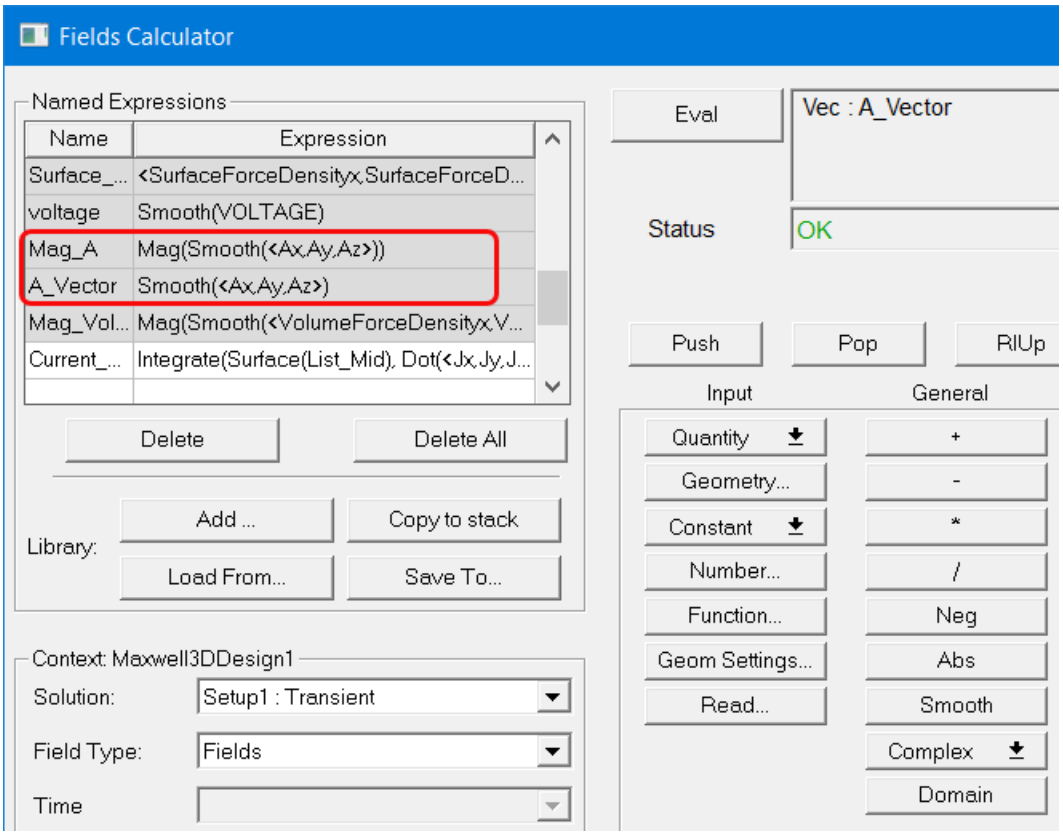
The A-Phi transient solver supports all the postprocessing features of the transient solver. Additionally, the following features are supported:

- Voltage in the Field Plot and in the Fields Calculator; there are cases where no voltage is defined in an object (in a stranded object, or in an object with no eddy or displacement

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

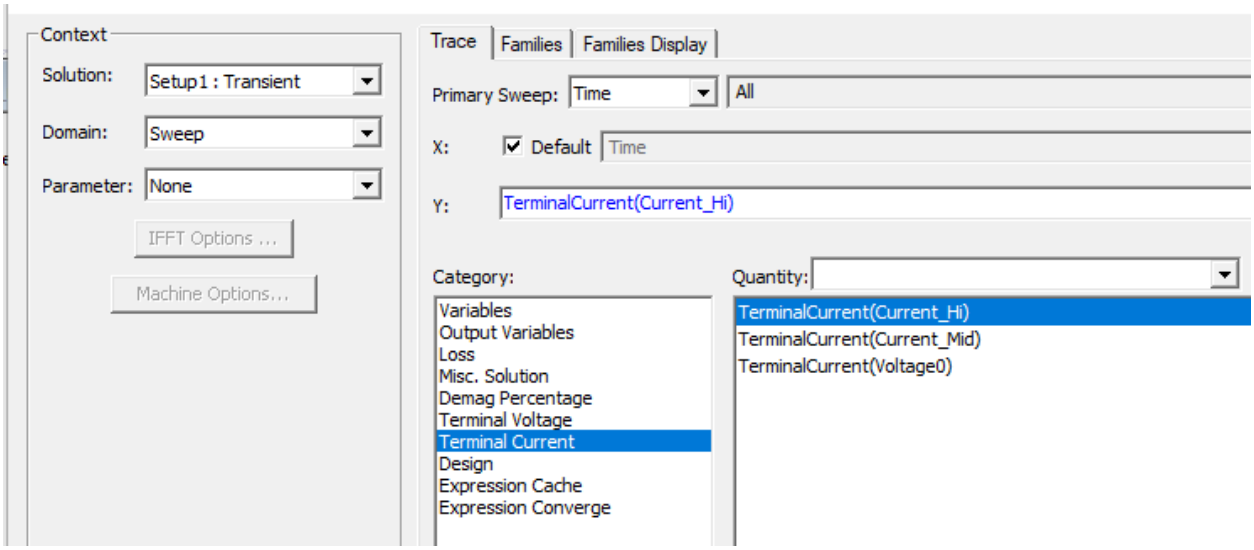
effect setting).

- Magnetic vector potential in the Fields Calculator:



- Voltage and current quantities at the Voltage and Current Terminals; the current computed at any terminal is assumed to be flowing into the problem domain.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Electrostatic Solver

The electrostatic field simulator computes static electric fields in 2D or 3D due to

- stationary charge distributions.
- applied potentials.

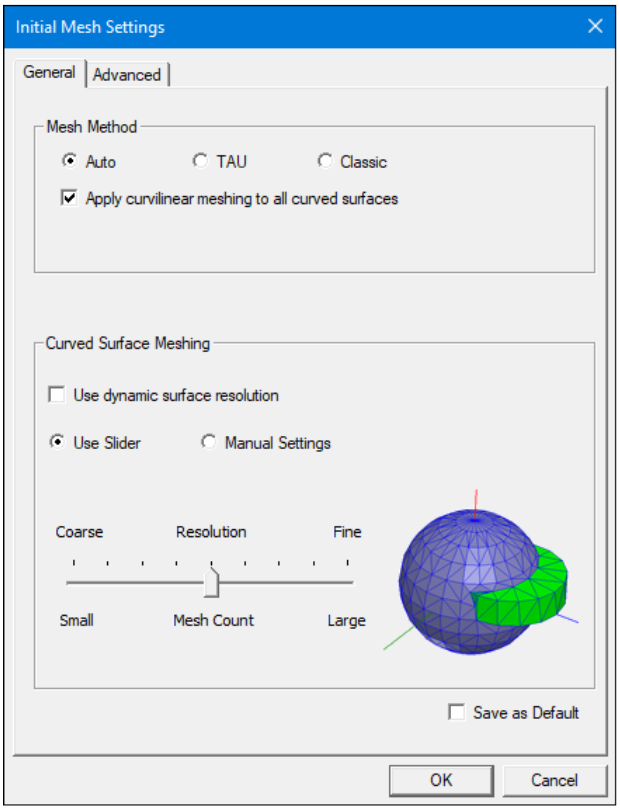
The quantity for which the electrostatic field simulator solves is the scalar electric potential, Φ; the electric field (**E**-field) and the electric flux density (**D**-field) are automatically calculated from the potential. Derived quantities such as forces, torques, energy, surface charge density, and capacitance matrix may be calculated from these basic field quantities.

Meshing Recommendations for the Electrostatic Solver

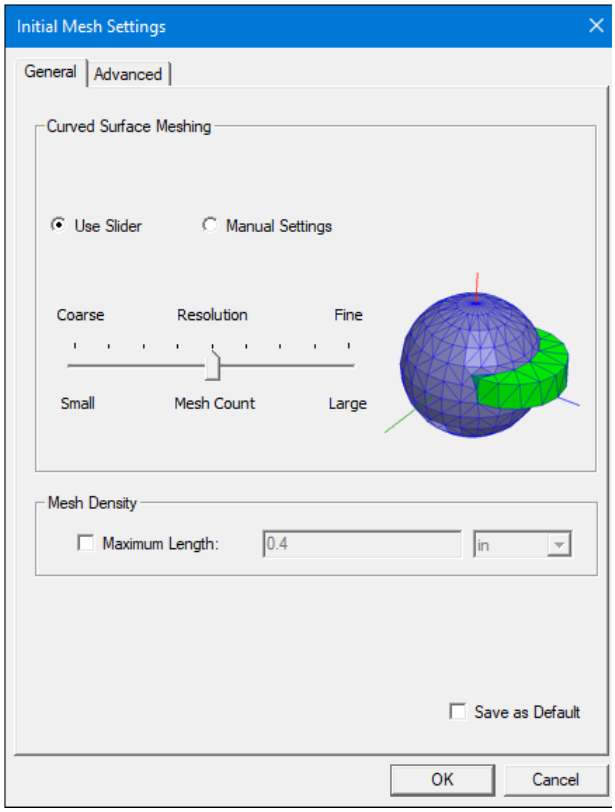
There are two mesh methods available for electrostatic simulations: **TAU** and **Classic**. For a 3D electrostatic solution, the user can let the mesher select the method (**Auto**), or select between **TAU** and **Classic**. For a 2D solution, the TAU mesher is the only mesher available, so there are no selection options.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3D Electrostatic Initial Mesh Settings



2D Electrostatic Initial Mesh Settings



Classic Mesh

The Classic mesh option offers the following advantages:

- It resolves the grid points on the 2D/3D boundary at the initial mesh creation.
- It also adds the grid point on the geometry boundary during adaptive mesh process.
- It provides accurate results on the object boundary, especially for the electrode surface.

TAU Mesh

The TAU mesh option offers the following advantages:

- The **Manual** Curved Surface Meshing approximation setting creates a good, smooth surface mesh on electrode surface.
- A small enough **Surface Deviation** tolerance is the key to get accurate surface electric field results.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2D AC Conduction

The 2D AC conduction field simulator computes steady-state 2D electric fields in conductors due to applied potentials.

The AC conduction field solver allows you to analyze conduction currents due to time-varying electric fields in conductors and lossy dielectrics. Use it to analyze current distributions, electric field distributions, potential differences, admittances, lossy materials, and stored energy. For instance, the admittance matrix associated with a structure can be computed using the AC conduction field solver. In addition, any quantity that can be derived from the basic electromagnetic quantities can then be analyzed.

The AC conduction field solver can compute conduction currents for Cartesian (XY) and axisymmetric (Cylindrical about Z) models. It assumes that all sources are sinusoids oscillating at the same frequency. Optionally, can specify different phase angles for different sources.

You are expected to specify material properties and the electric potential at one or more object interfaces or boundaries in the model. The AC conduction field solver then computes the electric potential, $\Phi(t)$, for the model. From the electric potential, it derives the electric field, **E**(t), the electric flux density, **D**(t), and the current density, **J**(t).

3D AC Conduction

AC conduction analysis is the study of

- electric field
- current
- losses
- lumped parameters (CG)

The AC conduction field solver allows you to analyze both conduction and displacement currents due to time-varying electric fields in conductors and perfect and lossy dielectrics.

Use it to analyze current distributions, electric field distributions and potential differences, and CG parameters. For instance, the admittance matrix associated with a structure can be computed using the AC conduction field solver. In addition, any quantity that can be derived from the basic electrical quantities (Electric Field, current densities, etc.) can then be analyzed.

The AC conduction field solver assumes that all sources are sinusoids oscillating at the same frequency. Optionally, you may specify different phase angles for different sources.

You need to specify these related material properties: permittivity, conductivity (for lossy materials), frequency-dependent dielectric loss tangent. You also must specify the current, electric potential at one or more object interfaces or boundaries in the model. The AC conduction

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

field solver then computes the electric potential for the model. From the electric potential, it derives the electric field, **E**, the electric flux density, **D**, and the current density, **J**.

The supported 3D material models can be found in the table in Maxwell Solvers Material Properties.

DC Conduction Solver

The electric conduction field simulator computes steady-state 2D or 3D electric fields in conductors due to

- current excitations.
- applied potentials.

The quantity for which the electric conduction field simulator solves is the electric potential, Φ; the electric field (**E**-field**)** and the current density (**J**-field) are automatically calculated from the potential. The resistance matrix, a derived quantity, may be calculated from these basic field quantities. As an additional option, perfect insulators, that is, non-conduction regions, surrounding the conductors can also be added to the simulation domain allowing to calculate the electric field everywhere including the insulators.

Electric Transient Solver

The 3D electric transient field simulator computes time-varying electric fields excited by

- time-varying applied potentials.
- total charge and volume charge density.
- time-varying current excitations.

The quantity for which the electric transient field simulator solves is the electric potential, Φ, by solving the following equation:

$$-\nabla \cdot \left(\varepsilon \nabla \frac{\partial \Phi}{\partial t}\right) - \nabla \cdot (\sigma \nabla \Phi) = 0$$

The electric field (**E**-field), the current density (**J**-field), and the electric flux density (**D**-field) are automatically calculated from the potential.

Derived solution parameters such as the electric energy, Ohmic loss, surface charge density, and maximum electric field can be obtained for each solid.

The temporal discretization is adaptive and, besides the initial and maximum timestep, it also depends on temporal tolerance. The default temporal tolerance is 0.005, but it depends on the problem to be solved.

Normally, problems with high time derivative field solutions would require smaller temporal tolerances.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the case of nonlinear problems, temporal tolerance also depends on the nonlinearity of materials, and its value needs to be set around 1e-12 or even smaller.

Maxwell Solvers Material Properties

The tables below list material properties supported by the Maxwell 2D and 3D solvers. Additional information may be found in the Adding Materials, Assigning Material Property Types, and Materials topics.

Material Properties for Maxwell 3D Solvers

Material property	Electrostatic	DC Conduction	AC Conduction	Electric Transient	Magnetostatic	AC Magnetic (T-Omega)	AC Magnetic (A-Phi)	Magnetic Transient (T-Omega)	Magnetic Transient (A-Phi)
Permittivity									
Linear (Simple)	X	X	X	X		X	X		X
Nonlinear	X	X	X						
Isotropic	X	X	X	X		X	X		X
Anisotropic	X	X	X	X		X	X		X
Nonlinear anisotropy									
Frequency dependent			X			X	X		
Conductivity									
Linear (Simple)		X	X	X	X	X	X	X	X
Nonlinear		X	X	X					
Isotropic		X	X	X	X	X	X	X	X
Anisotropic		X	X	X	X	X	X	X	X
Nonlinear anisotropy									
Frequency dependent			X			X	X		
Coreloss						X	X	X	X
Dielectric loss tangent							X		

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Material property	Electrostatic	DC Conduction	AC Conduction	Electric Transient	Magnetostatic	AC Magnetic (T-Omega)	AC Magnetic (A-Phi)	Magnetic Transient (T-Omega)	Magnetic Transient (A-Phi)
Linear (Simple)			X			X	X		
Nonlinear									
Isotropic			X			X	X		
Anisotropic			X				X		
Nonlinear anisotropy									
Frequency dependent			X			X	X		
Permeability									
Linear (Simple)					X	X	X	X	X
Nonlinear					X	X (1)	X	X	X
Isotropic					X	X	X	X	X
Anisotropic					X	X	X	X	X
Nonlinear anisotropy					X	X		X	X
Vectorial hysteresis								X	
Frequency dependent						X	X		
Magnetic loss tangent									
Linear (Simple)						X	X		
Nonlinear									
Isotropic						X	X		
Anisotropic						X	X		
Nonlinear anisotropy									
Frequency dependent						X	X		

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Material property	Electrostatic	DC Conduction	AC Conduction	Electric Transient	Magnetostatic	AC Magnetic (T-Omega)	AC Magnetic (A-Phi)	Magnetic Transient (T-Omega)	Magnetic Transient (A-Phi)
Composition									
Solid					X	X		X	X
Lamination					X (2)	X (2)		X (2)	X
Litz Wire					X (3)	X (3)		X (3)	X
Magnetostriction					X(4)			X(4)	
Notes:									
(1) This specific nonlinear model is different from other solvers									
(2-3) Lamination and Litz wire have different models in the magnetostatic, AC magnetic and magnetic transient solvers									
(4) Magnetostriction has different models in the magnetostatic and magnetic transient solvers									

Material Properties for Maxwell 2D Solvers

Material property	Electrostatic	DC Conduction	AC Conduction	Magnetostatic	AC Magnetic	Magnetic Transient
Permittivity						
Linear (Simple)	X		X		X	
Nonlinear						
Isotropic	X		X		X	
Anisotropic	X		X		X	
Nonlinear anisotropy						
Frequency dependent			X		X	
Conductivity						
Linear (Simple)	X	X	X	X	X	X
Nonlinear						

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Material property	**Electrostatic**	**DC Conduction**	**AC Conduction**	**Magnetostatic**	**AC Magnetic**	**Magnetic Transient**
Isotropic	X	X	X		X	X
Anisotropic		X	X		X	X
Nonlinear anisotropy						
Frequency dependent			X		X	
Coreless						
					X	X
Dieletric loss tangent						
Linear (Simple)			X		X	
Nonlinear						
Isotropic			X		X	
Anisotropic					X	
Nonlinear anisotropy						
Frequency dependent			X		X	
Permeability						
Linear (Simple)				X	X	X
Nonlinear				X	X (1)	X
Isotropic				X	X	X
Anisotropic				X	X	X (5)
Nonlinear anisotropy				X		X(5)
Vectorial hysteresis						X
Frequency dependent					X	
Magnetic loss tangent						
Linear (Simple)					X	
Nonlinear						

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Material property	Electrostatic	DC Conduction	AC Conduction	Magnetostatic	AC Magnetic	Magnetic Transient
Isotropic					X	
Anisotropic					X	
Nonlinear anisotropy						
Frequency dependent					X	
Composition						
Solid				X	X	X
Lamination				X	X (2)	X (2)
Litz Wire					X (3)	X (3)
Magnetostriction				X(4)		X(4)
Notes:						
(1) This specific nonlinear model is different from other solvers						
(2-3)Lamination and Litz wire have different models in the magnetostatic, AC magnetic and magnetic transient solvers						
(4) Magnetostriction has different models in the magnetostatic and magnetic transient solvers						
(5) Supported only in 2D transient designs with Cartesian XY symmetry						

Disconnected Solution Domains

There are two ways to create a problem that contains multiple disconnected solution domains:

- By not creating or by excluding (not solving in) the background object, when the included objects are not all touching.
- By using excluded objects or perfect conductors that cut the included objects into different regions.

The Maxwell solver identifies multiple disconnected solution domains.

For the electric solvers (electrostatic, conduction, and conduction+electrostatic), the solution is allowed even if multiple disconnected solution domains are detected. In this case, an error

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

message appears only if a solution domain with inconsistent or unsolvable solution setup is detected.

For the magnetic solvers (Magnetostatic, AC Magnetic, and Transient EM), an error message appears when multiple disconnected solution domains are detected.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6 - Assigning Materials

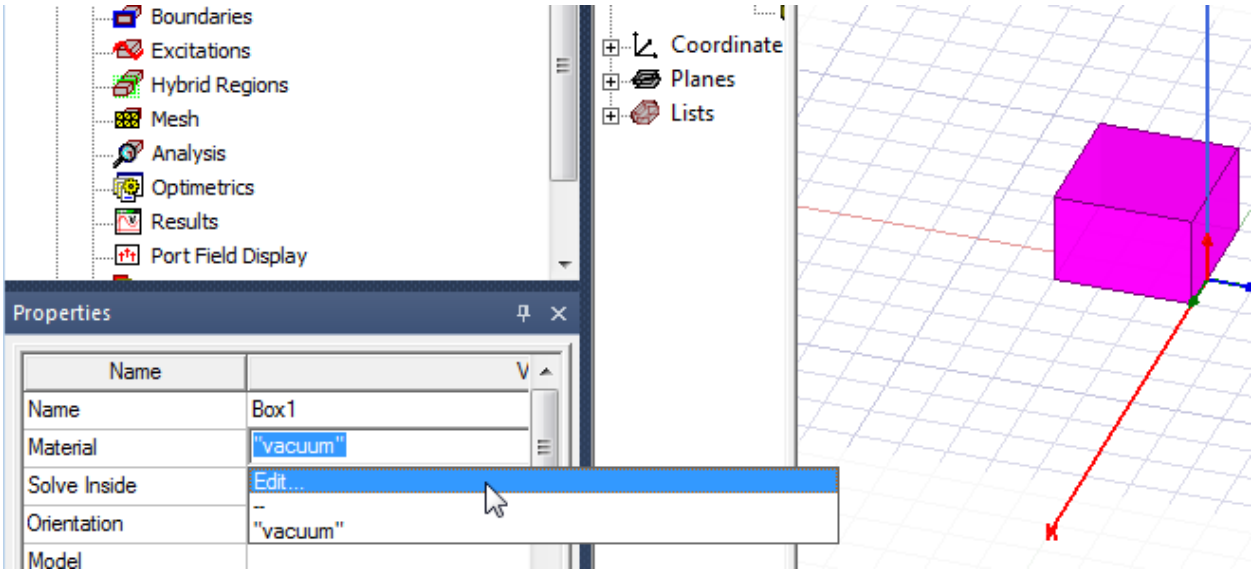
You can add, remove, and edit materials in two main ways:

- Using the **Tools** > **Edit Libraries** > **Materials** menu command.
- Under **Definitions** in the Project Manager, right-click **Materials** and select **Edit Library**.

Regardless of which of the preceding two methods you use to edit a library, any new material you create exists only in the current project. Similarly, if you edit an existing material, the edited version becomes a *Project* material and exists only within the current project. To make a new or modified material available for other projects, you must export it to a user library and choose that library (or select **Show all libraries**) within the **Select Definition** dialog box.

To assign a material to an object, follow this general procedure:

1. Select the object to which you want to assign a material.
2. Click **Modeler** > **Assign Material** or select the Material field in the **Properties** window for the selected object, and select **Edit...** from the drop-down menu:

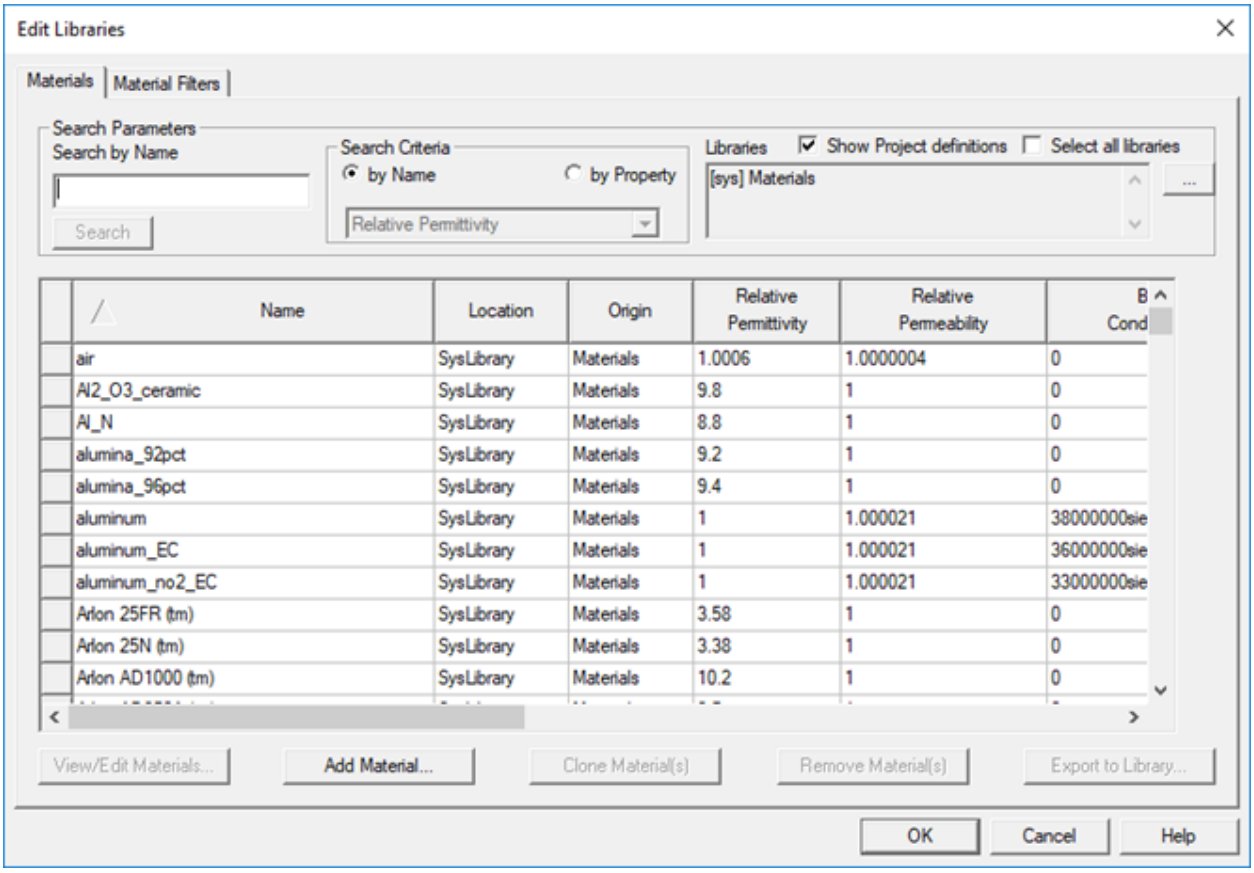


The **Select Definition** window appears. The current material is highlighted, with the Name, Location, Origin library, and parameter values shown.

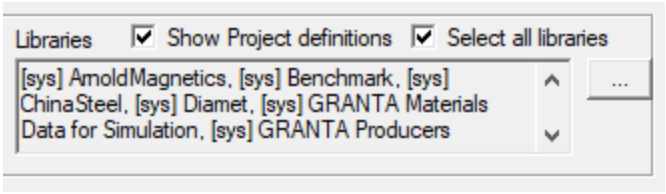
> **Note:** You can resize the dialog box to show more of the materials and/or material properties without having to use the scroll bars.

Select Definition Dialog Box - Materials

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

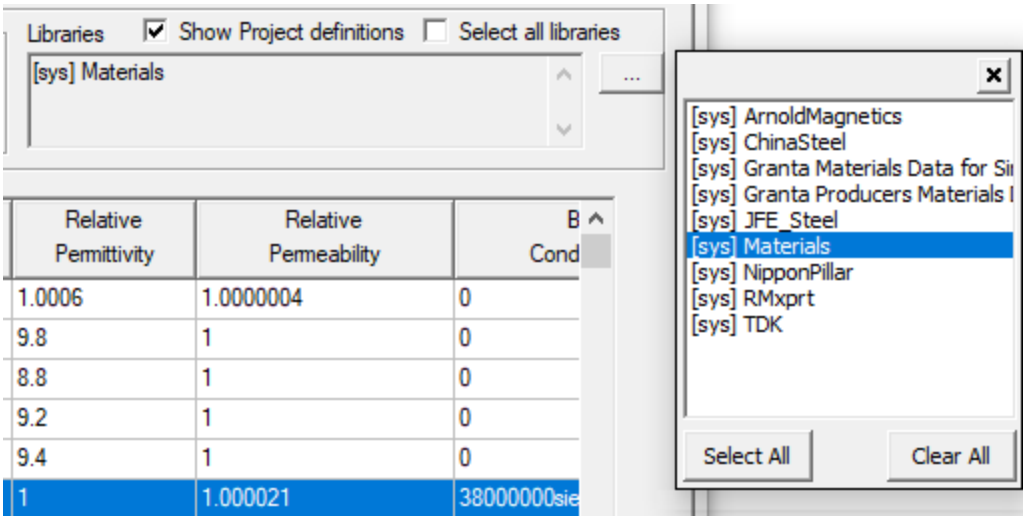


When the **Select all libraries** check box is selected, the window lists all of the materials in Ansys Electronics Desktop's global material library as well as the project's local material library

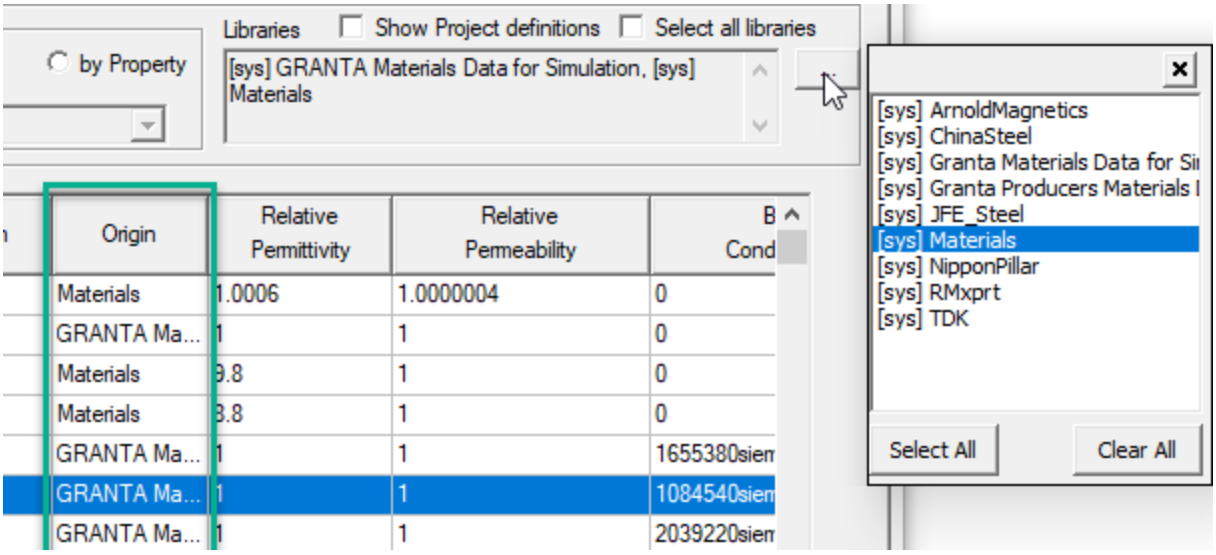


If you click the ellipsis button **[...]**, you see a list of all available libraries. Any selected libraries are highlighted. Shift-Click allows you select a range. Ctrl-Click allows you to select any libraries.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Origin** column shows the originating library for each material, whether the **sys** library, or one of the additional libraries listed in the Libraries pane.



For further information on the materials and their intended uses, you can refer to the published information on materials from those libraries. For example, the Schott materials are described in detail in the Wily Series in Materials for Electronic & Optoelectronic Applications, *Microwave Dielectric Materials and Applications*, edited by M. T. Sebastian, Rick Ubic, and Heli Jantunen, volumes 1 and 2.

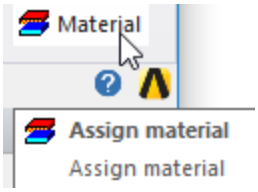
GRANTA Materials Data for Simulation (MDS) is an optional, licensed feature, including more than 700 generic materials and over 2100 producer-specific magnetic materials and PCB materials. When you use or view the GRANTA licensing libraries, the GRANTA

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

license is checked out and held for 30 minutes. Viewing the contents of a GRANTA library in the Materials dialog box or modifying an object to use a GRANTA material causes a license check out and hold. If the license is already checked out, the hold time is reset to 30 minutes.

You can also open the **Select Definition** window in one of the following ways:

- In the **Properties** dialog box for the object, click the material name under the **Attributes** tab. A drop-down menu shows an **Edit...** button that opens the **Select Definition** window.
- With an object selected in the **Modeler** window, on the **Draw** ribbon, select the **Material** icon.

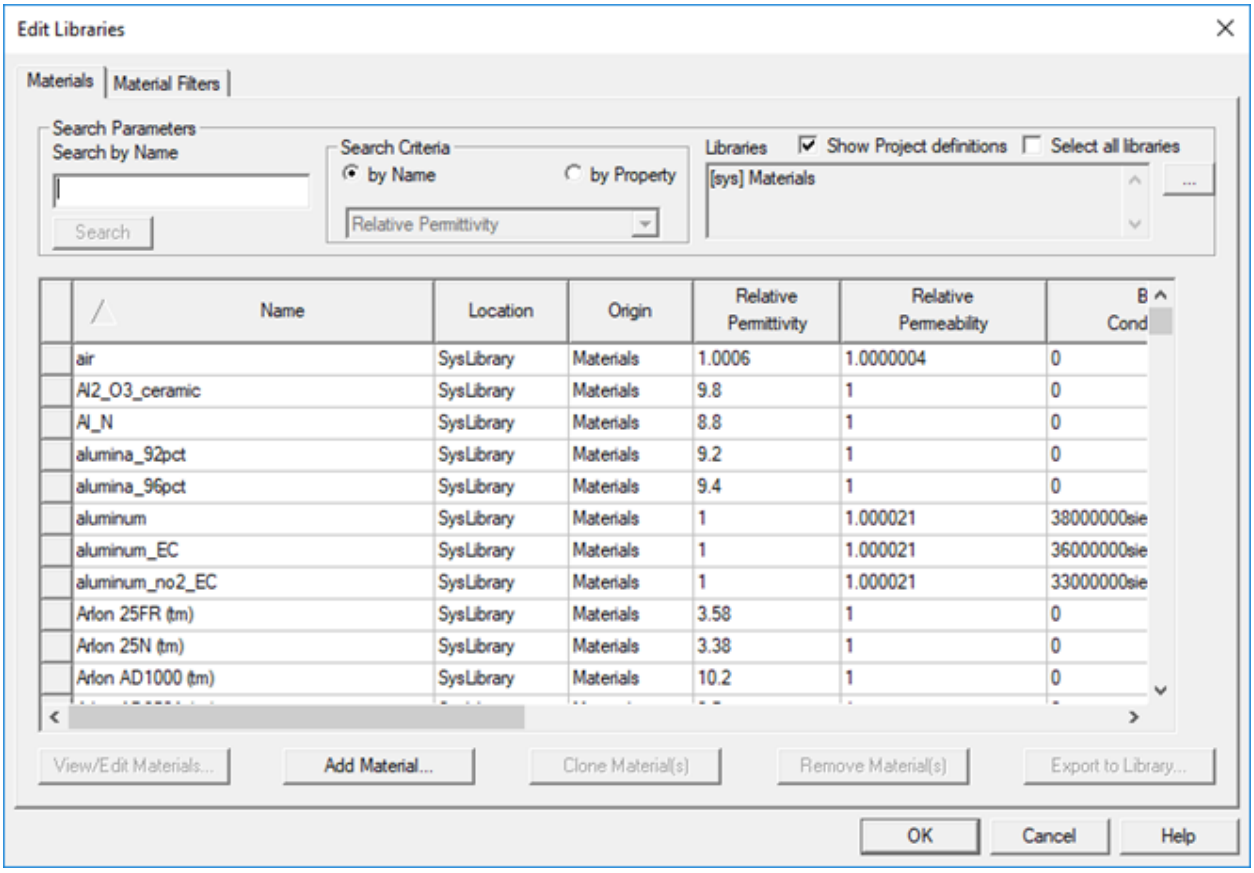


The menu also lists materials included in the current project. Selecting one of these materials provides another way to assign materials to an object.

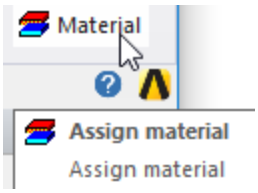
- Right-click **Model** in the project tree, and then click **Assign Material** on the shortcut menu.
- Right-click the object in the history tree, and then click **Assign Material** on the shortcut menu.

Select Definition Dialog Box - Materials

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. With an object selected in the **Modeler** window, on the **Draw** ribbon, select the **Material** icon.



4. Select a material from the list.

> **Note:**
>
> You can search the listed materials by name or property value.

If the material you want to assign is not listed, add a new material to the global or local material library, and then select it.

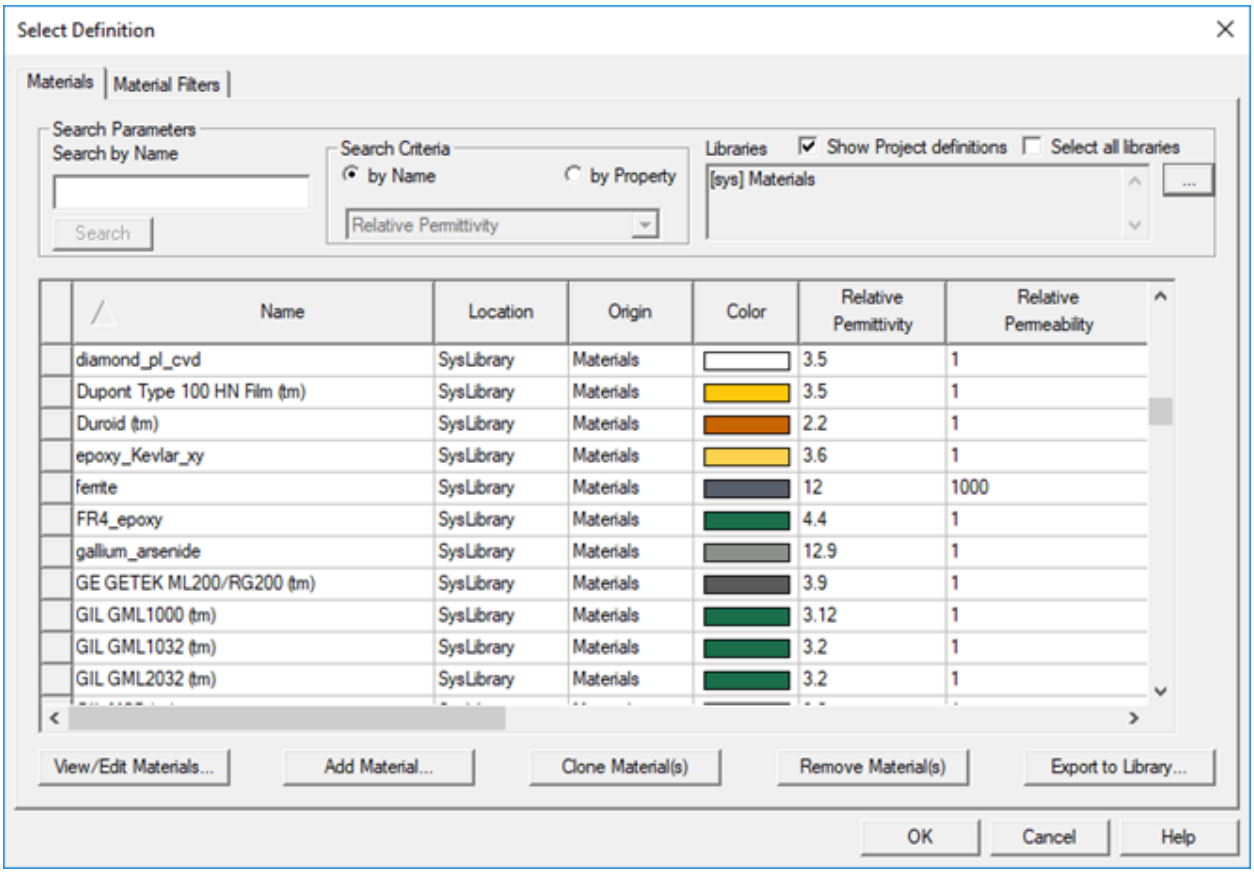
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click **OK**.

   The material you chose is assigned to the object.

> **Note:**
>
> In the History Tree, by default, objects are grouped by material. You can easily change the default grouping. Using the menu bar, click **Modeler** > **History Tree Layout** and clear **Group Objects by Material** along with any other listed option you may wish to disable. You can also perform this action from the **Tree View** drop-down menu on the **View** ribbon tab or from the shortcut menu that appears when you right-click most of the items in the History Tree.

If you check **Show Material Colors** on the Material Filters tab of the **Select Definition** or **Edit Library** dialog box, the **Materials** tab will include a **Color** column showing a color swatch for each listed material.



You can edit the color and transparency values for materials in the View/Edit Material dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solving Inside or on the Surface

When you assign a material to an object, you can specify whether to generate a field solution inside the object or on the surface of the object. If you elect to generate a solution inside the object, a mesh is created inside the object and generate a solution from the mesh. If you elect to generate a solution on the surface of the object, only a surface mesh is created for the object.

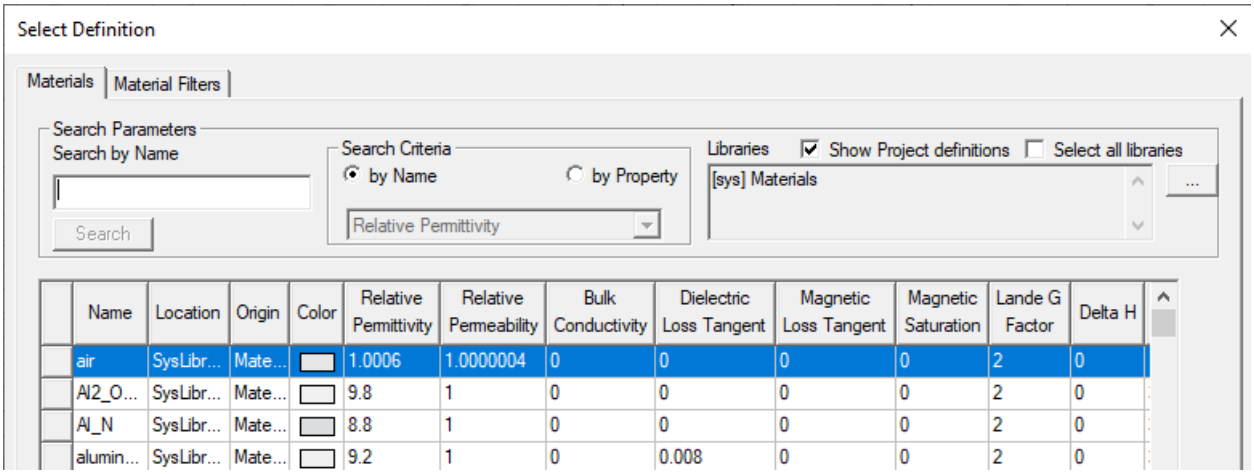
If you want a solution to be generated inside an object, select **Solve Inside** in the **Properties** window. Conversely, if you want a solution to only be generated on the surface of an object, clear the **Solve Inside** option in the **Properties** window.

By default, **Solve Inside** is selected for all objects with a bulk conductivity less than 10^5 siemens/meter and for perfect insulators. By default, the **Solve Inside** option in the **Properties** window is clear for perfect conductors.

You can set the default values for perfect conductors and insulators in the Maxwell 2D or Maxwell 3D Material Threshold options panel (refer to *Setting Maxwell 2D Options*, or *Setting Maxwell 3D Options* for details.

Searching for Materials

You can search for materials in the **Select Definition** dialog box. The default Search Criteria is **by Name**, which is shown in the following example. Alternatively, you can choose to search by Property.



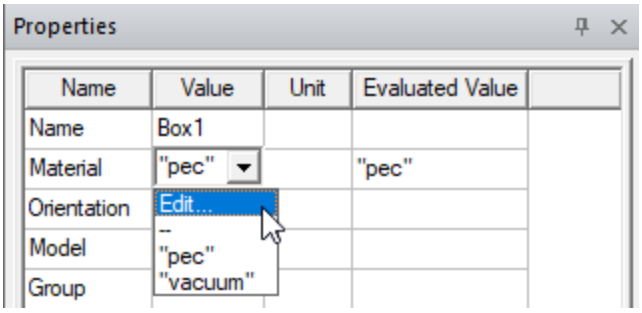
To search for a material **by Name**:

1. Access the **Select Definition** dialog box using one of the following methods:

   With one or more objects selected (that is, to assign a material to selected objects):

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

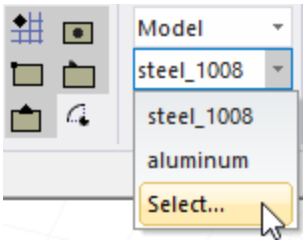
- Click **Modeler** > **Assign Material**
- Click the **Material** *value* in the docked **Properties** window, and select **Edit...** from the drop-down menu.



- On the **Draw** ribbon tab, click **Assign material**.

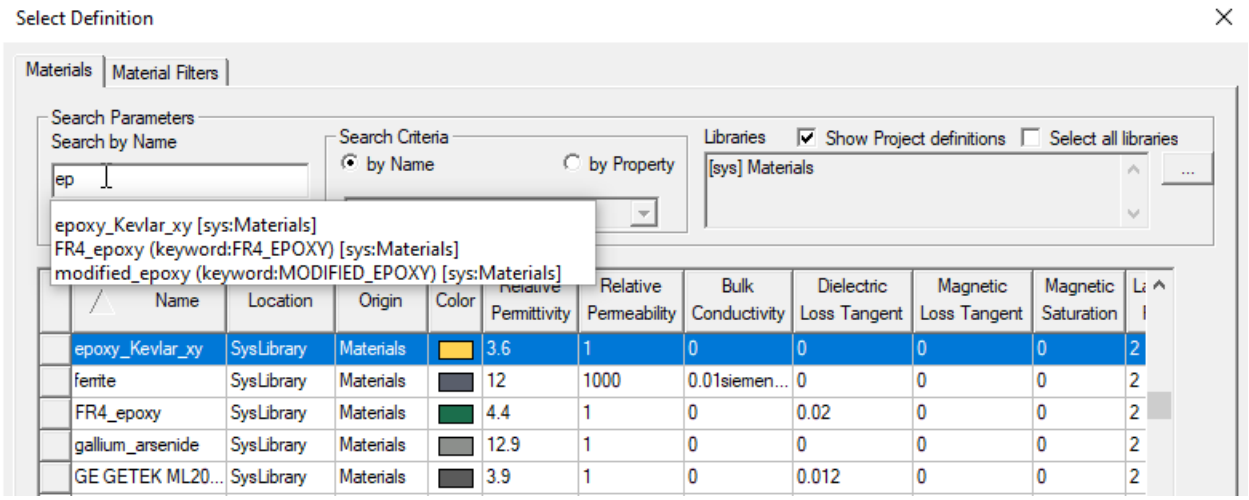
With nothing selected (that is, to set the default material):

- On the **Draw** ribbon tab, choose **Select** from the **Default material** drop-down menu:



2. In the **Search Criteria** section, ensure that **by Name** is selected, and in Libraries, specify the Libraries that you want to search. Only loaded libraries participate in text and keyword matching.
3. In the **Search by Name** text box, type a portion of the desired material name. The search text is case-insensitive.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



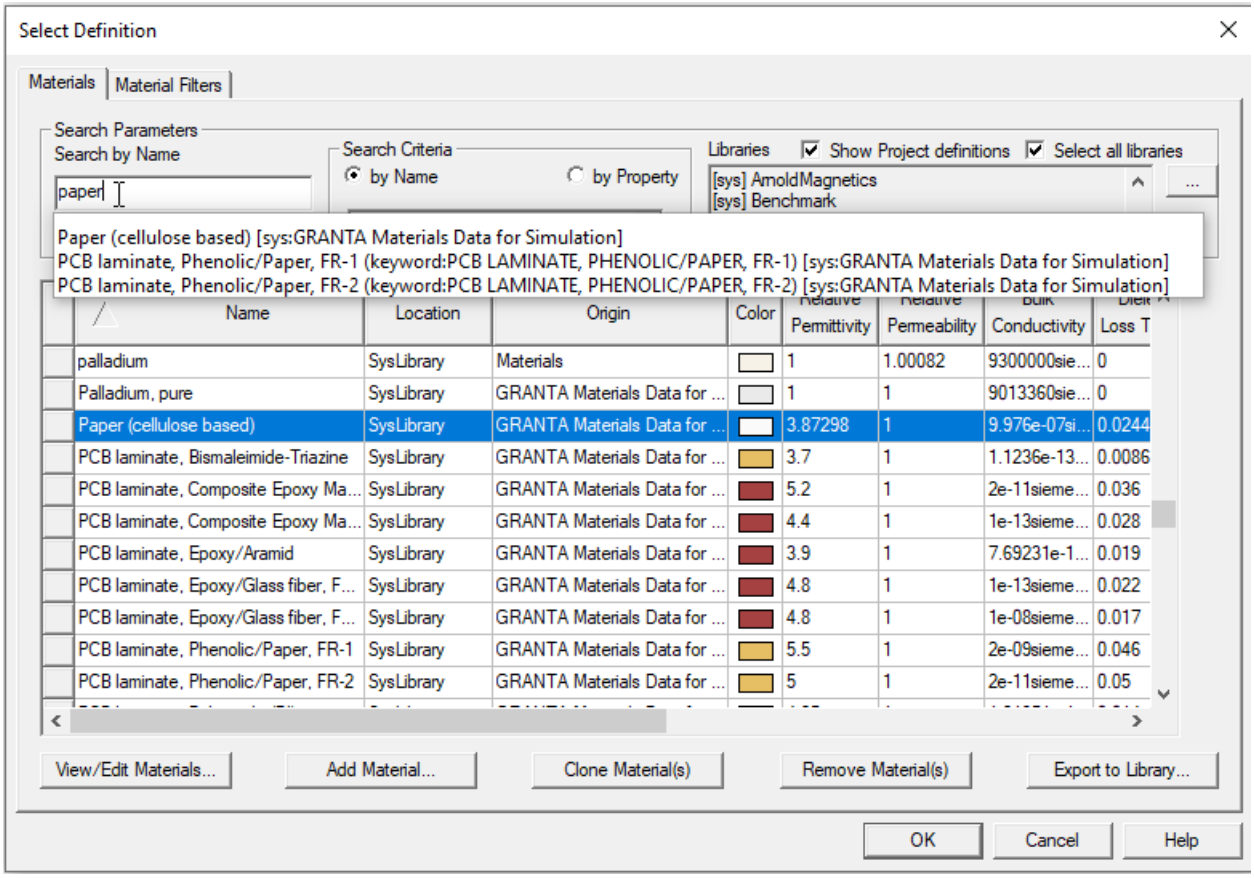
A drop-down menu under the Search Parameters field will show the a list of the materials from selected libraries matching any part of the current text, and, if the Granta Materials library is selected, matching keywords in the material definitions. The first row containing the material name most similar to the characters you typed will be selected.

If the selected material is not the one you are searching for, do one of the following:

- Use the keyboard's arrow keys to select the material above or below the currently selected row.
- Use the scroll bar to scroll the listed materials upward or downward and click the desired material when it is visible.
- Type different characters in the **Search by Name** text box.

When the **Select all libraries** option is selected, the window lists all of the materials in Ansys Electronics Desktop's global material libraries that are applicable to the current design type as well as those in the project's local materials library. The Granta Materials Data for Simulation library now includes keywords that are used in the auto-complete matching. For example, entering "glass" will generate an auto-complete list of all materials with a name containing "glass" or with a keyword that contains "glass", all case-insensitive, for materials showing in the grid. The drop down of potential matches under the search field applies to all libraries. What shows in the grid is defined by which libraries are selected, what material filters are selected, project/design-specific validation, and whether project materials are selected for display.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Searching by Material Property

1. Access the **Select Definition** dialog box using one of the following methods:

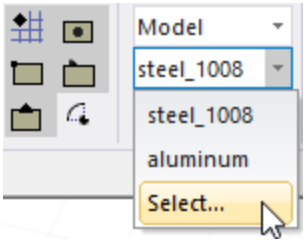
   With one or more objects selected (that is, to assign a material to selected objects):

   - Click **Modeler** > **Assign Material**.
   - Click the **Material** *Value* in the docked *Properties* window, and select **Edit...** from the drop-down menu.
   - On the **Draw** ribbon tab, click **Assign material**.

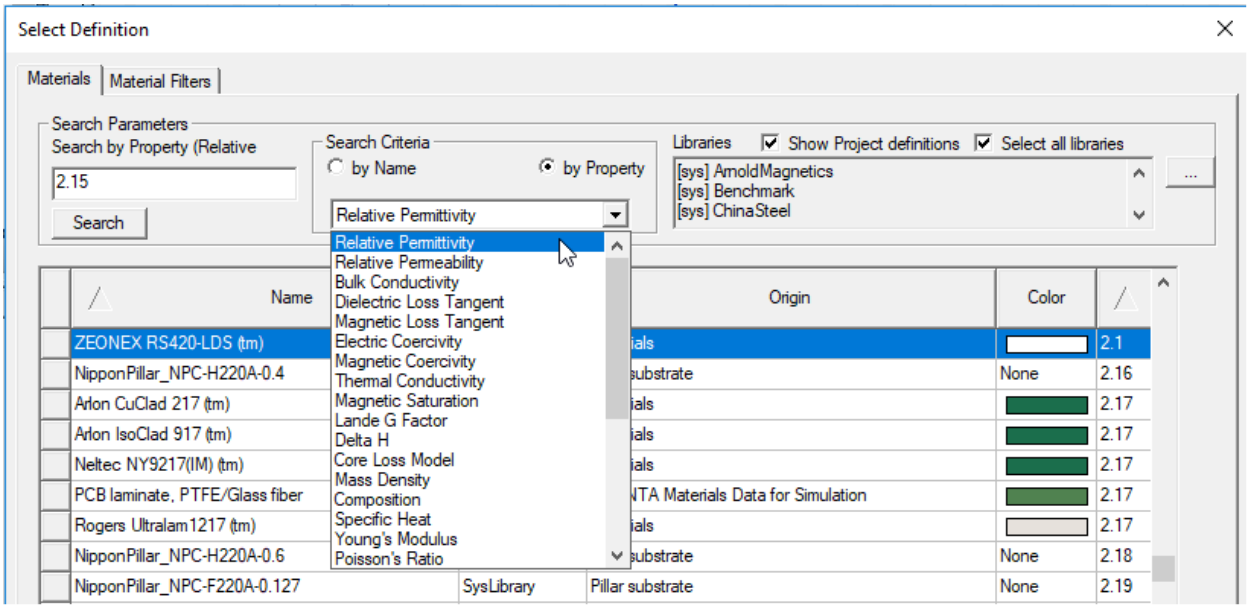
   With nothing selected (that is, to set the default material):

   - On the **Draw** ribbon tab, choose **Select** from the **Default material** drop-down menu:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. In the **Search Criteria** section, select **by Property**.
3. Select a material property from the drop-down menu:



> **Note:**
>
> By default, not all of the available properties are displayed in the materials table. Only the properties commonly used by the product are displayed in the table, though all properties are available in this drop-down menu. To view the complete table of properties, see Filtering Materials.

4. In the **Search Parameters** area, type a numerical value in the **Search by Property** text box and then click **Search**.

   The materials are sorted according to the property you selected. Additionally, the material with the property value closest to the one you typed, but without exceeding it, is selected.

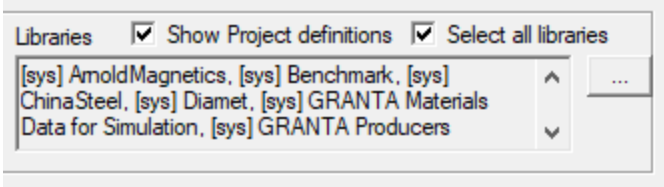
   If the selected material is not the one you are searching for, do one of the following:

   - Use the keyboard's arrow keys to select the material above or below the currently selected row.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Use the scroll bar to scroll the listed materials upward or downward and click the desired material when it is visible.
- Type a different numerical in the **Search by Property** text box and click **Search** again.

When the **Select all libraries** option is selected, the window lists all of the materials in Ansys Electronics Desktop's global material libraries that are applicable to the current design type as well as those in the project's local materials library.



Adding New Materials

You can add a new material to a project or to the global user-defined material library. To make the new project material available to all projects, you must export the material to a global user-defined material library.

Materials are added using the **View/Edit Material** window, which can be opened from either the **Select Definition** window or the **Edit Libraries** window.

To open the **Select Definition** window:

- Click **Modeler** > **Assign Material**.

The **Select Definition** window appears.

To open the **Edit Libraries** window:

- Click **Tools** > **Edit Libraries** > **Materials**. (In the project tree, you can also right-click **Materials**, and select **Edit Library**.)

The **Edit Libraries** window appears.

To add a new material:

1. From either the **Select Definition** dialog box or the **Edit Libraries** window, click **Add Material**.

   The **View/Edit Material** window appears.

2. Type a name for the material in the **Material Name** text box, or accept the default.
3. Select one of the following from the **Material Coordinate System Type** drop-down menu:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Cartesian** (default)
- **Cylindrical**
- **Spherical**

4. Under **View/Edit Material for** select:
   - **Active Design** to display properties used in the active design.
   - **Active Project** to display properties used in the active project.
   - **All Properties** to display all properties available. This enlarges the table of properties to show all properties possible for the selected Physics type(s). You can use the scroll bars or size the dialog to see all properties.

     Selecting **All Properties** also enables the three **Physics** check boxes: **Electromagnetic**, **Thermal**, and **Structural**. These check boxes are used to filter the properties displayed in the table by physics type. Clearing a check box hides properties of that physics type.
5. You can also enable the **View/ Edit Modifier** check box for **Thermal Modifier**.

   Checking this box causes the Thermal Column to display at the right side of the Properties of the Material table. Selecting **Edit** rather than **None** causes display of the Edit Thermal Modifier dialog box.
6. Select a material property type – **Simple** (i.e., *linear*), **Anisotropic**, **Nonlinear**, **Vector**, or **Custom** – for each property from the **Type** drop-down menu. For Composition, values are **Solid**, **Lamination**, or **Litz Wire**. Some properties only use the Simple type. For thermal materials, select **Solid** or **Fluid** for the **Thermal Material Type**.

   Available selections vary with the property and solution type, and with the **View/Edit** selection as described above.

   For example, for an AC Magnetic active design, if the material is **Simple** (i.e., *linear*), enter values for the following material properties in the **Value** boxes:

   - Relative Permeability (See also *Relative Permeability* in the Technical Notes.)
   - Relative Permittivity
   - Bulk Conductivity
   - Dielectric Loss Tangent
   - Magnetic Loss Tangent
   - Core Loss Model
   - Composition: **Solid**, **Lamination**, or **Litz Wire**.

     > **Note:** The **Litz Wire** selection that appears under Composition is valid only for Maxwell 2D/3D Transient, and 2D/3D AC Magnetic designs. Selecting **Litz Wire** for any other design type will generate an error message.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Mass Density

7. If the material is a *ferrite*, enter a value greater than 0 in the Magnetic Saturation **Value** box. You may also choose to enter values in the Lande G Factor and Delta H **Value** boxes. Because Delta H values are measured at specific frequencies, you should also enter a - Measured Frequency value (default 9.4 Ghz).

   > **Note:** You may enter a variable name or mathematical expression in the **Value** box.

8. To modify the units for a material property, click the **Units** box, and then select a new unit system.
9. For **Material Appearance**, you can check the **Use Material Appearance** box to enable the fields for you to specify a color and transparency for the material.
   - Clicking the **Color** bar opens a standard color selection window.
   - Clicking the **Transparency** box opens a **Transparency** dialog with a text field and slider bar for selection.
10. If you want to add descriptive notes for the new material, click the ellipsis button [**...**] by the **Notes** field. This opens a dialog in which you can enter text. To enter multiple lines of notes, use **CTRL-Enter** to start a new line. (**Enter** by itself will save the Notes field and close the dialog.)
11. Below the list, in the **Calculate Properties for** drop-down menu, do the following if your project contains a permanent magnet:
    a. Select **Permanent Magnet**.
       The **Properties for Permanent Magnet** dialog box appears.
    b. Select which parameters you want to define (**Mu**, **Hc**, **Br**, **Mp**).
    c. Type values in the text boxes, and select the units.
    d. Click **OK**.

    > **Note:** An error message displays if your project type does not support permanent magnets.

12. For transient and 3D AC magnetic projects, you may also be able to set the core loss from the **Calculate Properties for** drop-down menu.
13. Click **Validate Material**.
14. Click **OK**.

    The new material is added to the local material library.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning Material Property Types

Material properties can be assigned using the **View/Edit Material** dialog box, using one of the following material property types:

Simple	The material is homogeneous and linear.
Anisotropic	The material's characteristics vary with direction. • If the material property is **Anisotropic**, its characteristics are defined by its anisotropy tensor. You must define three diagonals for: anisotropic permittivity, electric loss tangent, conductivity, permeability, and magnetic loss tangent. Each diagonal represents a tensor of your model along an axis. These tensors are relative to the coordinate system specified as the object's Orientation property. By specifying different orientations, several objects can share the same anisotropic material but be oriented differently.
Nonlinear	For Relative Permittivity (valid only for 3D Electrostatic designs): • The material's relative permittivity varies according to its associated D-E curve. Refer to Relative Permittivity for a Maxwell Material for details on using nonlinear relative permittivity. For Bulk Conductivity (valid only for 3D DC Conduction designs): • The material's bulk conductivity varies according to its associated J-E curve. Refer to Bulk Conductivity for a Maxwell Material for details on using nonlinear bulk conductivity. For Relative Permeability: • The material's relative permeability varies according to its associated BH curve. Refer to Relative Permeability for a Maxwell or RMxprt Material for details on using nonlinear relative permeability.
Vector	For Magnetic Coercivity: • If the Relative Permeability Type is either Simple or Anisotropic, enter a Value for the Magnitude. • If the Relative Permeability Type is Nonlinear, Magnitude becomes uneditable, and three additional fields of Type Unit Vector: X Component, Y Component, and Z Component appear in which you can enter values or specify functions. For Electric Coercivity, enter a Value for the Magnitude.
Custom	• Magnetostriction can be Linear, Nonlinear, or Anisotropic. • Inverse Magnetostriction can be either Linear or Nonlinear.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Refer to Magnetostriction and Inverse Magnetostriction Materials for details on using these properties.

Select a material property type for each property from the **Type** drop-down menu. Some properties only use the Simple type

If the material property is anisotropic, its characteristics are defined by its anisotropy tensor. You must define three diagonals for anisotropic permittivity, electric loss tangent, conductivity, permeability, and magnetic loss tangent. Each diagonal represents a tensor of your model along an axis.

These tensors are relative to the coordinate system specified as the object's Orientation property. By specifying different orientations, several objects can share the same anisotropic material but be oriented differently.

Relative Permeability for a Maxwell or RMxprt Material

For relative permeability, select one of the following and specify the units:

Type	Value
Simple	Enter a value for the **Relative Permeability**.
Anisotropic	The following parameters appear: • T(1,1) • T(2,2) • T(3,3) Select either a **Simple** or **Nonlinear** type for each of these parameters.
Nonlinear	Click **BH Curve**, and specify the coordinates for the B-H curve (or curves if multiple temperature dependencies are used) in the BH Curve window. Also enables X, Y, and Z Component unit vector fields for Magnetic Coercivity.

1. For the **Nonlinear** type, you can define one of the following three B-H curves via the **B-H Curve** button: a normal B-H curve, a demagnetization curve, or a hysteresis curve; see Specifying BH Curves for Nonlinear Relative Permeability for more details.
2. For the **Anisotropic** type, you can select a **Simple** or **Nonlinear** type for each principal direction. For a **Nonlinear** type, you can define a normal B-H curve or a hysteresis curve, but not a demagnetization curve, via **B-H Curve** button; see Specifying BH Curves for Nonlinear Relative Permeability for more details.
3. The **Magnetic Coercivity** is a vector type that includes a **Magnitude** field and three **Unit Vector** component fields.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For the **Simple** or linear **Anisotropic** Relative Permeability type (select **Simple** type for all three principal directions), the **Magnitude** field is enabled for input to specify a permanent magnet coercivity. A non-zero Magnitude value will enable the X, Y, and Z Unit Vector component fields to specify the permanent magnet's magnetization direction.
- For the **Nonlinear** Relative Permeability type, the Magnitude field is non-editable; it displays the magnetic coercivity value of the input BH curve, and X, Y, and Z Unit Vector component fields are enabled. If the input BH curve is a normal BH curve (zero value for Magnitude field), it is not necessary to specify Unit Vector components. If the input BH curve is a demagnetization curve (non-zero value for Magnitude field), a non-zero Unit Vector component is required to specify the permanent magnet's magnetization direction.
- For the nonlinear **Anisotropic** Relative Permeability type (select at least one **Nonlinear** type for three principal directions), the Magnitude field is non-editable; it displays the maximum magnetic coercivity value of the input BH curves, and the X, Y, and Z Unit Vector component fields are enabled. If the input BH curves are normal BH curves (zero value for Magnitude field), it is not necessary to specify Unit Vector components.
- For the **Nonlinear** or nonlinear **Anisotropic** Relative Permeability type defined with a hysteresis BH curve (non-zero value for Magnitude field), a non-zero Unit Vector can be used to specify the initial magnetization direction, and the successive magnetic field will start from (0, Br) in the initial magnetization direction. In this way, the anisotropic hysteresis model can be used to model an anisotropic permanent magnet without completing the magnetization process. A zero Unit Vector means the successive magnetic field will start from (0, 0).

> **Note:** The Anisotropic type and Hysteresis Curve are not applicable to an RMxprt design.

Specifying B-H Curves for Nonlinear Relative Permeability

For Maxwell 2D and 3D AC magnetic, magnetostatic, and transient solution types, when you define a new material or edit an existing material with a nonlinear relative permeability in the **View/Edit Materials** window, you need to specify the magnetization characteristics using B-H curves.

You can also model temperature dependencies of the nonlinear material property by providing two or more temperature-dependent B-H curves. For temperature dependencies of permanent magnetic materials, refer to Interpolation of Temperature-Dependent Demagnetization Curves.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** The thermal modifier for permeability is disabled when temperature-dependent B-H curves are specified.

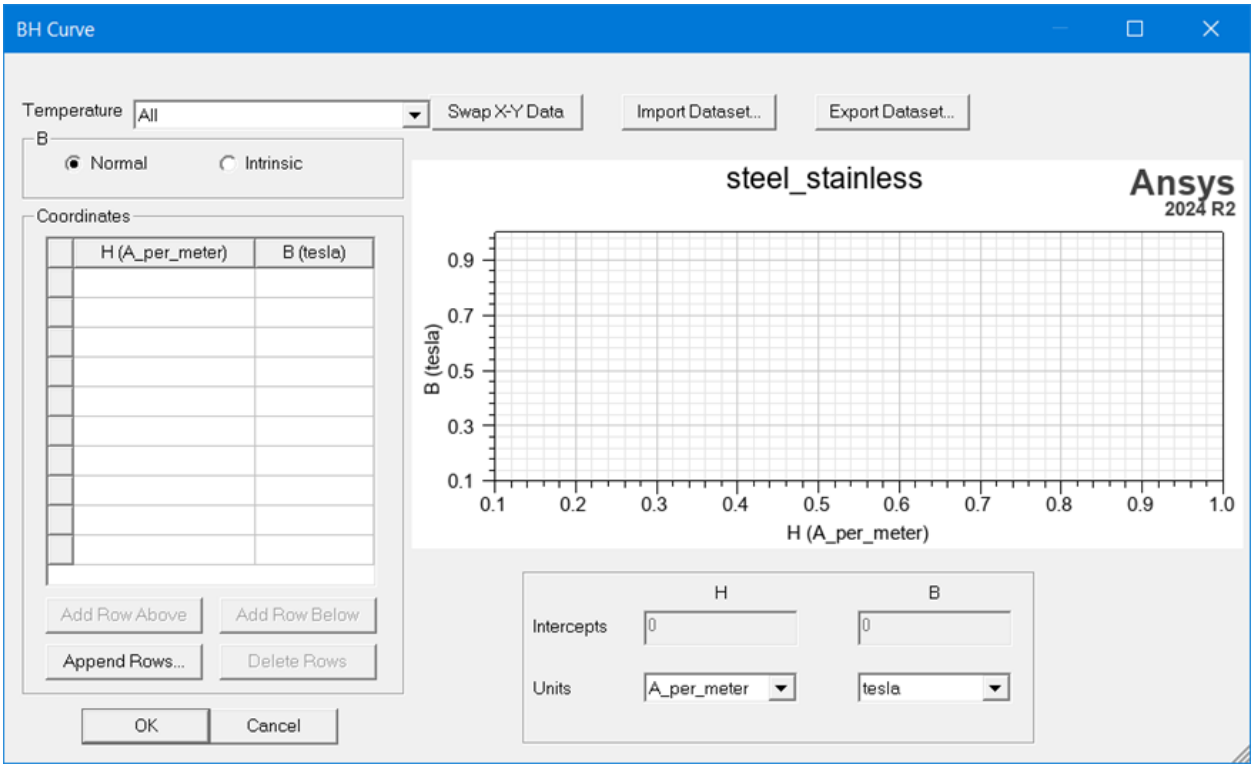
If a user-input B-H curve is used without smoothing, the non-smoothed B-H curve is treated as piece-wise linear between user-input data points inside the software, which may cause nonlinear convergence issues. However, if you chose to use a smoothed B-H curve constructed by the software, the operating point may not exactly reside on the user-input B-H curve. **Smooth B-H curve** is not selected by default. You can select it from the **Solve Setup**, on the **Solver** panel. For isotropic nonlinear materials, you can force Maxwell to use the original user-input data points without smoothing by software by clearing the **Smooth B-H Curve** option.

1. Open the **View / Edit Materials** dialog from the **Edit Materials** window either by:
   - Selecting an existing material that you need to edit, and click **View / Edit Material**.
   - Clicking **Add Material.**

   Either of these actions open the **View/ Edit Materials** window.
2. For the **Relative Permeability** property, do one of the following (depending on the type of material you are defining):
   - Select **Nonlinear** as the **Type**. A **B-H Curve** button appears in the **Value** column.
   - Select **Anisotropic** as the **Type** to display the additional parameters: **T(1,1)**, **T(2,2)**, **T(3,3)**. Selecting **Nonlinear** for any of these additional parameters also causes a **B-H Curve** button to appear in the **Value** columns.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. To input B-H curve(s), click the **B-H Curve** button to open the **BH Curve** window:



You can define one of the following three kinds of B-H curves: normal B-H curve, demagnetization curve, or hysteresis curve:

- A **normal B-H curve** is defined in the first quadrant and starts from the origin (0, 0). Here, “normal” means the curve starts from (0,0), instead of a curve type of **Normal** or **Intrinsic**.
- A **demagnetization curve** is defined in the second quadrant and may extend to the first and/or third quadrants, in which the first and last points are not oddly symmetric.
- A **hysteresis curve** is a branch of a major hysteresis loop: the ascending or descending branch. The ascending branch is defined through the third, fourth, and first quadrants, and the descending branch is defined through the third, second, and first quadrants. For either branch, the first and last points must be oddly symmetric.

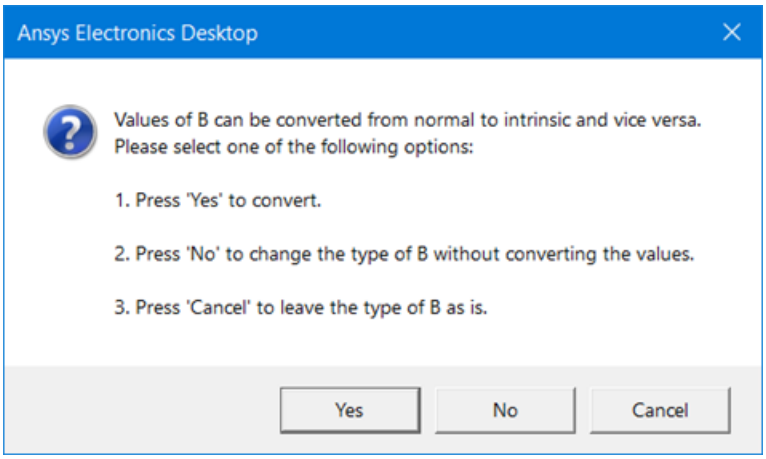
4. Set the **Units** for **H** and **B** by selecting from the drop-down menus. If temperature-dependent B-H curves are used, the selection of units is applied to all of the temperature-dependent curves.
5. Choose the type of curve you want to define by selecting either **Normal** or **Intrinsic**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For a material property without an existing B-H curve definition, the default type will be **Normal**. For a property with existing B-H curve definition, the selected radio button corresponds to the existing B type.

> **Note:**
>
> - The **Intrinsic** B-H curve is supported only in Maxwell 2D/3D magnetostatic, AC magnetic, and transient design types. A material property defined using an Intrinsic B-H curve will fail validation check in all the other product/design types.
> - When an **Intrinsic** B-H curve is added, the **Relative Permeability Value** button label in the **View/Edit Material** dialog box changes to **Bi-H Curve** as visual indication of the type of curve currently defined for the material.

You can change the type at any time. For an existing curve, validation checks are performed on the coordinate list when you attempt to change the type. If the data is not valid, an error message will be displayed and the type of B will not be changed. If data is valid, a query dialog box displays asking if the coordinates should be converted.

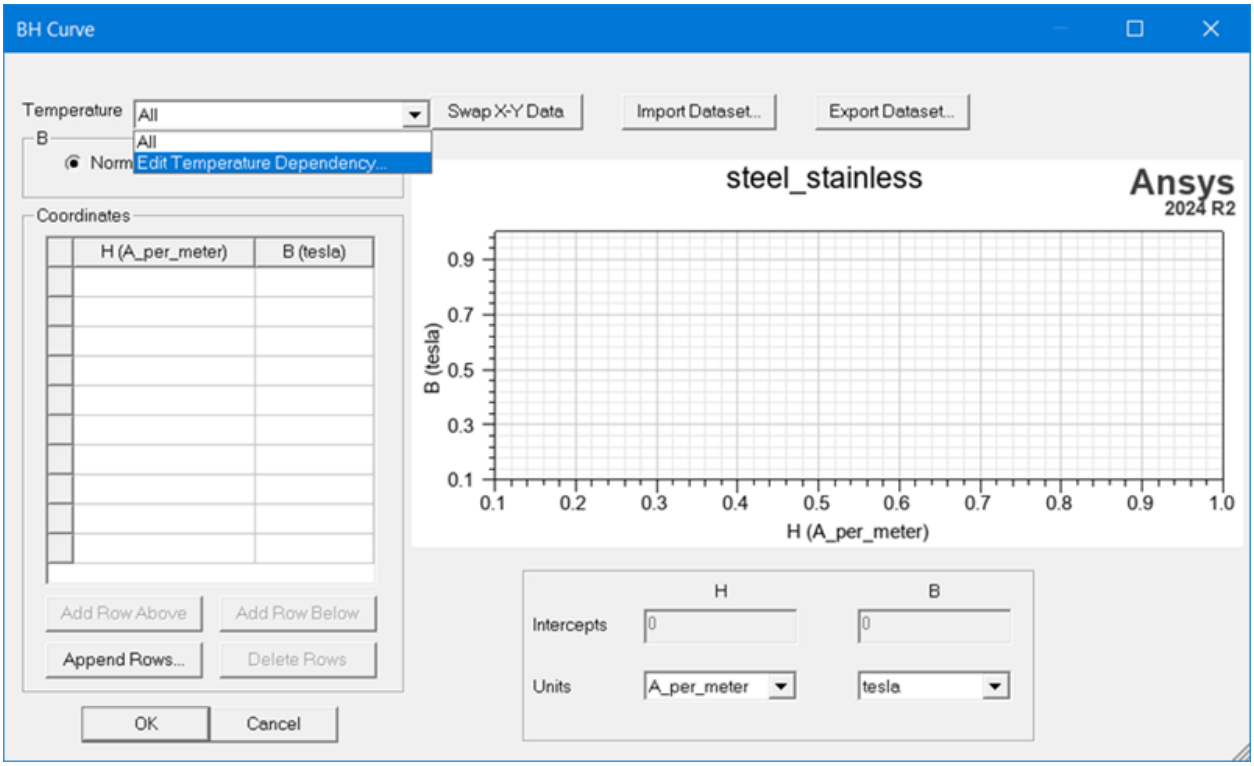


Click **No** if you have specified the B-H coordinates and then realize that you haven't select the desired type.

> **Note:** Changing the type of the B-H curve invalidates all solution data.

6. The initial selections in the **Temperature** drop-down menu include **All** and **Edit Temperature Dependency…**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The default selection is **All**, which means the B-H curve applies to all temperatures – in other words, the B-H curve is *not* temperature-dependent. To define B-H curves with temperature dependency, see step 9.

> **Note:** Temperature Dependency is applicable only to normal B-H and demagnetization curves.

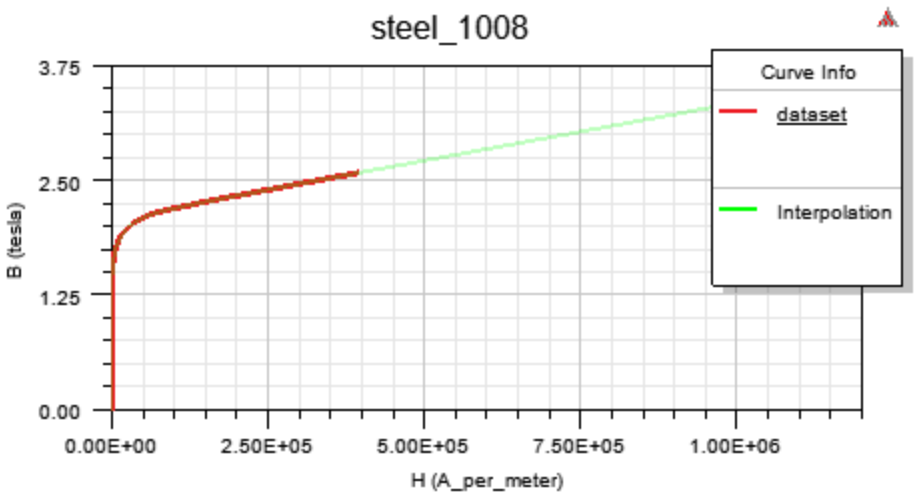
7. Enter B and H values in each row of the **Coordinates** table. Placing the cursor in a table cell enables the **Add Row Above**, **Add Row Below**, and **Delete Rows** buttons. **Append Rows** allows you to specify the number of rows to append to the table.

> **Note:**
>
> - When using temperature-dependent B-H curves, the settings in the B-H Curve dialog apply to the currently selected temperature value.
> - When adding (or editing) temperature-dependent B-H curves, repeat this step for each specified temperature as needed.

Note the following requirements for creating a valid curve:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For a **Normal** B-H curve, the slope of the curve can not be less than that of free space anywhere along the curve.
- For an **Intrinsic** B-H curve, the slope of the curve can not be less than 0.
- The value of B must increase along the curve.
- The initial value of B must be 0 (zero).
- Because B-H operating points in the FEA solution may extend beyond the input B-H data set, the B-H data set is extrapolated in Maxwell. The slope of the last two user-defined data points is used to extrapolate the B-H curve, and thus should be as close to μ_0 as possible.
- The data points representing the B-H curve should have enough points for accurate representation of the curve. Twenty (20) or more points should be specified with increased representation on the "knee" of the curve.
- Normal B-H curves with a positive B value at the first point will be extrapolated (interpolation). Intrinsic curves are not extrapolated.



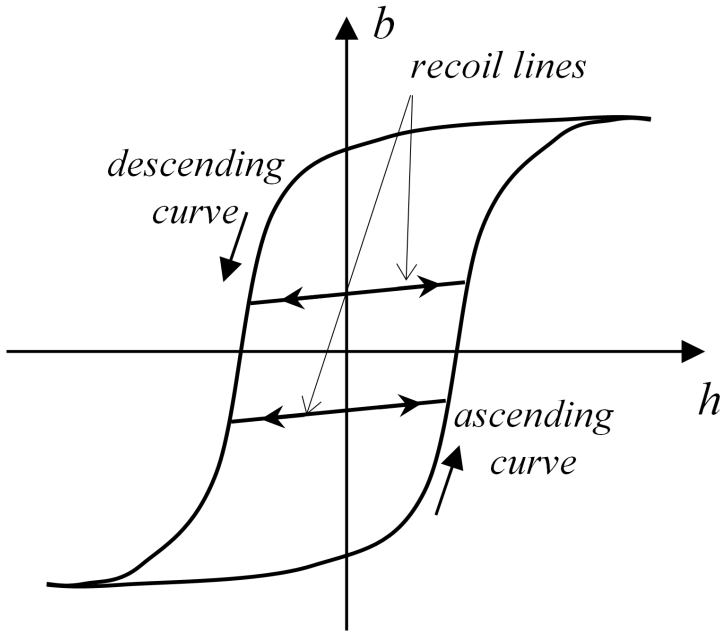
As you enter values, the graph is updated. If you are using multiple temperature-dependent B-H curves, a **Show all curves** check box is present. Checking this box enables all of the temperature-dependent B-H curves to be displayed simultaneously.

- Optionally, click **Import Dataset** to import B-H curve data from a file, and if they are in the wrong columns, click **Swap X-Y Data** to switch the B values and H values in the graphics display. You can also use the SheetScan tool to extract curve data from sources such as manufacturer datasheets to a dataset, which can then be exported to a tab-delimited file, and imported via **Import Dataset**. (Refer to Adding Datasets and Exporting Datasets for related information on working with datasets. Refer to Using SheetScan for working with the SheetScan tool.)

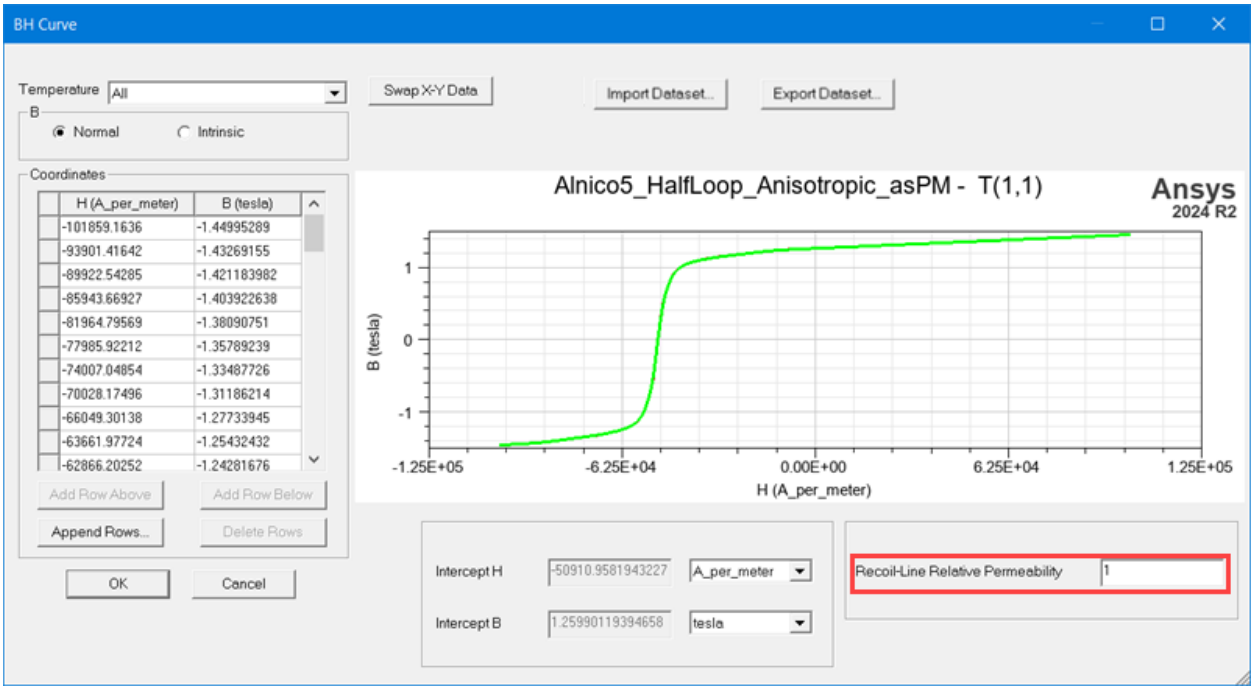
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** When using temperature-dependent B-H curves, selecting **Import Dataset** will import a single curve for the currently selected temperature.

8. **For a hysteresis curve only**, set the **Recoil-Line Relative Permeability**. When a magnetic field is fully saturated, a (B, H) operation point will trace the major hysteresis loop, which consists of an ascending curve and a descending curve. The operation point will trace a recoil line when B or H reverses its direction locally, as shown in the following figure. For most hysteresis materials, the recoil-line relative permeability is 1.0. When you input a hysteresis curve, you need to ensure that the slope between any two adjacent points are not smaller than the recoil-line slope.

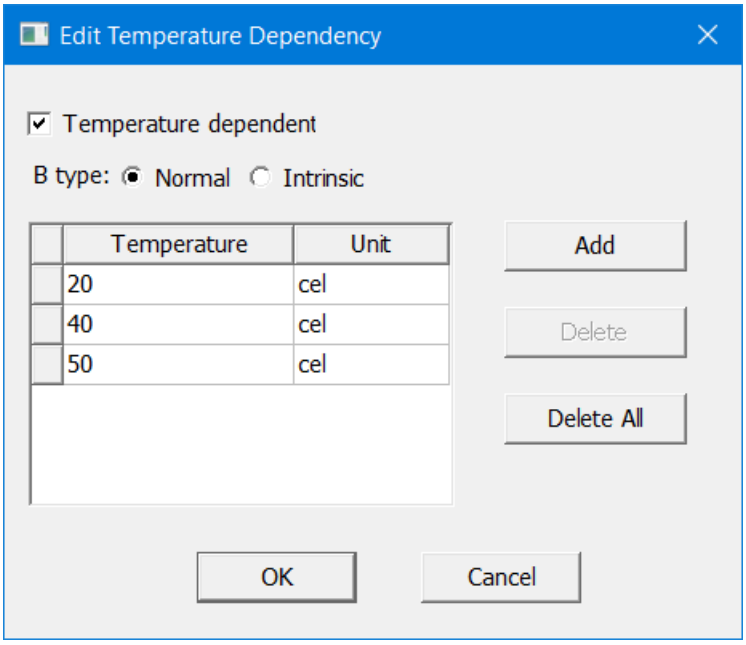


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



9. Optionally, for Maxwell 2D and 3D AC magnetic, magnetostatic and transient solution types, you can model temperature dependencies of the nonlinear material property by providing multiple temperature-dependent B-H curves. At least two B-H curves must be defined when using this feature. If you want to change the **Temperature** setting to specify temperature-dependent B-H curves:
    a. Select **Edit Temperature Dependency** to open the Edit Temperature Dependency dialog box.

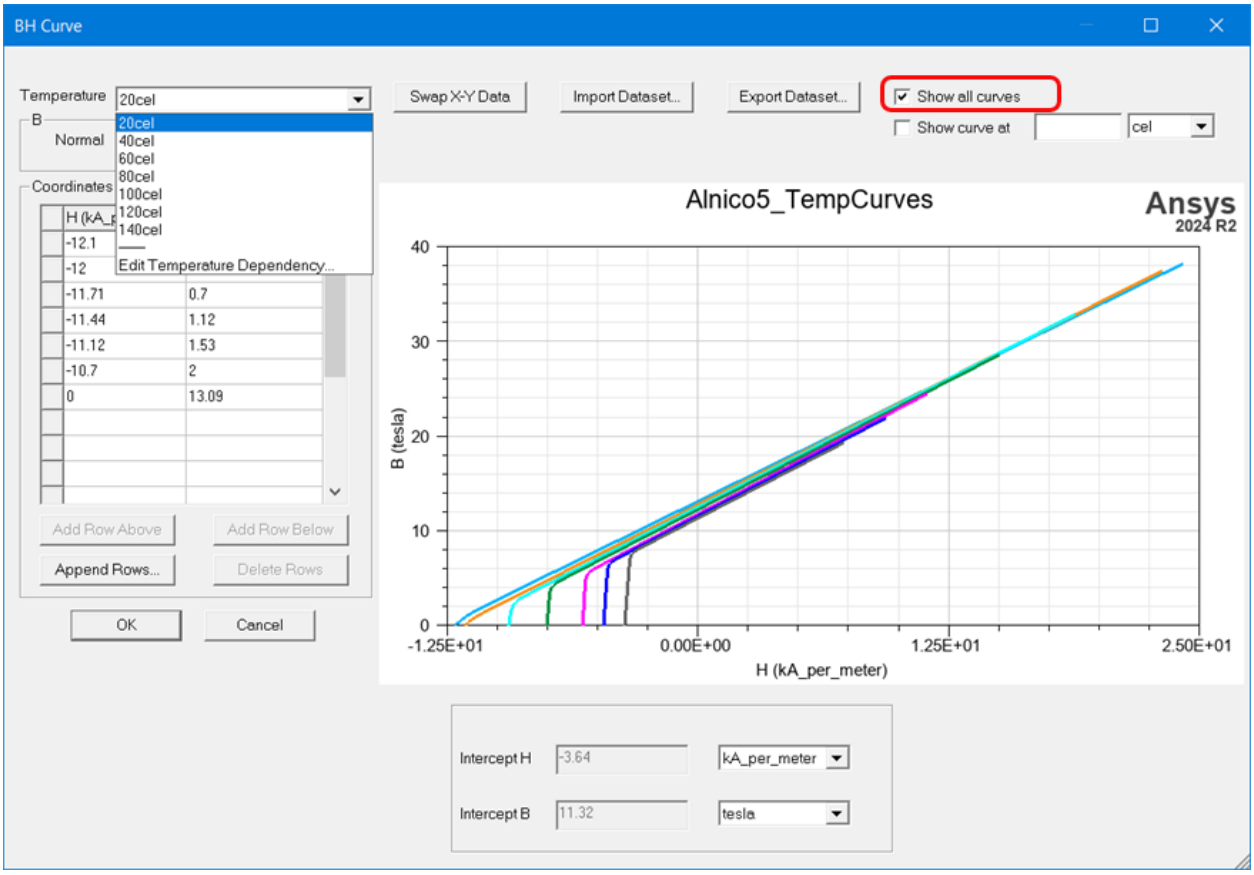
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



b. Use the **Add** button to add rows to the table. In each row, enter the desired **Temperature**, and select the appropriate **Unit**. The **Delete** button can be used to remove selected rows from the table. **Delete All** removes all of the rows.
c. Check **Temperature dependent** to enable temperature-dependent BH curves to be added.
d. Click OK to return to the **BH Curve** dialog box.

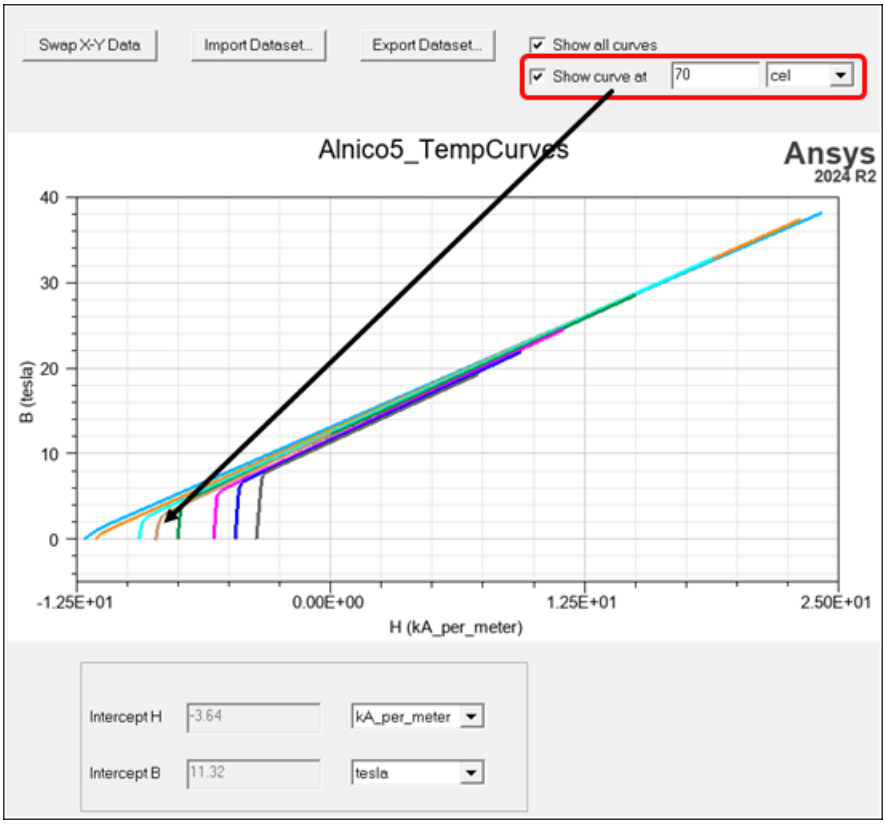
   The **Temperature** drop-down menu now lists all of the temperatures you specified above. Select the one for which you want to enter B and H coordinate values in the following step. Also, a **Show all curves** check box is now present. Checking this enables all of the B-H curves to be displayed simultaneously.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Show curve at** option can be used to show the interpolated curve for a temperature value not specified in the B-H data.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:** The **Show curve at** option only works for demagnetized curves that have **Intercept H** and **B** starting at non-zero values.

10. When finished entering data, click **OK** to close the window.

    An error message displays if a slope is out of tolerance, identifying the data points between which the slope is less than that of free space. Out of tolerance data points must be corrected before you can successfully exit the dialog box. If the slope of the last two points is more than twice the permeability of free space, a warning message is issued during the solution. The slope of the last two points should be as close to μ_0 as possible to represent a fully saturated material.

The B-H curve (or curves if multiple temperature dependencies are used) you have defined is associated with the Relative Permeability property of the material.

> **Note:** When an **Intrinsic** B-H curve is added, the **Relative Permeability Value** button label changes to **Bi-H Curve** as visual indication of the type of curve currently defined for the material.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining Anisotropic Relative Permeability Tensors

Eddy or Frequency Domain Solutions Only

The relative permeability of materials is assumed to be a complex quantity expressed as the following:

$$\mu = \mu' - j \cdot \mu''$$

where μ' is the real part of the complex permeability (same as the usual relative permeability), μ'' is the imaginary part of the complex permeability, and j is the imaginary unit.

The relationship above can also be written as the following:

$$\mu = \mu' \left(1 - j \cdot \frac{\mu''}{\mu'}\right) = \mu' \left(1 - j \cdot tan\delta\right)$$

where tanδ is the magnetic loss tangent.

For anisotropic magnetic materials, a tensor material property needs to be used. The anisotropic relative permeability tensor has the following expression:

$$\overline{\mu} = \begin{vmatrix} \mu'_1(1 - j \cdot tan\delta_1) & 0 & 0 \\ 0 & \mu'_2(1 - j \cdot tan\delta_2) & 0 \\ 0 & 0 & \mu'_3(1 - j \cdot tan\delta_3) \end{vmatrix}$$

Magnetostatic, Eddy, and Transient Solutions Only

Maxwell allows the anisotropic magnetic material to exhibit nonlinear characteristics (magnetostatic, Eddy, and transient solutions only) on all three principle axes of the chosen coordinate system, in which case the respective nonlinear characteristics must be specified.

For example, for laminations with anisotropic behavior in the plane of the lamination, the **Anisotropic** type of material should be specified for the **Relative Permeability** property, and the two respective B-H curves should be entered. For the third direction (lamination stacking direction), one of the specified two B-H curves should also be applicable. The composition field should be set to lamination, and the stacking factor and stacking direction should be specified (see also lamination model).

General

If the material property is anisotropic, its characteristics are defined by its anisotropy tensor. You must define three diagonals for anisotropic permeability. Each diagonal represents a tensor of your model along an axis.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These tensors are relative to the coordinate system specified as the object's Orientation property. By specifying different orientations, several objects can share the same anisotropic material but be oriented differently.

1. In the **Relative Permeability** row in the **View/Edit Material** window, select **Anisotropic** from the **Type** drop-down menu.

   Three rows named **T(1,1)**, **T(2,2)** and **T(3,3)** are added below the **Relative Permeability** row.

2. Enter the relative permeability along one axis of the material's permeability tensor in the **Value** box of the **T(1,1)** row.
3. Enter the relative permeability along the second axis in the **Value** box of the **T(2,2)** row.
4. Enter the relative permeability along the third axis in the **Value** box of the **T(3,3)** row.

If the relative permeability is the same in all directions, use the same values for each axis.

These values can also be defined as variables.

Temperature-Dependent B-H Curves

While a material's temperature dependency can be specified via the thermal modifier, which is described in terms of a mathematical expression, this might not be good enough for applications where it is difficult to describe the temperature dependency in terms of an expression. For such cases, an alternative method is available to let users directly input data for multiple, temperature-dependent, B-H curves. Maxwell uses this data to derive a corresponding B-H curve for each specified temperature using an interpolation algorithm as follows:

Suppose there are m temperature dependent curve inputs in the form of

$$T_0 :: \quad (0,0), (h_{01},\ b_{01}) \cdots\cdots (h_{0i},\ b_{0i}) \cdots\cdots (h_{0N_0},\ b_{0N_0})$$

$$\cdots\cdots\cdots\cdots\cdots$$

$$T_k :: \quad (0,0), (h_{k1},\ b_{k1}) \cdots\cdots (h_{ki},\ b_{ki}) \cdots\cdots (h_{kN_k},\ b_{0N_k})$$

$$\cdots\cdots\cdots\cdots\cdots$$

$$T_m : \ : \quad (0,0), (h_{m1},\ b_{m1}) \cdots\cdots (h_{mi},\ b_{mi}) \cdots\cdots (h_{mN_m},\ b_{mN_m})$$

Where T_0, T_k, and T_m …($T_0 < T_k < T_m$) are reference temperatures at which B-H curves were measured.

The number of *(b,h)* pairs in each BH curve can be different.

In Maxwell, if the local temperature of temperature-dependent nonlinear magnetic material is T.. $(T_k < T < T_{k+1})$, the local B-H curve can be obtained by interpolation:

If $T \le T_0$, B-H curve input at temperature T_0 is applied.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If $T \geq T_m$, B-H curve input at temperature T_m is applied.

If $T_k < T < T_{k+1}$, (h_i, b_i) pairs are calculated as follows:

For a given h_i, $b_{k,i}$, and b_{k+1}, i is the magnetic flux densities obtained from inputted B-H curves at temperatures T_k , T_{k+1}, and b_i is obtained by interpolation:

$$b_i = b_{k,i} + \frac{T - T_k}{T_{k+1} - T_k}(b_{k+1,i} - b_{k,i})$$

Note: Thermal modifier for permeability is disabled when temperature-dependent B-H curves are specified.

Dielectric Loss Tangent for a Maxwell Material

Specify the following for dielectric loss tangent.

Type	Value
Simple	Type a value for the **Dielectric Loss tangent**.
Anisotropic	The following parameters appear: • T(1,1) • T(2,2) • T(3,3) Type a simple value for each.

Defining Anisotropic Dielectric Loss Tangent Tensors

To represent a dielectric that dissipates the power of a high-frequency electric field, enter a dielectric loss tangent, ε"/ε', property value for the material. The smaller the loss tangent, the less lossy the material.

If the material property is anisotropic, its characteristics are defined by its anisotropy tensor. You must define three diagonals for electric loss tangent. Each diagonal represents a tensor of your model along an axis.

These tensors are relative to the coordinate system specified as the object's Orientation property. By specifying different orientations, several objects can share the same anisotropic material but be oriented differently.

If electric loss tangent is anisotropic, do the following:

1. In the **Dielectric Loss Tangent** row in the **View/Edit Material** window, select **Anisotropic** from the **Type** drop-down menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Three rows named **T(1,1)**, **T(2,2)** and **T(3,3)** are added below the **Dielectric Loss Tangent** row.

2. Enter the ratio of the imaginary relative permittivity to the real relative permittivity in one direction in the **Value** box of the **T(1,1)** row.
3. Enter the ratio of the imaginary relative permittivity to the real relative permittivity in the second direction in the **Value** box of the **T(2,2)** row.
4. Enter the ratio of the imaginary relative permittivity to the real relative permittivity in the third orthogonal direction in the **Value** box of the **T(3,3)** row.

If the electric loss tangent is the same in all directions, use the same values for each direction.

These values can also be defined as variables.

The dielectric loss tangent may vary with frequency. To simulate the variances, define a function for the dielectric loss tangent.

Relative Permittivity for a Maxwell Material

Specify the following for Relative Permittivity and specify the units.

Type	Value
Simple	Type a value for the **Relative Permittivity**.
Anisotropic	The following parameters appear: • T(1,1) • T(2,2) • T(3.3) Type a simple value for each.
Nonlinear	The material's relative permittivity varies according to its associated D-E curve. Refer to Specifying a D-E Curve for Nonlinear Relative Permittivity for details.

Defining Anisotropic Relative Permittivity Tensors

Eddy or Frequency Domain Solutions Only

The relative dielectric permittivity of dielectrics is assumed to be a complex quantity, expressed as the following:

$$\varepsilon = \varepsilon' - j \cdot \varepsilon''$$

where ε' is the real part of the complex permittivity (the same as the usual relative dielectric constant), ε" is the imaginary part of the complex permittivity, and j is the imaginary unit.

The relationship above can also be written as the following:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\varepsilon = \varepsilon\left(1 - j \cdot \frac{\varepsilon''}{\varepsilon'}\right) = \varepsilon'\left(1 - j \cdot tan\delta\right)$$

where $tan\delta$ is the dielectric loss tangent.

For anisotropic dielectrics, a tensor material property needs to be used. The anisotropic relative permittivity tensor has the following expression:

$$\overline{\varepsilon} = \begin{vmatrix} \varepsilon_1'(1 - j \cdot tan\delta_1) & 0 & 0 \\ 0 & \varepsilon_2'(1 - j \cdot tan\delta_2) & 0 \\ 0 & 0 & \varepsilon_3'\left(1 - j \cdot tan\delta_3\right) \end{vmatrix}$$

If the material property is anisotropic, its characteristics are defined by its anisotropy tensor. You must define three diagonals for anisotropic permittivity. Each diagonal represents a tensor of your model along an axis.

These tensors are relative to the coordinate system specified as the object's Orientation property. By specifying different orientations, several objects can share the same anisotropic material but be oriented differently.

1. In the **Relative Permittivity** row in the **View/Edit Material** window, select **Anisotropic** from the **Type** drop-down menu.

   Three rows named **T(1,1)**, **T(2,2)** and **T(3,3)** are added below the **Relative Permittivity** row.

2. Enter the material's relative permittivity along one tensor axis in the **Value** box of the **T (1,1)** row.
3. Enter the relative permittivity along the second axis in the **Value** box of the **T(2,2)** row.
4. Enter the relative permittivity along the third axis in the **Value** box of the **T(3,3)** row.

If the relative permittivity is the same in all directions, use the same values for each axis.

These values can also be defined as variables.

Specifying a D-E Curve for Nonlinear Relative Permittivity

When you define a new material or edit an existing material in the **View /Edit Material**s window with a nonlinear relative permittivity, you need to specify the electric flux density vs. electric field strength characteristics (**D-E Curve**).

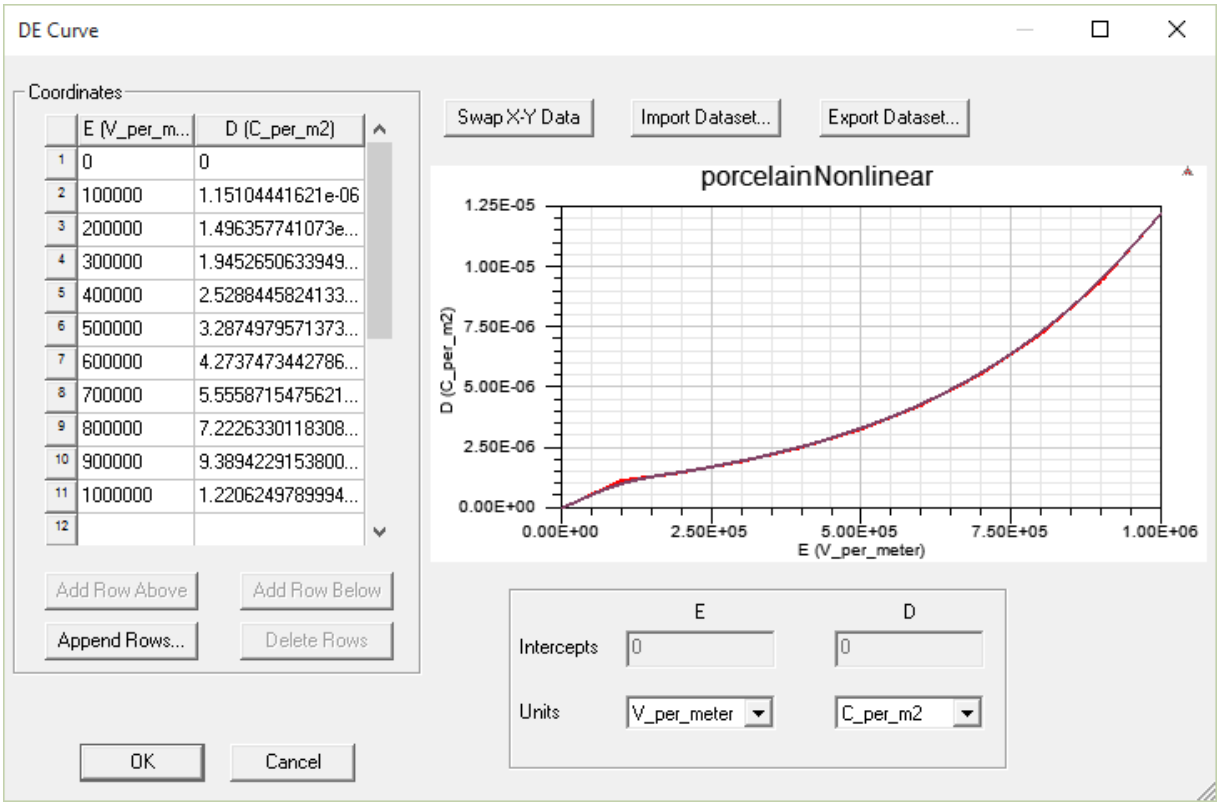
1. Open the **View / Edit Materials** dialog from the **Edit Materials** window either by

   - selecting an existing material that you need to edit, and click **View / Edit Material**.
   - clicking **Add Material.**

   Any of these actions open the **View/ Edit Materials** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. For the **Relative Permittivity** select **Nonlinear** as the **Type**. A **D-E Curve** button appears in the **Value** column.
3. Input the D-E curve by clicking the **D-E Curve** button opens the **DE Curve** dialog box.



4. Set the **Units** for **D** and **E** by selecting from the drop-down menus.

   For a material property without an existing D-E curve definition, the dialog opens with an empty list of coordinates.

5. Enter D and E values in each row of the **Coordinates** table. Placing the cursor in a table cell enables the **Add Row Above**, **Add Row Below**, and **Delete Rows** buttons.

   Note the following requirements for creating a valid curve:

   - The slope of the curve can not be less than 0.
   - The values of D and E must increase along the curve.
   - The initial values of D and E must be 0 (zero) - the curve must begin at the origin.
   - Because D-E operating points in the FEA solution may extend beyond the input D-E data set, the D-E data set is extrapolated. The slope of the last two user-defined data points is used to extrapolate the D-E curve.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The data points representing the D-E curve should have enough points for accurate representation of the curve. A minimum of three data points must be specified. Twenty (20) or more points should be specified with increased representation on the "knee" of the curve.

As you enter values, the graph is updated.

To Add or Edit rows, you can click the following buttons:

- **Add Row Above**
- **Add Row Below**
- **Append Rows** (to specify the number of rows to append to the table)
- **Delete Rows**

6. Optionally, click **Import Dataset** to import D-E curve data from a file, and if they are in the wrong columns, click **Swap X-Y Data** to switch the J values and E values in the graphics display. You can also use the SheetScan tool to extract curve data from sources such as manufacturer datasheets to a dataset, which can then be exported to a tab-delimited file, and imported via **Import Dataset**. (Refer to Adding Datasets and Exporting Datasets for related information on working with datasets. Refer to Using SheetScan for working with the SheetScan tool.)
7. When finished entering data, click **OK** to close the window.

   When you **OK** the dialog box, an error message displays if a slope is out of tolerance, identifying the out of tolerance parameter. Out of tolerance data points must be corrected before you can successfully exit the dialog.

Bulk Conductivity for a Maxwell Material

Specify the following for bulk conductivity and specify the units:

Type	Value
Simple	Type a value for the **Bulk Conductivity**.
Anisotropic	The following parameters appear: • T(1,1) • T(2,2) • T(3,3) You can enter a simple value for each of these parameters.
Nonlinear	The material's bulk conductivity varies according to its associated J-E curve. Refer to *Specifying a J-E Curve for Nonlinear Bulk Conductivity* for details.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining Anisotropic Conductivity Tensors

Electric conductivity can be simple or anisotropic. In either case, the corresponding values are entered in S/m (Siemens per meter).

If the material property is anisotropic, its characteristics are defined by its anisotropy tensor. You must define three diagonals for anisotropic conductivity. Each diagonal represents a tensor of your model along an axis.

These tensors are relative to the coordinate system specified as the object's Orientation property. By specifying different orientations, several objects can share the same anisotropic material but be oriented differently.

1. In the **Bulk Conductivity** row in the **View/Edit Material** window, select **Anisotropic** from the **Type** drop-down menu.

   Three rows named **T(1,1)**, **T(2,2)** and **T(3,3)** are added below the **Bulk Conductivity** row.

2. Enter the conductivity along one axis of the material's conductivity tensor in the **Value** box of the **T(1,1)** row.
3. Enter the conductivity along the second axis in the **Value** box of the **T(2,2)** row.
4. Enter the conductivity along the third axis in the **Value** box of the **T(3,3)** row.

The values of the conductivity along the first and second axis apply to all axes that lie in the xy cross-section being modeled. The values of the conductivity along the third axis applies to the z-component. These values affect current flowing in dielectrics between the conductors.

These values can also be defined as variables.

Specifying a J-E Curve for Nonlinear Bulk Conductivity

> **Note:** Only 3D AC conduction, 3D DC Conduction, and 3D Electric transient solution types support nonlinear conductivity with J-E curve. A material property defined using a J-E curve will fail a validation check in all the other product/design types.

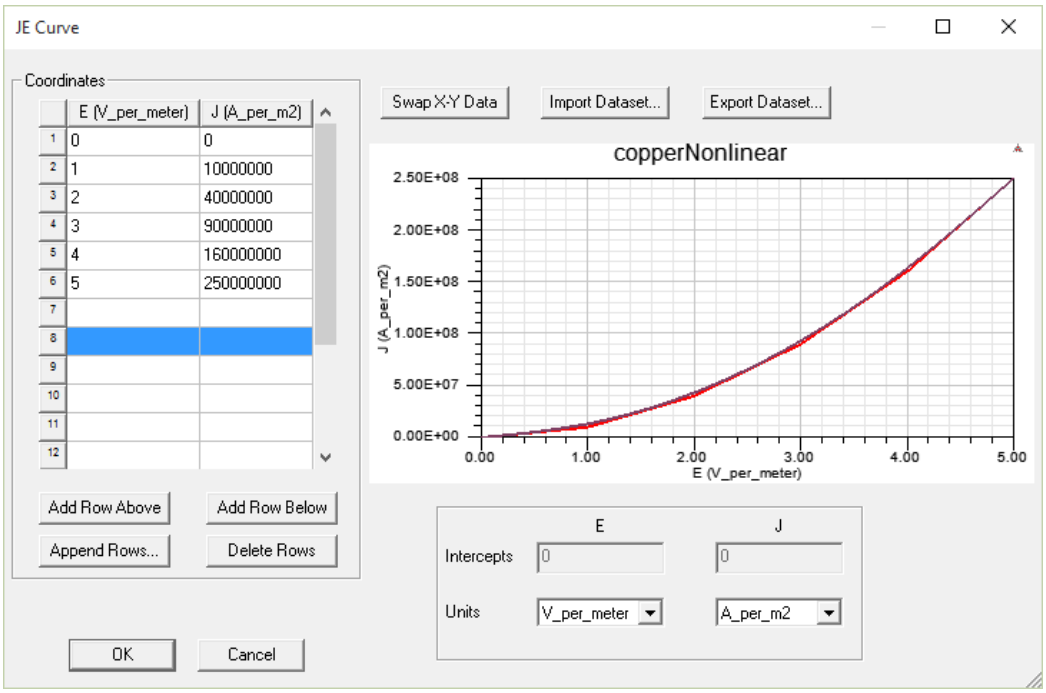
When you define a new material or edit an existing material in the **View /Edit Material**s window with a nonlinear bulk conductivity, you need to specify the current density vs. electric field characteristics (**J-E Curve**).

1. Open the **View / Edit Materials** dialog from the **Edit Materials** window either by
   - selecting an existing material that you need to edit, and click **View / Edit Material**.
   - clicking **Add Material.**

   Any of these actions open the **View/ Edit Materials** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. For the **Bulk Conductivity** select **Nonlinear** as the **Type**. A **J-E Curve** button appears in the **Value** column.
3. Input the J-E curve by clicking the **J-E Curve** button opens the **JE Curve** dialog box.



4. Set the **Units** for **J** and **E** by selecting from the drop-down menus.

   For a material property without an existing J-E curve definition, the dialog opens with an empty list of coordinates.

5. Enter J and E values in each row of the **Coordinates** table. Placing the cursor in a table cell enables the **Add Row Above**, **Add Row Below**, and **Delete Rows** buttons.

   Note the following requirements for creating a valid curve:

   - The slope of the curve can not be less than 0.
   - The values of J and E must increase along the curve.
   - The initial values of J and E must be 0 (zero) - the curve must begin at the origin.
   - Because J-E operating points in the FEA solution may extend beyond the input J-E data set, the J-E data set is extrapolated. The slope of the last two user-defined data points is used to extrapolate the J-E curve.
   - The data points representing the J-E curve should have enough points for accurate representation of the curve. A minimum of three data points must be specified. Twenty (20) or more points should be specified with increased representation on the "knee" of the curve.

   As you enter values, the graph is updated.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To Add or Edit rows, you can click the following buttons:

- **Add Row Above**
- **Add Row Below**
- **Append Rows** (to specify the number of rows to append to the table)
- **Delete Rows**

6. Optionally, click **Import Dataset** to import J-E curve data from a file, and if they are in the wrong columns, click **Swap X-Y Data** to switch the J values and E values in the graphics display. You can also use the SheetScan tool to extract curve data from sources such as manufacturer datasheets to a dataset, which can then be exported to a tab-delimited file, and imported via **Import Dataset**. (Refer to Adding Datasets and Exporting Datasets for related information on working with datasets. Refer to Using SheetScan for working with the SheetScan tool.)
7. When finished entering data, click **OK** to close the window.

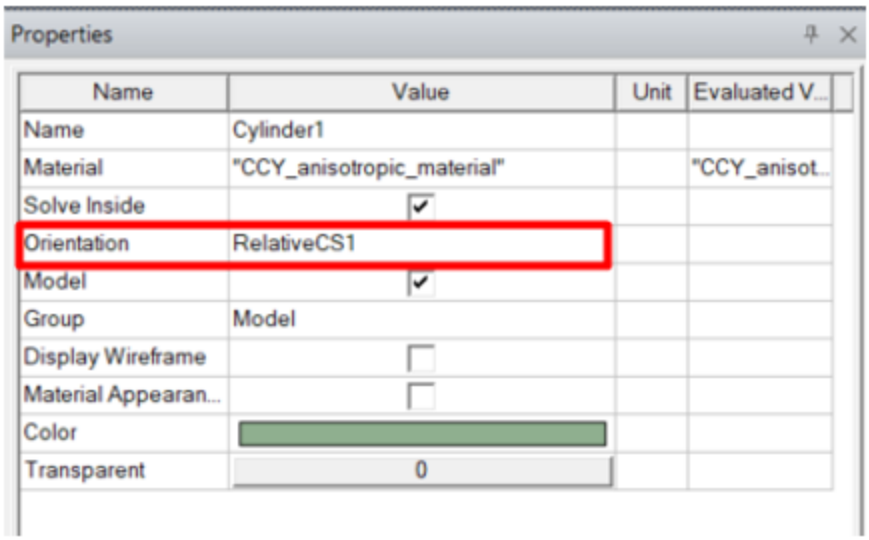
   When you **OK** the dialog box, an error message displays if a slope is out of tolerance, identifying the out of tolerance parameter. Out of tolerance data points must be corrected before you can successfully exit the dialog.

Defining Anisotropic Tensors

If a material property is anisotropic, its characteristics are defined by its anisotropy tensor. Each diagonal represents a tensor of your model along an axis. These tensors are relative to the coordinate system specified as the object's *Orientation* property (when supported). Otherwise, the tensors conform to the global Cartesian coordinate system. By specifying different orientations, several objects can share the same anisotropic material but be oriented differently. Spherical and Cylindrical tensors must be converted to Cartesian coordinates, as described below. In such cases you must specify the coordinate system carefully.

> **Note:** Spatial-dependent material properties are calculated according to coordinates in the 'Orientation CS'.This CS should be set carefully.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



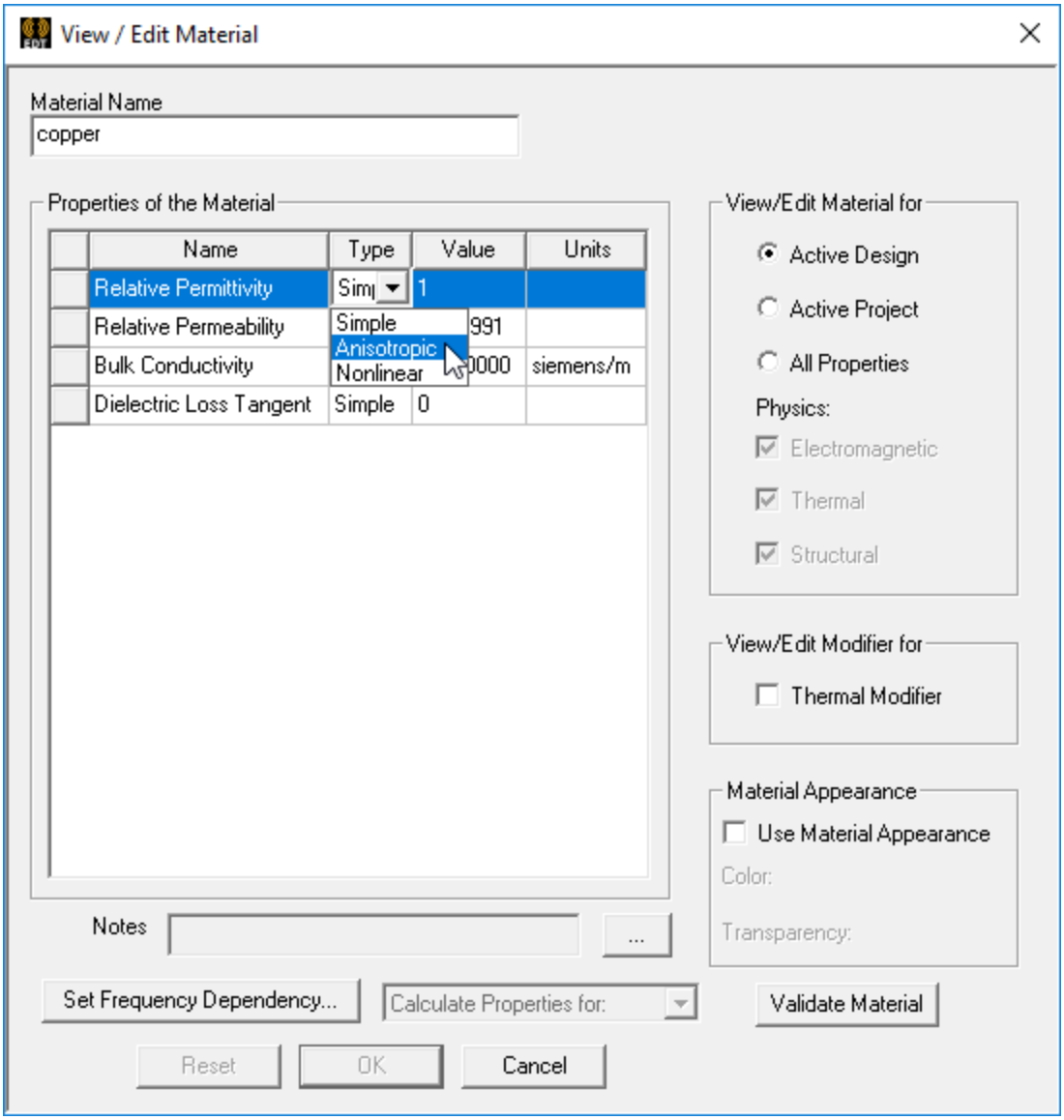
> **Important:**
>
> Anisotropic materials are supported in 2D models but are not available for Q3D.

Assigning Anisotropic Tensors

To assign anisotropic tensors to permittivity, electric loss tangent, conductivity, permeability, and magnetic loss tangent:

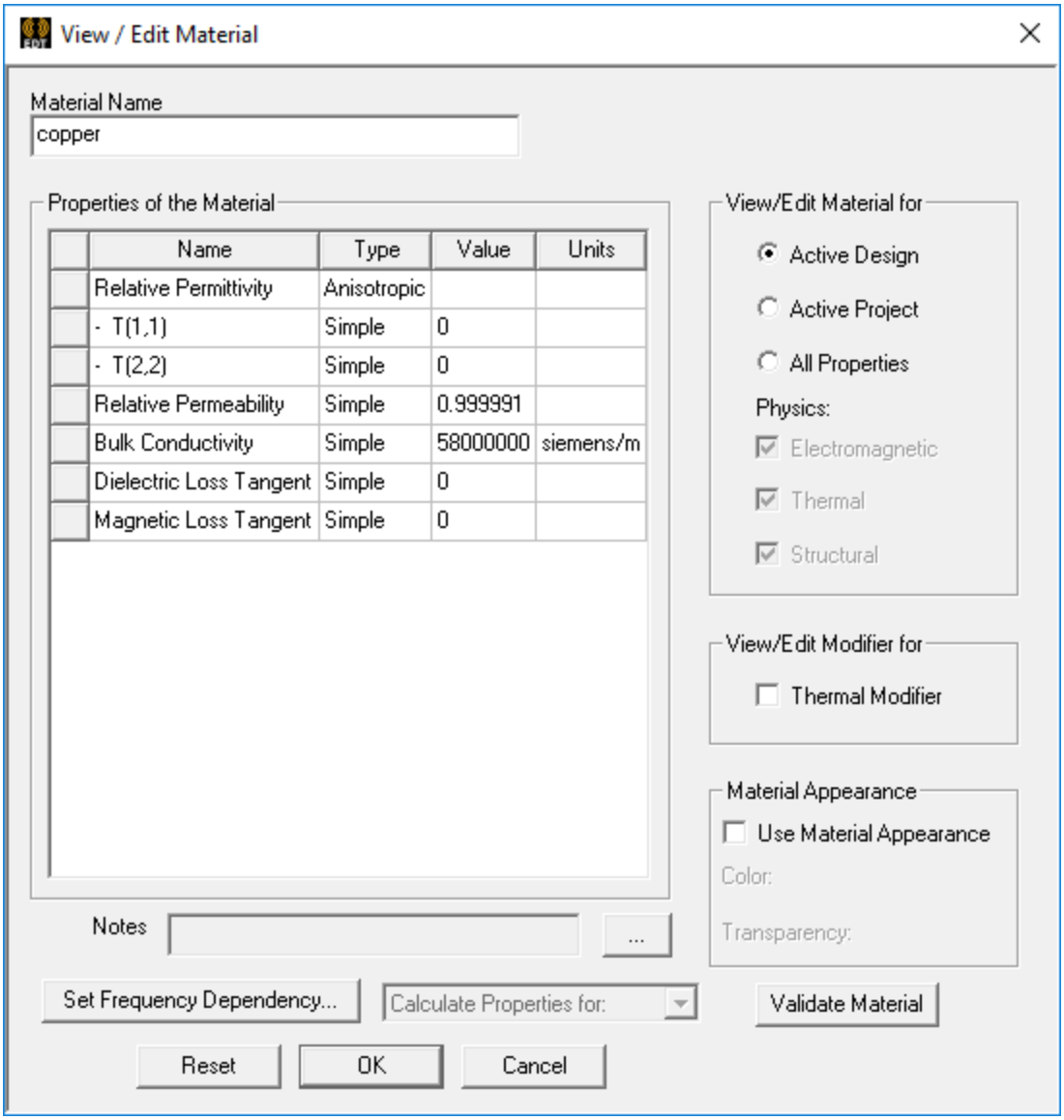
1. From the **View/Edit Material** window, use the **Type** drop-down menu to select **Anisotropic**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Two rows, labeled **T(1,1)** and **T(2,2)** appear, as shown below:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. For each of the new anisotropic property rows, use the **Type** drop-down menu to select either **Simple** or **Nonlinear** (when supported). This setting determines the type of **Value** you can enter.
3. In the **Value** field, enter the permittivity, electric loss tangent, conductivity, permeability, and magnetic loss tangent. along each axis of the material's tensor.
    - For **Relative Permeability** – enter the relative permeability along each axis of the material's permeability tensor. This can be a simple value, a variable, a constant, or a Nonlinear BH Curve.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For **Relative Permittivity** – enter the material's relative permittivity along each tensor axis. This can be a simple value or a variable. If the relative permittivity is the same in all directions, use the same Simple values for each axis.
- For **Conductivity** – enter the material's conductivity along each tensor axis. This can be a simple value or a variable.
- For **Dielectric Loss Tangent** – enter the ratio of the imaginary relative permittivity to the real relative permittivity in one direction. This can be a simple value or a variable.
- For **Magnetic Loss Tangent** – enter the ratio of the imaginary relative permeability to the real relative permeability in one direction. This can be a simple value or a variable.

4. Click **OK** to save the values and return to the **Select Definition** window.

Cylindrical Anisotropic Material Properties: Tensor Conversion from Cylindrical to Cartesian CS

Maxwell accepts definitions for spatial-dependent martial properties only using definitions in a Cartesian coordinate system. Cylindrical and Cartesian bases are related as

$$\begin{bmatrix}\hat{e}_x\\ \hat{e}_y\\ \hat{e}_z\end{bmatrix} = \begin{bmatrix}cos\varphi & -sin\varphi & 0\\ sin\varphi & cos\varphi & 0\\ 0 & 0 & 1\end{bmatrix}\begin{bmatrix}\hat{e}_\rho\\ \hat{e}_\varphi\\ \hat{e}_z\end{bmatrix} = \boldsymbol{M}_{Cyl\to Cart}\begin{bmatrix}\hat{e}_\rho\\ \hat{e}_\varphi\\ \hat{e}_z\end{bmatrix}$$

where M is a transformation matrix.

Tensors, describing material properties, could be transformed accordingly as

$$\bar{\bar{A}}_{xyz} = \boldsymbol{M}_{Cyl\to Cart}\bar{\bar{A}}_{\rho\varphi z}\boldsymbol{M}^T_{Cyl\to Cart} = \begin{bmatrix}cos\varphi & -sin\varphi & 0\\ sin\varphi & cos\varphi & 0\\ 0 & 0 & 1\end{bmatrix}\bar{\bar{A}}_{\rho\varphi z}\begin{bmatrix}cos\varphi & sin\varphi & 0\\ -sin\varphi & cos\varphi & 0\\ 0 & 0 & 1\end{bmatrix}$$

If we assume only the diagonal components as non-zeros for the tensor in the Cylindrical basis

$$\bar{\bar{A}}_{\rho\varphi z} = \begin{bmatrix}A_{\rho\rho} & 0 & 0\\ 0 & A_{\varphi\varphi} & 0\\ 0 & 0 & A_{zz}\end{bmatrix}$$

the resulting tensor in the Cartesian basis could be derived as

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\bar{\bar{A}}_{xyz} = \begin{bmatrix} A_{\rho\rho}cos^2\varphi + A_{\varphi\varphi}sin^2\varphi & (A_{\rho\rho} - A_{\varphi\varphi})sin\varphi cos\varphi & 0 \\ (A_{\rho\rho} - A_{\varphi\varphi})sin\varphi cos\varphi & A_{\rho\rho}sin^2\varphi + A_{\varphi\varphi}cos^2\varphi & 0 \\ 0 & 0 & A_{zz} \end{bmatrix}$$

We should define and using X, Y and Z:

$$\rho = \sqrt{X^2 + Y^2}$$

$$\varphi = \begin{cases} \text{atan}\left(\frac{Y}{X}\right) + \begin{cases} 180° \text{ sign}(Y) & if \quad X < 0 \\ 0 \quad if \; X \geq 0 \end{cases} & if \quad \rho > 0 \\ \\ 0 \quad if \quad \rho = 0 \end{cases} \qquad -180° \leq \varphi \leq 180°$$

Related Maxwell Project variables:

$rho	sqrt(X^2+Y^2)
$phi	if($rho>0, atan(Y/X)+180deg*if(X<0,sgn(Y),0),0)

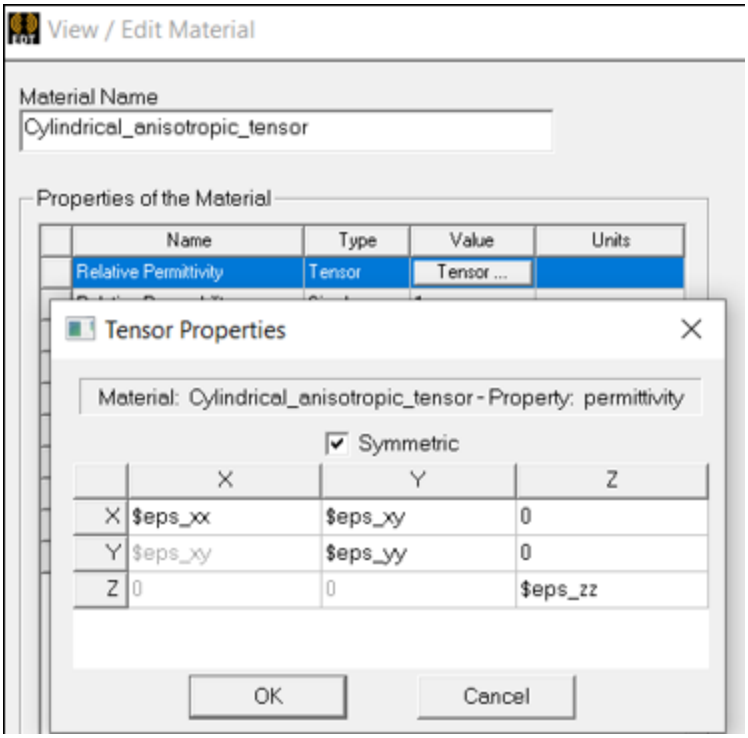
> **Note:** The material characteristics close to Z-axis could be slightly incorrect because of uncertainty of in the case of =0.

First you create all required Project variables.

Name	Value	Unit	Evaluated V...
$eps_rr	4		4
$eps_pp	8		8
$eps_zz	12		12
$rho	sqrt(X^2+Y^2)		*****
$phi	if($rho>0, atan(Y/X)+180deg*if(X<0,sgn(Y),0),0)		*****
$eps_xx	$eps_rr*cos($phi)^2+$eps_pp*sin($phi)^2		*****
$eps_xy	(eps_rr-eps_pp)*sin($phi)*cos($phi)		*****
$eps_yy	$eps_rr*sin($phi)^2+$eps_pp*cos($phi)^2		*****

You can then use the variables to create the material.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

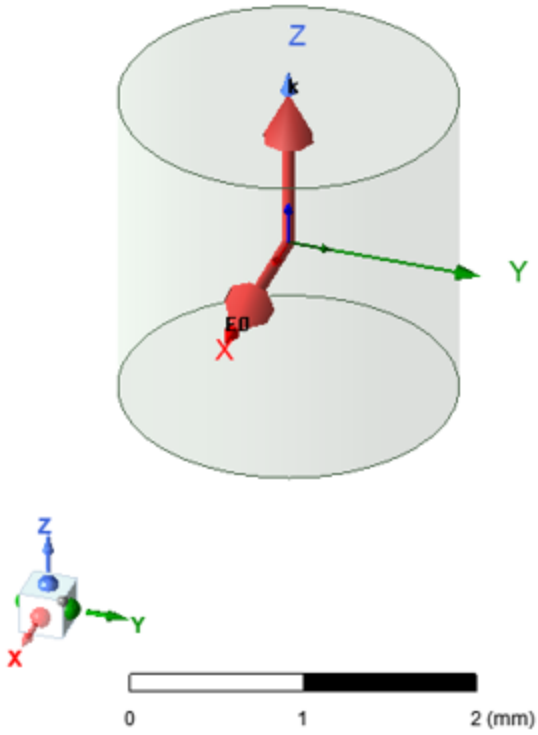


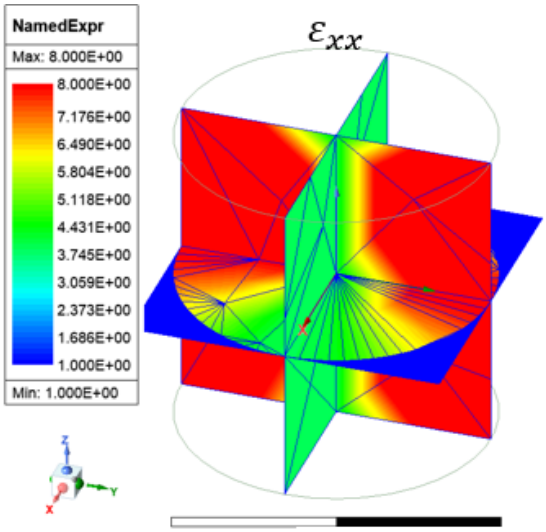
For the test case for cylindrical anisotropic material definition, using converted tensors based on variables:

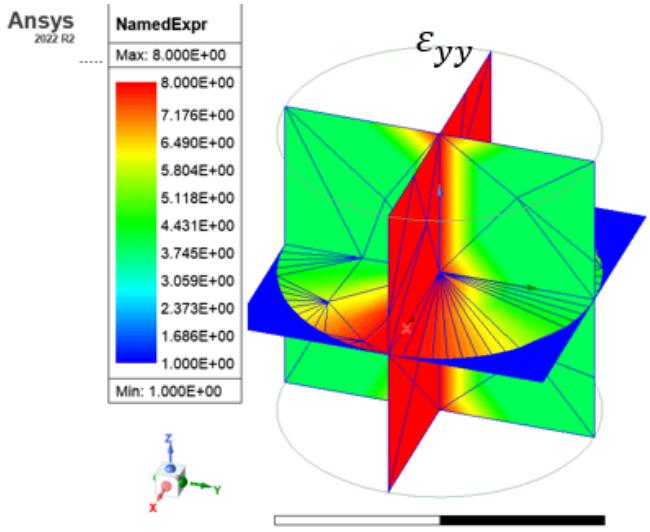
$$\varepsilon_{\rho\rho} = 4 \;\; \varepsilon_{\varphi\varphi} = 8 \;\; \varepsilon_{zz} = 12$$

For this problem type, consider that scattering on a cylinder where diameter and height are equal to 0.2 wavelength and a plane wave excitation along Z-axis.

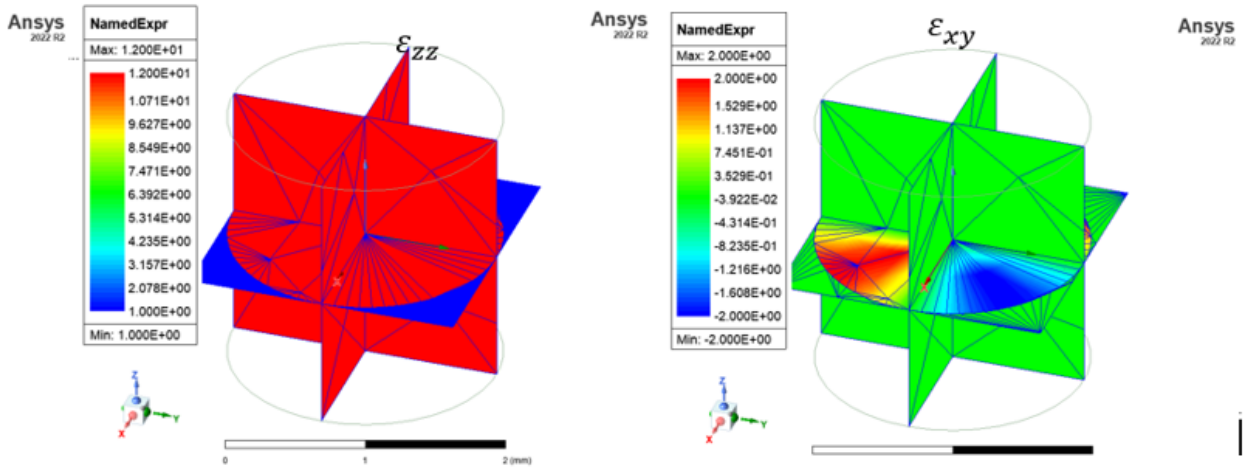
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.







Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Spherical Anisotropic Material Properties: Tensor Conversion from Spherical to Cartesian CS

Maxwell accepts definitions for spatial-dependent martial properties only using definitions in a Cartesian coordinate system. Spherical and Cartesian bases are related as

$$\begin{bmatrix}\hat{e}_x\\ \hat{e}_y\\ \hat{e}_z\end{bmatrix} = \begin{bmatrix}sin\theta cos\varphi & cos\theta sin\varphi & -sin\varphi\\ sin\theta sin\varphi & cos\theta cos\varphi & cos\varphi\\ cos\theta & -sin\theta & 1\end{bmatrix}\begin{bmatrix}\hat{e}_\rho\\ \hat{e}_\theta\\ \hat{e}_\varphi\end{bmatrix} = \boldsymbol{M}_{Sph\to Cart}\begin{bmatrix}\hat{e}_\rho\\ \hat{e}_\theta\\ \hat{e}_\varphi\end{bmatrix}$$

where **M** is a transformation matrix

Tensors, describing material properties, could be transformed accordingly as

$$\bar{\bar{A}}_{xyz} = \boldsymbol{M}_{Sph\to Cart}\bar{\bar{A}}_{\rho\theta\varphi}\boldsymbol{M}^T_{Sph\to Cart}$$

We should define ρ,θ , and φ using X, Y and Z

$$\rho = \sqrt{X^2+Y^2+Z^2} \qquad \rho_{XY} = \sqrt{X^2+Y^2}$$

$$\varphi = \begin{cases}\text{atan}\left(\frac{Y}{X}\right) + \begin{cases}180°\,\text{sign}(Y) & if\ \ X<0\\ 0 & if\ X\geq 0\end{cases} & if \quad \rho_{XY}>0\\ \\ 0 & if \quad \rho_{XY}=0\end{cases}$$

$$\theta = \begin{cases}\text{acos}\left(\frac{Z}{\rho}\right) & if \quad \rho>0\\ \\ 0 & if \quad \rho=0\end{cases}$$

$$0 \leq \theta \leq 180°$$
$$-180° \leq \varphi \leq 180°$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Related Maxwell Project variables

$rho	sqrt(X^2+Y^2+Z^2)
$rho_xy	sqrt(X^2+Y^2)
$theta	if($rho>0, acos(Z/$rho), 0)
$phi	if($rho_xy>0, atan(Y/X)+180deg*if(X<0,sgn(Y),0),0)

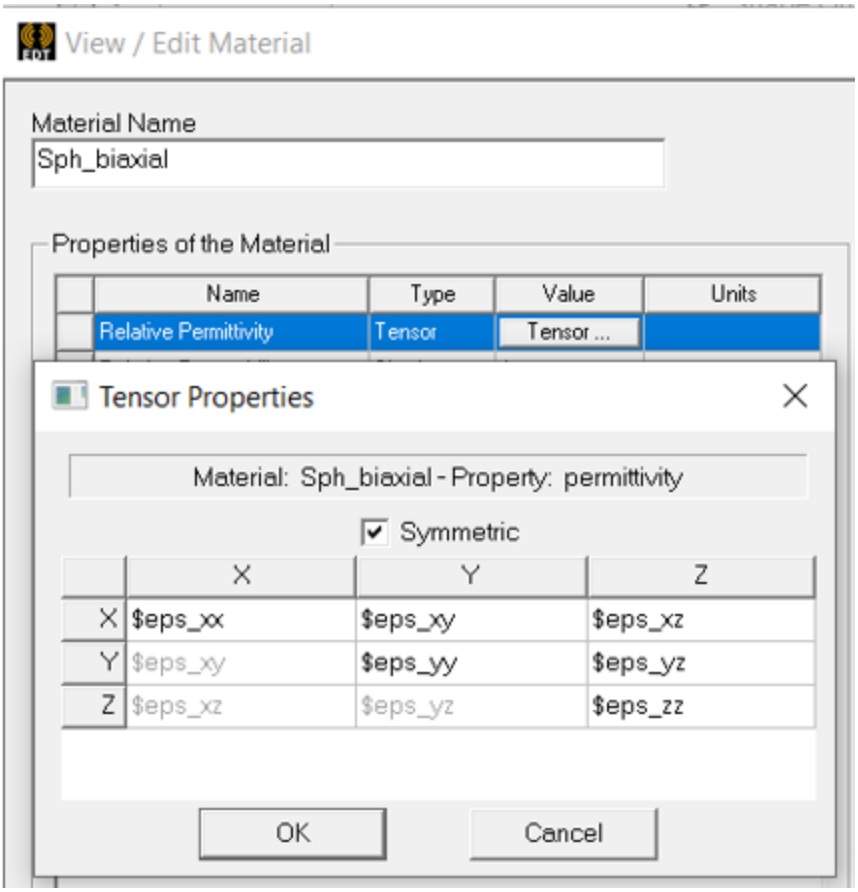
Note: the material characteristics close to the Z-axis could be slightly incorrect because of uncertainty of φ in the case of ρ_{xy}=0.

First you must create the necessary project variables.

Name	Value
$eps_rr	4
$eps_tt	8
$eps_pp	12
$rho	sqrt(X^2+Y^2+Z^2)
$rho_xy	sqrt(X^2+Y^2)
$theta	if($rho>0, acos(Z/$rho), 0)
$phi	if($rho_xy>0, atan(Y/X)+180deg*if(X<0,sgn(Y),0),0)
$eps_xx	($eps_rr*sin($theta)^2+$eps_tt*cos($theta)^2)*cos($phi)^2+$eps_pp*sin($phi)^2
$eps_xy	($eps_rr*sin($theta)^2+$eps_tt*cos($theta)^2-$eps_pp)*sin($phi)*cos($phi)
$eps_xz	(eps_rr-eps_tt)*sin($theta)*cos($theta)*cos($phi)
$eps_yy	($eps_rr*sin($theta)^2+$eps_tt*cos($theta)^2)*sin($phi)^2+$eps_pp*cos($phi)^2
$eps_yz	(eps_rr-eps_tt)*sin($theta)*cos($theta)*sin($phi)
$eps_zz	$eps_rr*cos($theta)^2+$eps_tt*sin($theta)^2

You can then use the variables to create the material.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



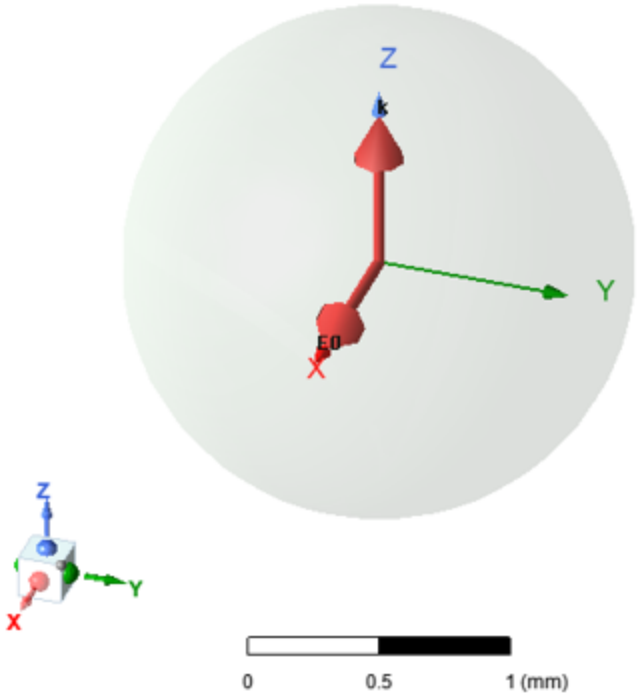
For the test case:

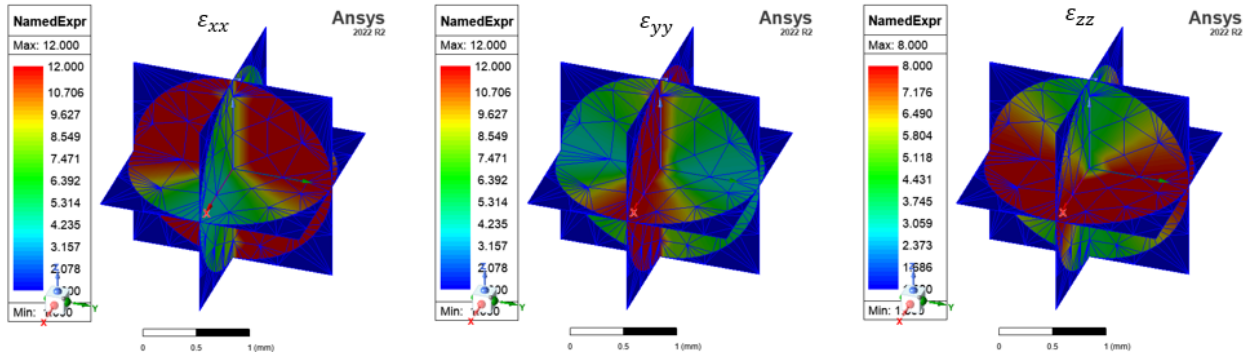
$$\varepsilon_{\rho\rho} = 4 \;\; \varepsilon_{\theta\theta} = 8 \;\; \varepsilon_{\varphi\varphi} = 12$$

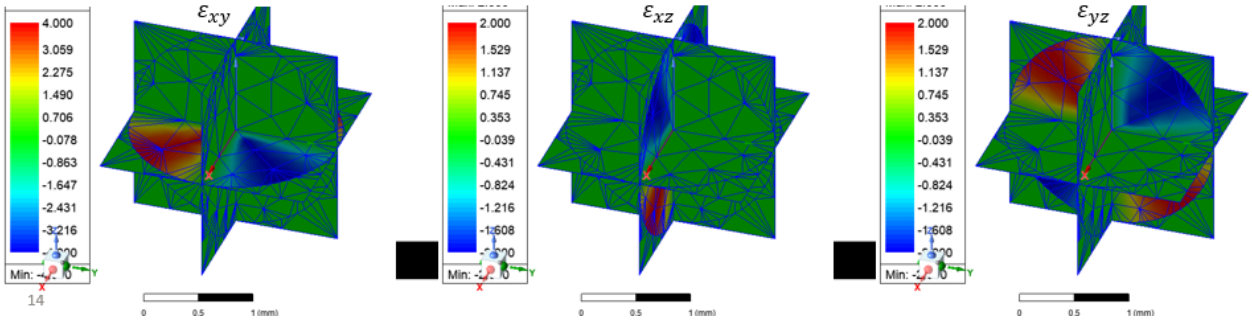
Problem type: scattering on a sphere(diameter is equal to 0.2 wavelength).

Plane wave excitation along Z-axis

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.







Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Magnetic Loss Tangent

To represent a magnetic material such as ferrite that dissipates the power of a high-frequency magnetic field, enter a magnetic loss tangent, μ''/μ', property value for the material. The smaller the loss tangent, the less lossy the material.

Note: If you plan to do a fast-frequency sweep for a design that includes dielectrics, make sure that the dielectric or magnetic loss tangent does not vary significantly over the requested frequency range. If they do, the results may not be what you expect. In cases where the loss tangent does vary significantly over the frequency range in which you are interested, copy and solve the design several times, adjusting the loss tangent and associated frequency range for the copied design so that the loss tangent is relatively stable over the design's requested frequency range.

Note: When magnetic loss tangent is used, the resulting hysteresis core loss will be reflected in the resistance term for both the impedance matrix and the Simplorer Dynamic AC Magnetic ROM.

Imaginary Permeability

AC Magnetic

Some materials exhibit a permeability that includes both a real and imaginary component. The imaginary component is used to model magnetic losses in a time-varying field using the relationship:

$$B = (\mu_r' - j(\mu_r''))\mu_0 H$$

where

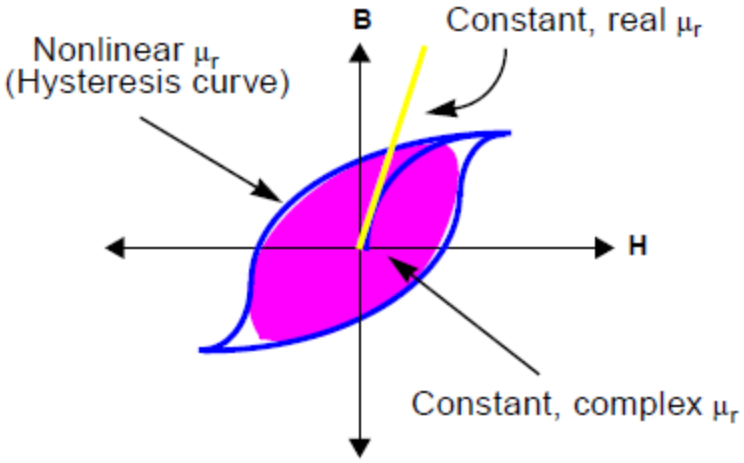
μ_r' is the real component of the relative permeability.

μ_r'' is the imaginary component of the relative permeability.

μ_0 is the permeability of free space.

As shown below, a complex relative permeability causes the B field to lag behind the H field – similar to the behavior of a nonlinear, lossy material. The power loss during this cycle (the color shaded area) is approximately equal to the hysteresis loss (the area within the blue lines). The hysteresis curve for a material with a constant, real permeability (the straight yellow line) is shown as a reference.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When the nature of losses is also due to magnetic hysteresis, the Maxwell application allows you to model such effects using complex permeability by allowing the specification of the tangent of magnetic loss. When a linear material exhibits hysteresis effects, it should be characterized using a complex permeability, as below:

$$\underline{\mu} = \mu' - j\mu'' = \mu'\left(1 - j\tan\delta\right)$$

where tan δ is the magnetic loss tangent, $\frac{\mu''}{\mu'}$

Enter the imaginary relative permeability of a material, μ'', in the **Imag. Permeability** field. The default imaginary permeability of zero is that of a material that exhibits no magnetic loss in a time-varying field.

Defining Magnetic Loss Tangent Tensors

If the material property is anisotropic, its characteristics are defined by its anisotropy tensor. You must define three diagonals for anisotropic magnetic loss tangent. Each diagonal represents a tensor of your model along an axis.

These tensors are relative to the coordinate system specified as the object's Orientation property. By specifying different orientations, several objects can share the same anisotropic material but be oriented differently.

1. In the **Magnetic Loss Tangent** row in the **View/Edit Material** window, select **Anisotropic** from the **Type** drop-down menu.

   Three rows named **T(1,1)**, **T(2,2)** and **T(3,3)** are added below the **Magnetic Loss Tangent** row.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Enter the ratio of the imaginary relative permeability to the real relative permeability in one direction in the **Value** box of the **T(1,1)** row.
3. Enter the ratio of the imaginary relative permeability to the real relative permeability in the second direction in the **Value** box of the **T(2,2)** row.
4. Enter the ratio of the imaginary relative permeability to the real relative permeability in the third direction in the **Value** box of the **T(3,3)** row.

If the magnetic loss tangent is the same in all directions, use the same values for each direction. These values can also be defined as variables.

Defining Variable Material Properties

When defining or modifying a material's properties, each material property value in the **View/Edit Material** dialog box can be assigned a project variable. Simply type the project variable's name in the appropriate **Value** box. Project variables are used for material properties because materials are stored at the project level.

For example, define a project variable with the name **MyPermittivity** and define its value as **4**. To assign this property value to a material, type **$MyPermittivity** in the **Relative Permittivity Value** box for the material. Be sure to include the prefix **$** before the project variable name, which notifies the software that the variable is a project variable.

> **Note:**
>
> By default, not all of the available properties are displayed in the materials table. Only the properties commonly used by the product are displayed. To view the complete table of properties, see Filtering Materials.

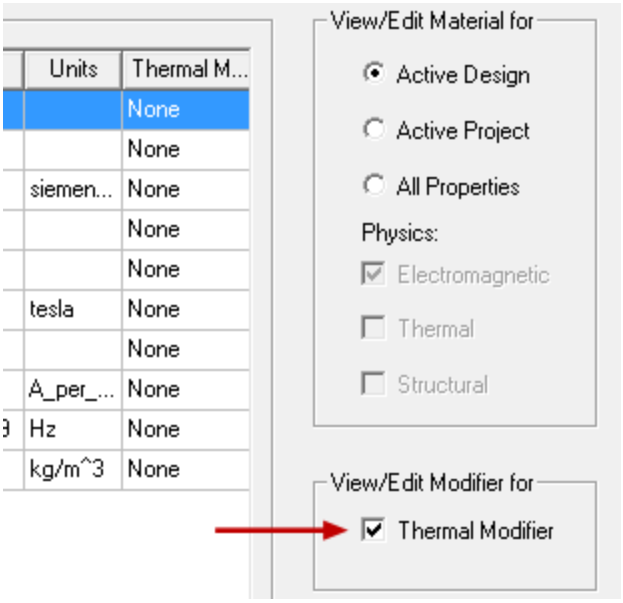
Specifying Thermal Modifiers

Thermal modifiers are multiplicative in nature and assigning them introduces spatial dependence. Because you cannot have spatial dependence and causality, Ansys Electronics Desktop ignores Auto Causal Materials options when a thermal modifier exists.

To specify thermal modifiers for a material:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

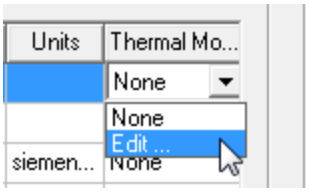
1. In the **View/Edit Material** dialog box, you must enable the **Thermal Modifier** check box:



This option causes the properties table to expand to include a Thermal Modifier column.

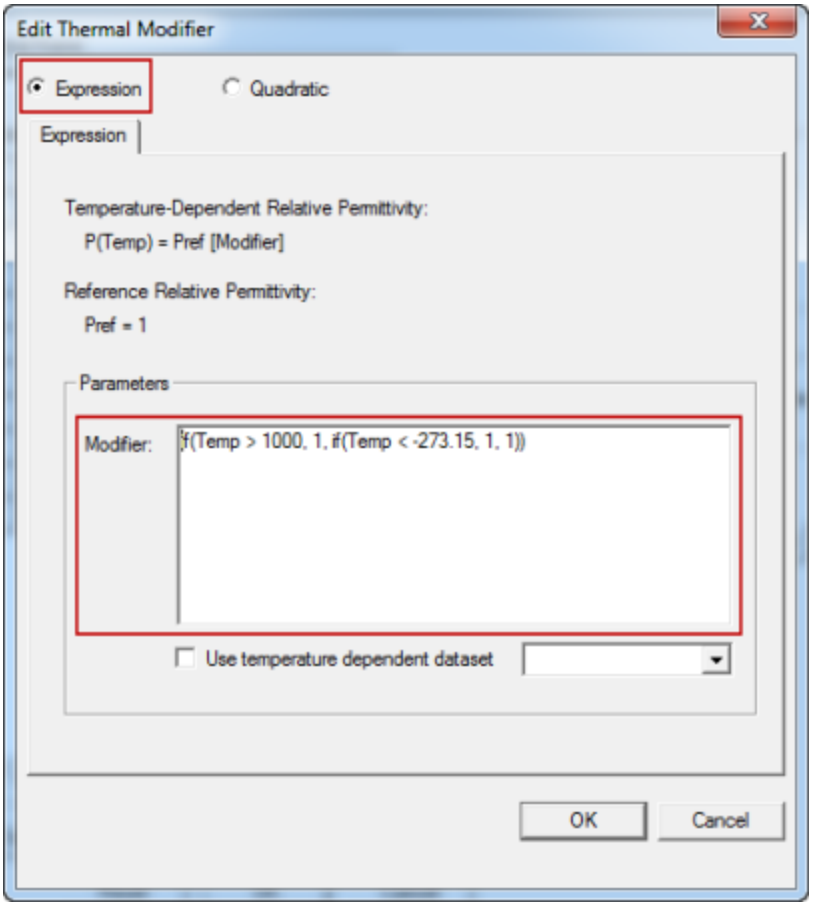
By default, the **Thermal Modifier** option is set to **None** for all material properties.

2. Click a **Thermal Modifier** cell and select **Edit...** from the drop-down menu:



The **Edit Thermal Modifier** dialog box appears with the **Expression** option selected.

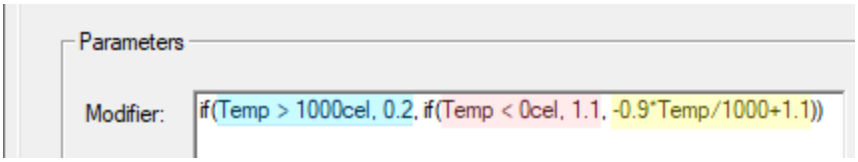
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



With this option selected, the **Modifier** text box is displayed under **Parameters**. This text box contains a conditional expression template, which you can use as a starting point for creating your own expression. The property for which the multiplier is being defined and its nominal value are indicated above the **Parameters** section.

- With **Expression** selected, you can write an equation for a thermal modifier in the **Modifier** text box.

  The following example defines a multiplier that varies linearly from 1.1 (at 0° C) to 0.2 (at 1000° C). The multiplier is constant at 1.1 for temperatures below 0° C and constant at 0.2 for temperature above 1000° C:

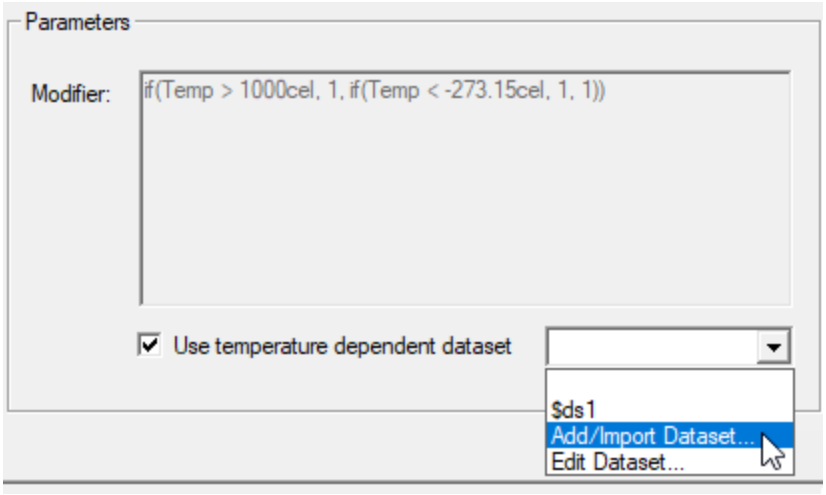


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this example, the linear equation **-0.9 * Temp / 1000 + 1.1** (highlighted in yellow) defines the multiplier for the range 0° C ≤ Temp ≤ 1000° C.

The default unit of temperature in thermal modifier expressions is Celsius (cel). However, you can override the units by appending the appropriate abbreviation to the numbers in the expression (mkel, ckel, dkel, kel, cel, rank, fah). Model temperatures, material properties, and thermal modifiers need not have consistent units; values are converted automatically as needed.

- Check **Use temperature dependent data set** to disable the Modifier text box. You can then use the drop-down menu at the bottom-right corner of the dialog box to select an existing project dataset or to create a new one by choosing **Add/Import Dataset**.

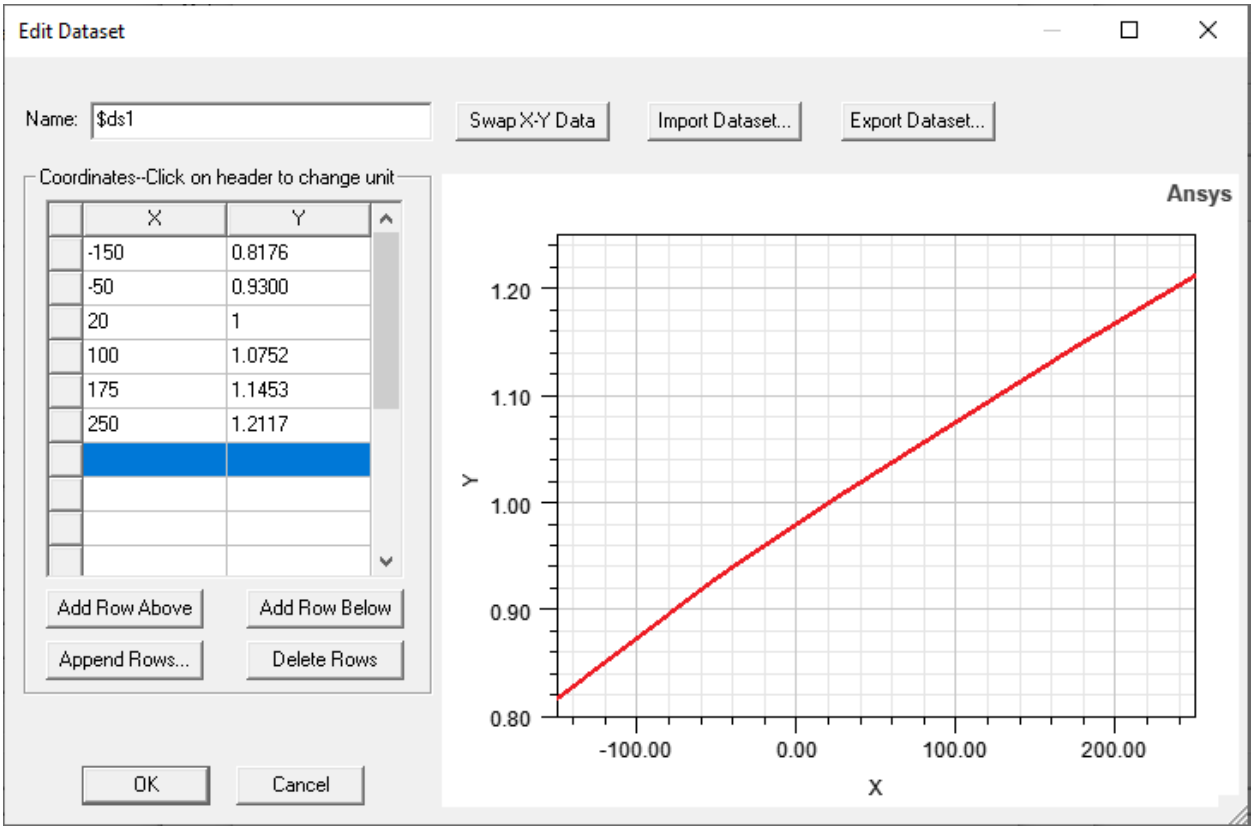


This command lets you define the thermal modifier dataset (a set of X and Y values). Alternatively, if a suitable project dataset already exists, it will be available for selection from the same drop-down menu.

When creating a new dataset, the **X** values are temperatures, and the **Y** values are the corresponding thermal modifiers. The value of the nominal material property is multiplied by this thermal modifier to determine the resultant property at a given temperature. The modifier is interpolated linearly between specified data points.

The following example is a dataset defining thermal multipliers for the thermal expansion coefficient (TEC) of an aluminum-beryllium alloy. The temperature range is -150 to 250° C, and the nominal value (corresponding to 20° C) is 1.37e-5/°C:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Based on these multipliers, the resultant TEC at -150° C is approximately 1.12e-5 (= 1.37e-5 * 0.8176), and the TEC at 250° C is 1.66e-5 (= 1.37e-5 * 1.2117).

In addition to filling the table of values manually, you can also **Import** an existing tab-delimited file to create the dataset.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

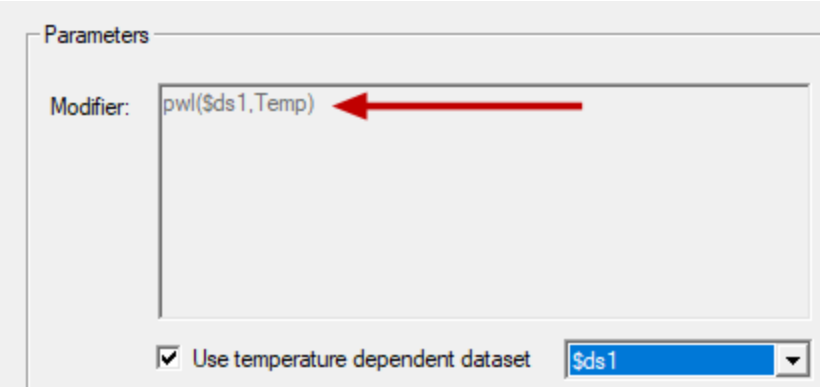
Important:

When you add a dataset from the **Edit Thermal Modifier** dialog box, you cannot specify the temperature units. The numbers in the **X** column will be interpreted as degrees Celcius and cannot be changed. Also, you cannot append a unit abbreviation (such as "fah" or "cel") to the numbers. You must enter temperature values in degrees Celcius for correct results. The values in the **Y** column are multipliers and therefore dimensionless.

Alternatively, you can create a dataset based on any desired temperature units before defining your temperature-dependent material properties. To do so, click **Project> Datasets** from the menu bar and then choose **Add**. If your project includes a design type that supports 3D datasets, a submenu appears offering a choice of a 1D or 3D dataset. If you see this choice, click **1D**. In either case, the **Add Dataset** dialog box appears.

For the dataset to be used later as a thermal modifier, you must click the **X** column header and choose **Temperature**. Then, click the **X** column header again and choose the desired temperature units. Finally, populate the table with your temperature and multiplier values and then click **OK** and **Done**. The dataset will now be available for selection from the drop-down menu in the **Edit Thermal Modifier** dialog box.

After choosing or creating a dataset, it appears in the **Modifier** field of the **Edit Thermal Modifier** dialog box, as shown in the following image:



The syntax of the dataset expression is explained as follows:

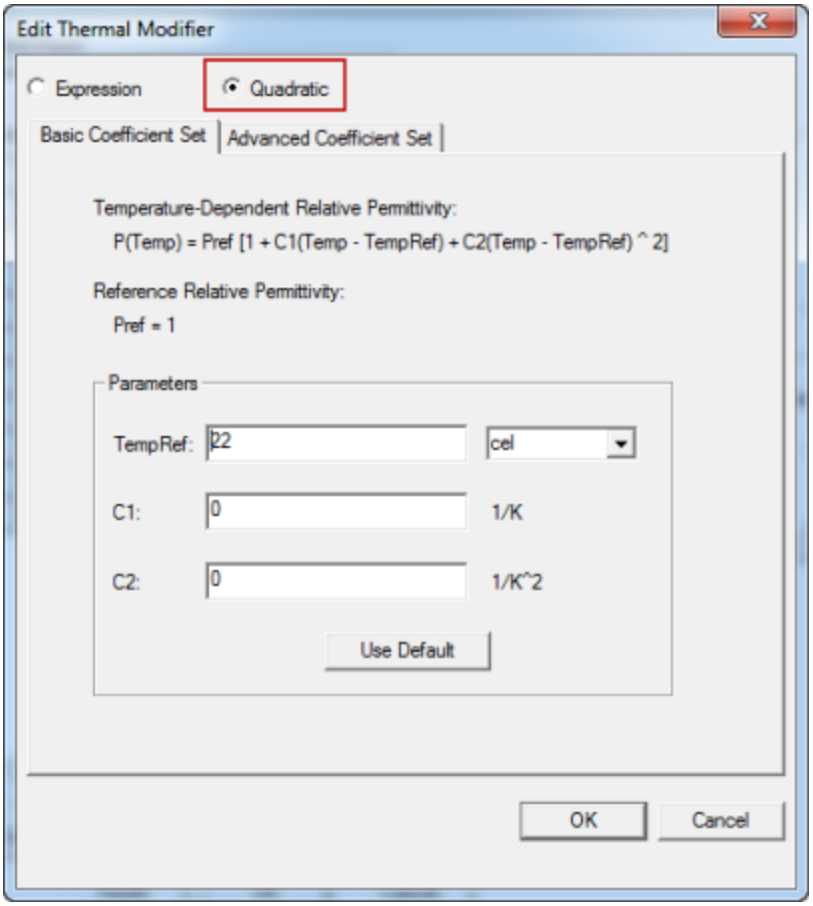
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **pwl:** Type of dataset – Piecewise linear
- **$ds1:** Name of an existing dataset – "$" indicates a *project* dataset, which is a requirement for thermal modifiers
- **Temp:** Type of X value – Temperatures

The Y values in this case are simple multipliers and therefore unitless.

> **Note:**
>
> To edit an existing dataset, select **Edit Dataset** from the drop-down menu in the **Edit Thermal Modifier** dialog box. The **Datasets** dialog box appears. Select the dataset to modify and click **Edit**.

3. Alternatively, select the **Quadratic** option to display the tabs for **Basic Coefficient Set** and **Advanced Coefficient Set** (as shown below):

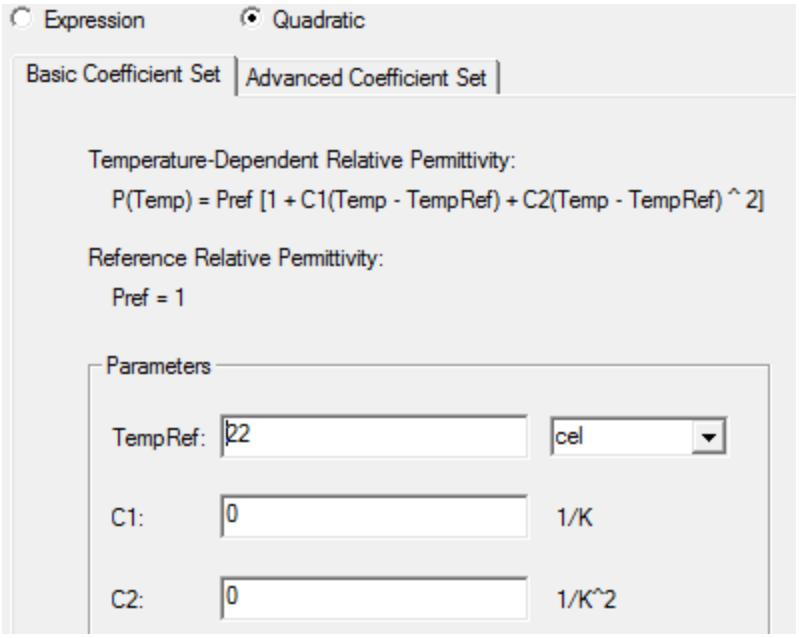


- Under the **Basic Coefficient Set** tab, you can edit fields for the TempRef and units, and fields for C1 and C2 for the following equation:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

P(Temp) = Pref[1+ C1(Temp - TempRef) + C2(Temp - TempRef)^2]

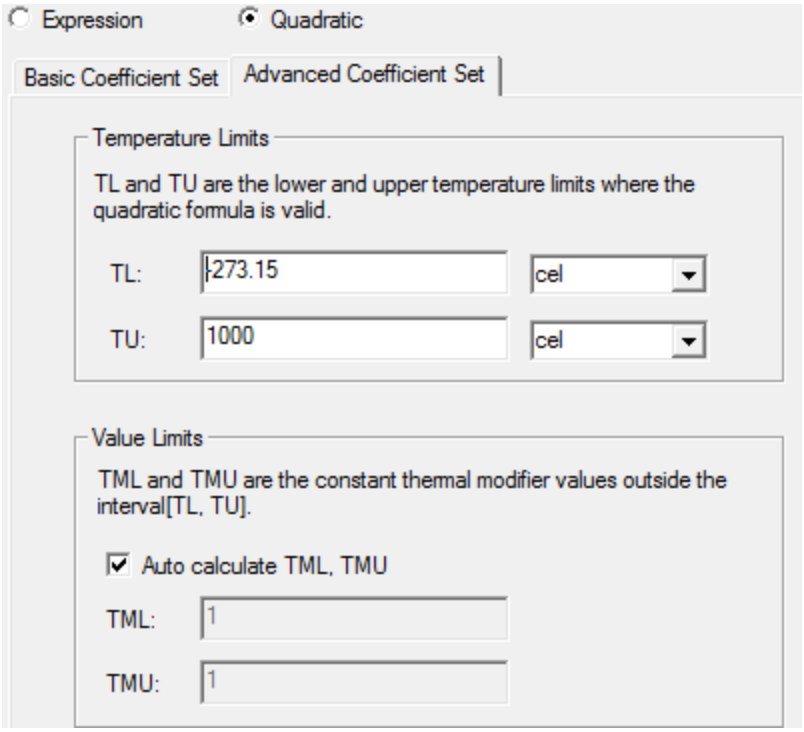
where TempRef is 22 cel by default and where the Pref is defined as the reference relative permittivity.



> **Note:**
>
> The coefficients, C1 and C2, should be negative to yield physical results.

- Under the **Advanced Coefficient Set** tab, you can edit fields for lower and upper temperature limits (TL and TU, respectively) and select their units from the drop-down menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can also edit the constant value limit for the thermal modifier values outside the limits. By default, these are automatically calculated. Uncheck the Auto Calculate TML and TMU to specify new values for thermal modifier lower (TML) and thermal modifier upper (TMU).

4. Click **OK** to accept your edits and return to the View/Edit materials dialog box.

Defining Material Properties as Expressions

When defining or modifying a material's properties, each material property value in the **View/Edit Material** window can be assigned a mathematical expression. Simply type the expression in the appropriate **Value** box. Expressions typically contain intrinsic functions, such as sin(x), and arithmetic operators, such as +, -, *, and /, but do not include project variables.

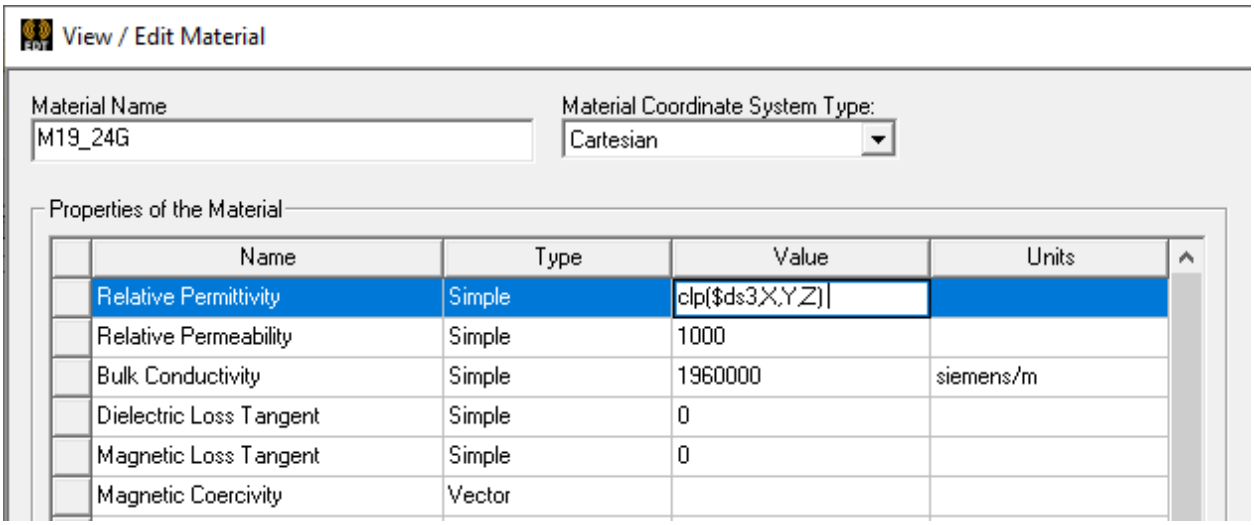
> **Note:**
>
> By default, not all of the available properties are displayed in the materials table. Only the properties commonly used by the product are displayed. To view the complete table of properties, see Filtering Materials.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining Functional Material Properties

Any material property that can be specified by entering a constant can also be specified using a mathematical function. This is useful when you are defining a material property whose value is given by a mathematical relationship — for instance, one relating it to frequency or another property's value. When defining or modifying a material's properties, simply type the name of the function in the appropriate **Value** box.

Optionally, for 2D and 3D transient designs, you can enter an expression for a 3D(X,Y,Z) dataset in the Value box for the properties of conductivity, permeability, and permittivity, as shown in the figure below. The 3D dataset models the spatially varying material property change with respect to location along the object. For detailed information on using 3D(X,Y,Z) datasets, please refer to Setting a Spatial Modifier.



Note:

By default, not all of the available properties are displayed in the materials table. Only the properties commonly used by the product are displayed. To view the complete table of properties, see Filtering Materials.

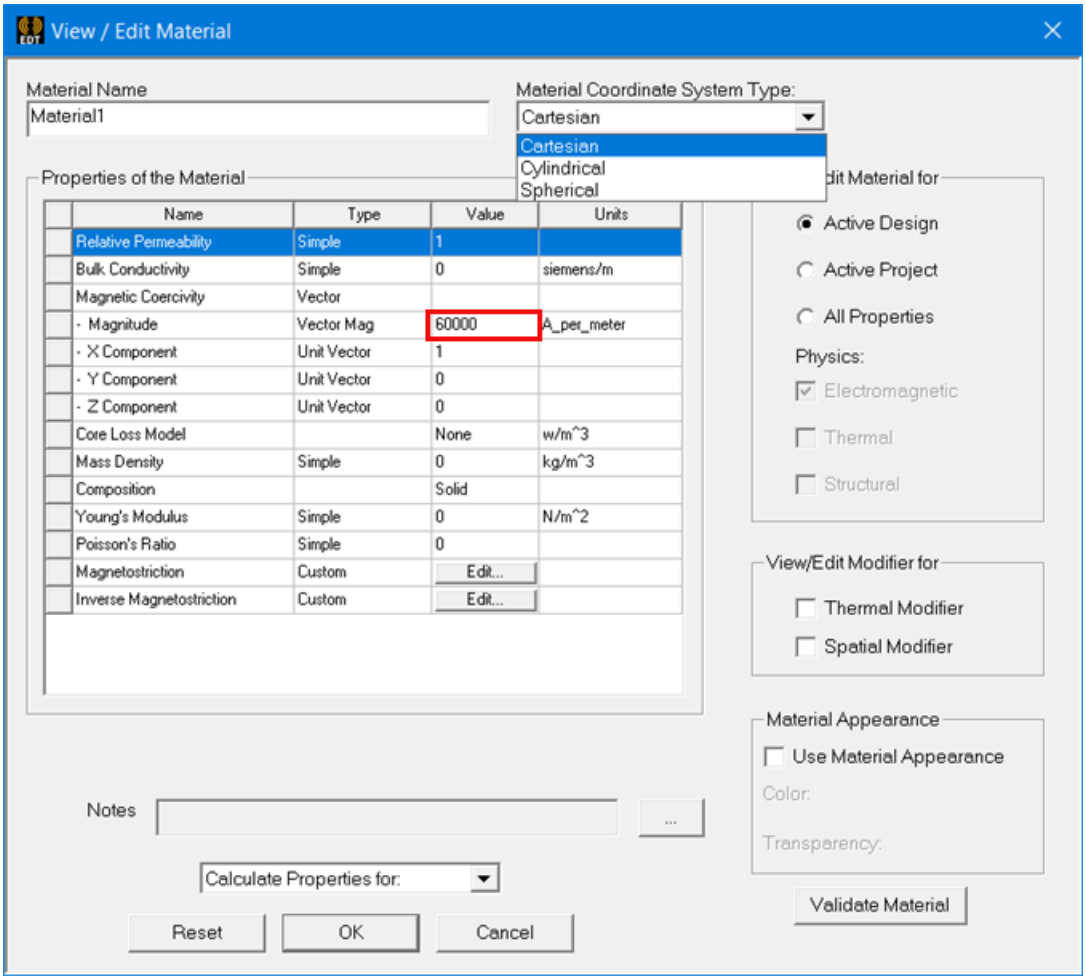
Material Coordinate Systems

Maxwell supports the following three types of coordinate systems, which can be used to define some vector or anisotropic material properties:

- Cartesian (defined by the X, Y, and Z axes)
- Cylindrical (defined by the R, Phi, and Z axes)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Spherical (defined by the Rho, Theta, and Phi axes)

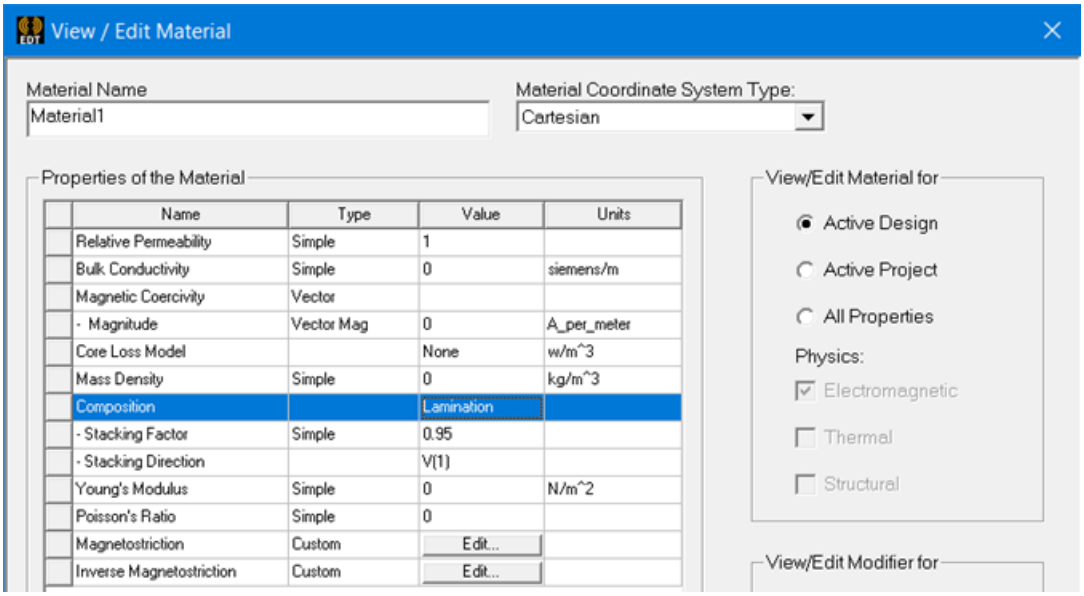


To get the fields for the vector magnetic coercivity vector, a non-zero value must first be specified for the magnitude of the vector. The magnitude can also be functional. The magnitude of the magnetic coercivity is expected to be < 0. If you use a positive value, the direction used will be the opposite of the unit vector specified.

Lamination Modeling

Lamination modeling allows you to specify a stacking factor and stacking direction, which represents the direction perpendicular to the plane of the lamination. Cartesian, cylindrical, or spherical coordinate systems can be used to specify the stacking direction.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This lamination model is an alternative way of specifying anisotropic behavior when using laminations (which is a special case that is frequently done). For the frequently encountered case of isotropic laminations (where the global model is anisotropic due to the existence of laminations but the laminations themselves are isotropic), the above picture shows a possible setup. Choose a nonlinear behavior for the material of the lamination with a user specified B-H curve, while the global anisotropy is modeled by specifying a stacking factor and the stacking direction, which is perpendicular to the plane of laminations. In this way, Maxwell can consider a global anisotropy with two orientations — one in the plane of the lamination, and the other in the corresponding orthogonal direction.

Note:

If a stacking factor less than 1 is applied to a magnetic core, the flux density plotted will represent the AVERAGE flux density in both the air and the steel (and not the flux density in the steel laminations). The average flux density is reduced as the stacking factor is reduced.

Note:

Lamination modeling is supported for magnetostatic, transient, and AC Magnetic solutions, where the stacking value is between 0 and 1. Make sure to use appropriate materials with suitable lamination definitions for the solver. Other solvers will ignore the lamination definition in the material.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

When a stacking factor (SF) is used, users need to modify the electrical steel core loss coefficients based on the stacking factor as shown below. As the stacking factor decreases < 1, the core loss coefficients increase.

Kh_scaled = Kh / SF

Kc_scaled = Kc / SF

Ke_scaled = Ke / sqrt(SF)

Stacking Direction

When defining a lamination model, the stacking direction must be specified. The direction depend upon the Material Coordinate System Type chosen as follows:

	Cartesian Coordinates	**Cylindrical Coordinates**	**Spherical Coordinates**
V(1)	X	r	r
V(2)	Y	θ	θ
V(3)	Z	Z	ϕ

Litz Wire Modeling

Note:

Litz wire modeling is currently supported for windings only (not for Conductor current sources) in Maxwell 2D/3D Transient, and 2D/3D AC Magnetic designs.

Litz wire is a multi-strand wire or cable whose primary benefit is in reducing AC losses in high frequency windings. It consists of many thin, individually insulated, wire strands, which are twisted or woven together in carefully prescribed patterns. The multiple strand construction of Litz wire allows current to divide uniformly between strands, thus reducing skin effect and proximity effect losses.

Some basic assumptions for applying this modeling approach are

- Each Litz wire strand is insulated individually.
- Each strand is continuously and perfectly transposed so that every strand occupies every location in the cross-sectional area of the bundle over the lay length.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The largest dimension of strands (for example, diameter or thickness) is smaller than the skin depth δ for the strand's material.

$$\delta = 1 \Big/ \sqrt{\pi f \mu \sigma}$$

- If the area of the coil with Litz wire modeling is smaller than *Strand number * Litz wire area*, which means the fill factor will be greater than 1, the fill factor will be truncated to 1, and the coil area is assumed to be full of Litz wire. If the filling factor is truncated, the resistance of Litz wire is still calculated based on Litz wire model, but the coil dimension will affect the resistance.

  Note that the fill factor (k_f) is defined as

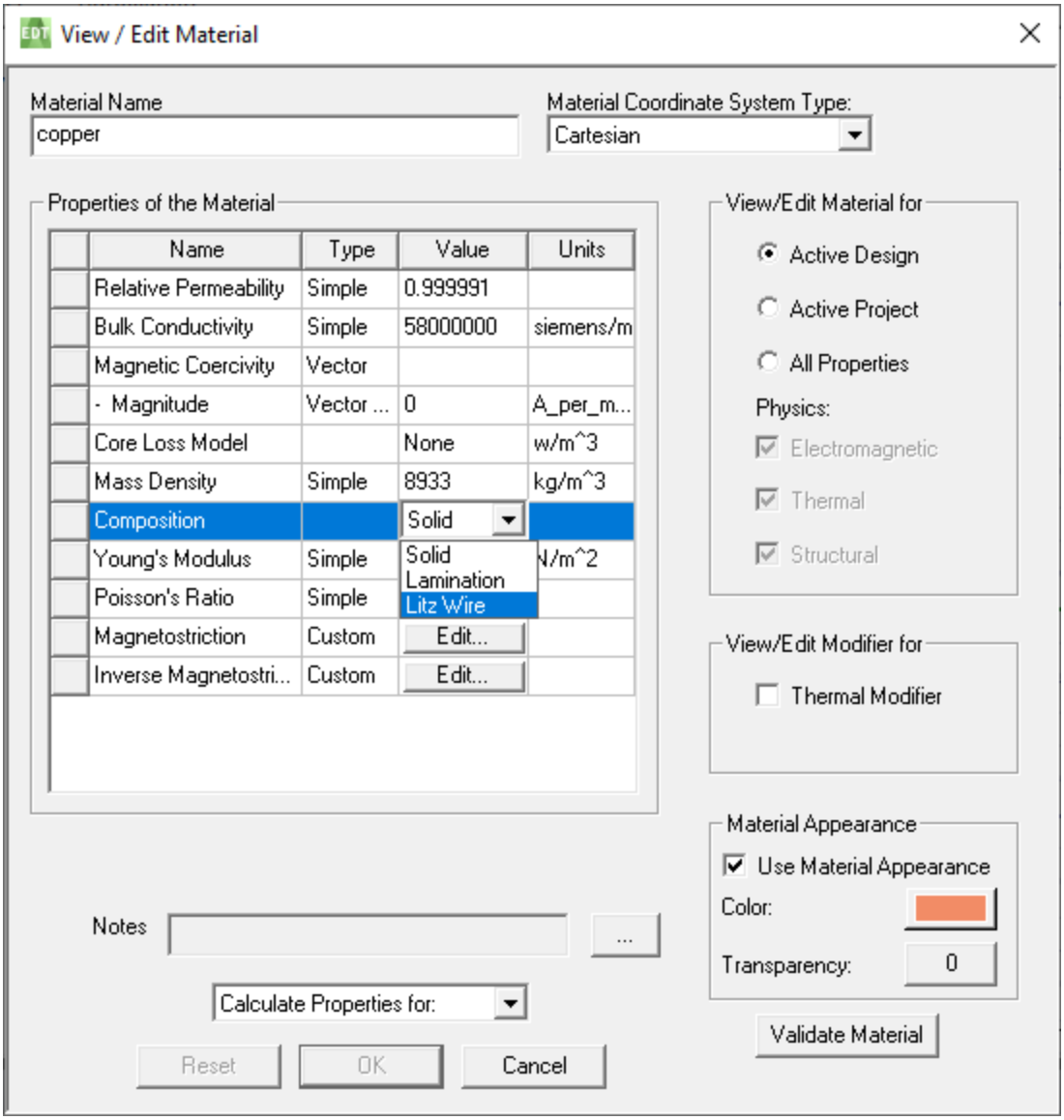
$$k_f = \frac{n_T n_s A_{wire}}{A_{term}}$$

  where n_T is the number of turns defined in the related coil terminal, n_s is the number of strands defined as a Litz-wire material property, A_{term} is the area of the related coil terminal, and A_{wire} is the area of a wire, which is given below:

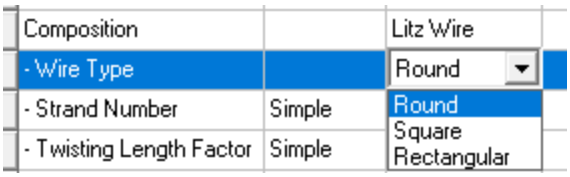
$$A_{wire} = \begin{cases} \pi d^2/4 & for\ a\ round\ wire \\ d^2 & for\ a\ square\ wire \\ d_w d_t & for\ a\ rectangular\ wire \end{cases}$$

Litz wire modeling allows you to specify the wire type, number of strands, and various other parameters that define the shape of the Litz wire bundle.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When **Litz Wire** is selected, **Wire Type** is displayed under **Composition**. The drop-down menu for Wire Type includes **Round**, **Square**, and **Rectangular**; these are the three possible shapes for a Litz wire.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> If **Litz Wire** is selected and any wire dimension is 0, the **Composition** material property will be treated as **Solid**, and a warning message to that effect will appear in the Message Manager.

If **Round** (the default) is selected, **Strand Number**, **Twisting Length Factor**, and **Wire Diameter** are displayed. Default values are as shown below. **Strand Number** is the integer value of the number of strands in the wire bundle. **Twisting Length Factor** is the ratio of the twisted-strand length to the bundle length. **Wire Diameter** is a real number that defines the diameter of a single wire in the wire bundle. You can also select the desired unit of measure.

Composition		Litz Wire	
- Wire Type		Round	
- Strand Number	Simple	1	
- Twisting Length Factor	Simple	1	
- Wire Diameter	Simple	0	mm

If **Square** is selected for **Wire Type**, **Strand Number**, **Twisting Length Factor**, and **Wire Width** are displayed. Default values are as shown below. **Strand Number** is the integer value of the number of strands in the wire bundle. **Twisting Length Factor** is the ratio of the twisted-strand length to the bundle length. **Wire Width** is a real number that defines the width of a single wire in the wire bundle. You can also select the desired unit of measure.

Composition		Litz Wire	
- Wire Type		Square	
- Strand Number	Simple	1	
- Twisting Length Factor	Simple	1	
- Wire Width	Simple	0	mm

If **Rectangular** is selected for **Wire Type**, the properties **Strand Number**, **Twisting Length Factor**, **Wire Width**, **Wire Thickness**, **Thickness Direction**, and **Width Direction** will be displayed. **Twisting Length Factor** is the ratio of the twisted-strand length to the bundle length. **Wire Width** is a real number that defines the width of a single wire in the wire bundle. Default values are as shown below. For thickness direction and width direction The drop-down menus include three options: V(1), V(2) and V(3), representing x, y, and z directions, respectively, for the **Cartesian** Material Coordinate System Type; and r, θ, z directions, respectively, for the **Cylindrical** Material Coordinate System Type.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Composition		Litz Wire	
	- Wire Type		Rectangular	
	- Strand Number	Simple	1	
	- Twisting Length Factor	Simple	1	
	- Wire Width	Simple	0	mm
	- Wire Thickness	Simple	0	mm
	- Thickness Direction		V(1)	
	- Width Direction		V(2)	

For inner/outer core types, the armature ac or dc windings inside core slots for large capacity machines rotating around z-axis usually have coils with wire thickness in r direction, and width in θ direction or, rarely, have coils with wire thickness in θ direction, and width in r direction.

For axial-flux core types, the armature ac or dc windings inside core slots of large capacity machines rotating in z-axis usually have coils with wire thickness in z direction, and width in θ direction, or rarely have coils with wire thickness in θ direction, and width in z direction.

The total loss from the Litz wire model (including DC loss and AC loss) is applied for thermal coupling. The effect of temperature on Litz wire loss calculation is taken into account in Maxwell by setting a thermal modifier for the bulk conductivity material property.

Core Loss Model for a Maxwell Material

The parameters and units for core loss model are listed in the table that follows.

> **Note:** All core loss property parameters for **Electrical Steel** and **Power Ferrite** materials can be temperature-dependent, as defined using a thermal modifier.

Name	Value
None	No core loss is to be calculated for this material.
Electrical Steel	The following parameters appear: • **Kh** – Type can be either **Simple** or **Anisotropic** • **Kc** – Type can be either **Simple** or **Anisotropic** • **Ke** – Type can be either **Simple** or **Anisotropic** • **Y** – Type can be either **Simple** or **Anisotropic**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Note: In 2023 R2 and earlier versions, hysteresis loss is computed from $Ph = Kh * f * Bm^2$ In 2024 R1 and later, hysteresis loss model is improved to be $Ph = Kh * f * Bm^Y$ • **Kdc** (not used for AC Magnetic solution type) • **Equiv. Cut Depth** (Refer to Cut Edge Effects on Core Loss for Lamination Steels for details on the use of this parameter.) For **Kh**, **Kc**, **Ke**, and **Y**, if the type is anisotropic, the following parameters appear: • T(1,1) • T(2,2) • T(3,3) These characteristics are defined by their anisotropy tensors. Each diagonal represents a tensor of your model along an axis. You can enter a simple value for each of these parameters. These tensors are relative to the coordinate system specified as the object's Orientation property. By specifying different orientations, several objects can share the same anisotropic material but be oriented differently. **Note:** Selecting **Electrical Steel** also enables the Calculate Properties for Core Loss Coefficient drop-down menu at the bottom of the dialog box. Selecting either **Core Loss at One Frequency** or **Core Loss versus Frequency** allows you to edit the **BP Curve**.
Power Ferrite	The following parameters appear: • **Cm** – Type can be either **Simple** or **Anisotropic** • **X** – Type can be either **Simple** or **Anisotropic** • **Y** – Type can be either **Simple** or **Anisotropic** • **Kdc** (not used for AC Magnetic solution type) • **Equiv. Cut Depth** (Refer to Cut Edge Effects on Core Loss for Lamination Steels for details on the use of this parameter.) For **Cm**, **X**, and **Y**, if the type is anisotropic, The following parameters appear: • T(1,1) • T(2,2) • T(3,3)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	These characteristics are defined by their anisotropy tensors. Each diagonal represents a tensor of your model along an axis. You can enter a simple value for each of these parameters. These tensors are relative to the coordinate system specified as the object's Orientation property. By specifying different orientations, several objects can share the same anisotropic material but be oriented differently. **Note:** • Selecting **Power Ferrite** also enables the Calculate Properties for Core Loss Coefficient drop-down menu at the bottom of the dialog box. Selecting either **Core Loss at One Frequency** or **Core Loss versus Frequency** allows you to edit the **BP Curve**. • In Transient Solver, X must be less than Y.
B-P Curve	The following parameter appears: • **Core Loss**. Click the **B-P Curve** button to define temperature-dependent core loss curves. (Refer to Calculating Properties for Temperature-dependent Core Loss Curves in Maxwell for details on the use of this parameter.) If view/edit parameters for Active Project is selected, the following additional parameters appear: • **Kdc** Coefficient Considering DC Bias Effects • **Equiv. Cut Depth** (Refer to Cut Edge Effects on Core Loss for Lamination Steels for details on the use of this parameter.)
Hysteresis Model	The following parameters appear: • **Intrinsic Coercivity Hci** • **Remanence Br** • **Kc** • **Equiv. Cut Depth** (Refer to Cut Edge Effects on Core Loss for Lamination Steels for details on the use of this parameter.) **Note:** Selecting **Hysteresis Model** also enables the Calculate Properties for Hysteresis Loop drop-down menu item at the bottom of the dialog box. Selecting **Hysteresis Loop** opens the **Properties for Hysteresis Loop** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Additional Considerations

DC-Biased Effects

The core loss computation is based on the traditional three core loss coefficients **Kh**, **Kc**, and **Ke**, plus the optional **Kdc**. In order to properly consider the impact of DC-bias on core loss, an additional factor derived from Kdc is applied to scale the hysteresis loss from the Bertotti's formula. The additional factor is expressed as Kph = 1 + Kdc * Bdc^2. If the user leaves the default value of Kdc as zero, this indicates that the impact of DC-bias will not be considered because of the additional factor Kph = 1. If a user wants to consider the DC-bias effect, the user needs to do an experiment to measure core loss at various DC-bias flux densities Bdc, and extract Kdc based on the Kph expression as given above using a linear regression. If the measured data are not available, you can input 0.65 for Kdc.

> **Note:** The use of Kdc when computing DC-biased effects has no effect on the hysteresis model-based core loss computation approach since in such a case, the hysteresis model has inherently considered the impact of DC-biased effects on hysteresis loss.

Hysteresis Model-Based Core Loss Computation Approach

Using this approach, the computation of the hysteresis loss component is based on the input of a hysteresis loop, and the computation of the eddy current loss component based on the classic eddy current loss coefficient Kc. This means that the impact of classic excess loss must be included in the hysteresis loss (sometime called: dynamic hysteresis loss). There are two ways to enable computation of core loss for the hysteresis model-based approach:

- Define the descending limiting branch of the hysteresis loop in the nonlinear BH curve input panel together with the input of **Kc**, which is defined by choosing **Electrical Steel** as the **Core Loss Model** on the **Properties of the Material** panel. Note that any values entered for **Kh** and **Ke** will be ignored.

  > **Note:** This approach requires use of a material that has a vector magnitude direction set to (0,0,0) to enable the hysteresis loss calculation; and must also have its **Core Loss Setting** checked in the **Set Core Loss** dialog box to include the eddy current component of core loss.

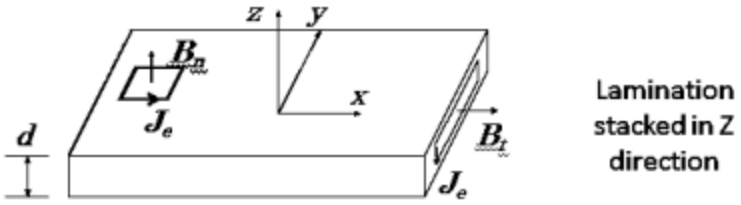
- Define the hysteresis loop by the input of a normal nonlinear BH curve together with the selection of **Hysteresis Model** as the **Core Loss Model** and the input of values for **Intrinsic Coercivity Hci**, **Remanence Br** (optional) and **Kc**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** The second approach is able to more accurately model minor loop loss behavior.

Additional Core Loss Due to Flux Normal to Laminations

When the lamination model is used in the Maxwell 3D transient solver, an additional eddy current component of core loss will be automatically calculated due to the normal component of flux (B_n) on the lamination stack and will be added to the total core loss if conductivity > 0 has been specified in the material manager. The basic idea of the algorithm for considering this additional eddy current component of core loss is to force induced eddy current only occurring in the plane of lamination by introducing anisotropic conductivity.



To compute this additional core loss component:

- The material conductivity, which is taken from the project material library (not from the core loss input panel), must be specified as non-zero.
- The object must be defined as a lamination.
- The eddy effect must always be turned off for laminated objects.
- Core Loss Setting should be activated on the Set Core Loss panel, General tab under Excitations.
- A fine mesh in the object is required to accurately calculate the additional eddy current loss due to this effect.
- For adjacent lamination stacks that are touching, insulating boundaries should be applied to prevent eddy currents from circulating between the stacks.

Additional Core Loss Due to Manufacturing Processes

Lamination steel is cut to the final shape by manufacturing processes such as punching or laser cutting. These processes deteriorate the magnetic properties of the material due to the plastic deformation and residual stress near the cut edge that directly cause core loss increase in the deformation region. These losses can be taken into account for the accurate prediction of the core losses using the **Equiv. Cut Depth** property value.

To compute this additional core loss component:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The Edge Cut Based mesh operation must be specified.
- The **Composition** property for the assigned material must be set to **Lamination.**
- The **Equiv. Cut Depth** material property must be set to a value greater than zero (default is 1 mm).

Cut Edge Effects on Core Loss for Lamination Steels

Lamination steel is cut to the final shape by manufacturing process such as punching and laser cutting, which deteriorates the magnetic properties of the material. This is because the plastic deformation and residual stress near the cut edge cause core loss increase in the deformation region. These losses should be taken into account for the accurate prediction of core losses.

Cut-Edge Loss Model

Assume core loss per unit volume without considering cut-edge effects is p_0.

If the cut-edge deformation depth is d_c, the cut-edge area is S_c, and the core loss increasing factor is k_0, the core loss considering cut-edge effects will be

$$P_c = (p_0 + k_0 p_0)\, d_c\, S_c$$

Thus, the loss increase due to cut-edge effects is

$$\Delta P_c = p_0 (k_0 d_c) S_c$$

Because parameters d_c and k_0 cannot normally be derived individually, an equivalent cut-edge depth is introduced:

$$d_{eq} = k_0 d_c$$

such that d_{eq} can be identified by

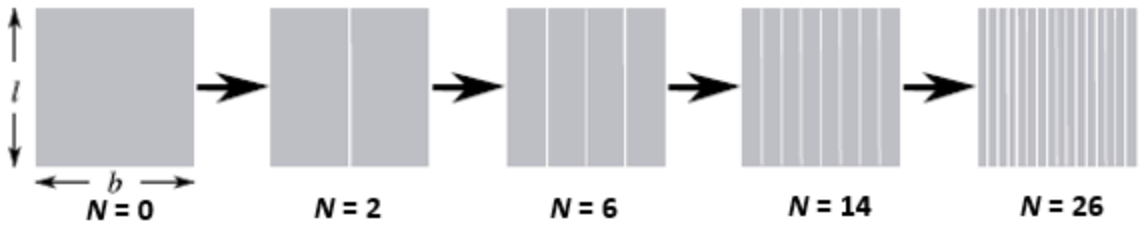
$$d_{eq} = \Delta P_c / (p_0 S_c)$$

Parameter d_{eq} derivation

Based on the measurement of core loss of lamination sheets with variable width to consider the cut-edge effects, manufacturers can use single sheet testing method to measure the impact of cut-edges on core loss. The steps for deriving the parameter d_{eq} are as follows:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Cut lamination to N edges of different widths as shown below.



- Measure the loss P_{ij} with different width (j) under different magnetic polarization (i).
- Derive the loss P_{ij}' using the cut-edge loss model to obtain one equation for every combination of i and j with parameter d_{eq} to be identified:

$$P_{ij}' = P_{i0} + \Delta P_{ij} = P_{i0}(1 + d_{eq} N_j / b)$$

- Find the error ε between derived and measured:

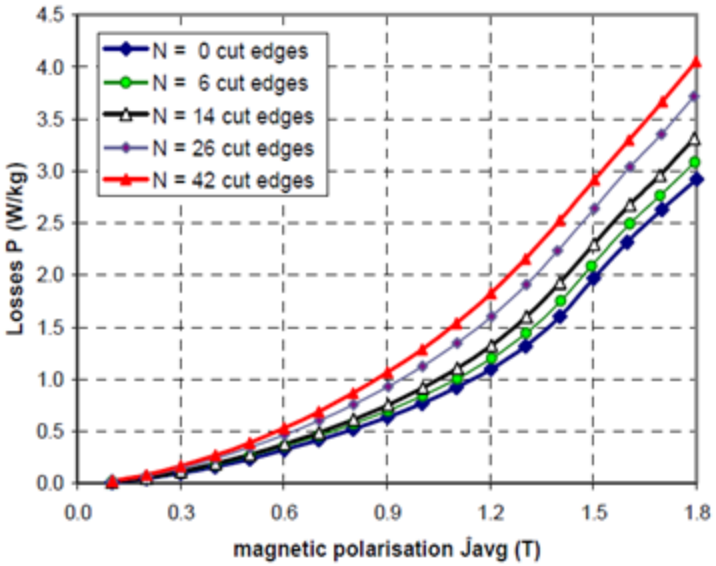
$$\varepsilon = \sum_{i=1}^{n} \sum_{j=1}^{m} (P_{ij}' - P_{ij})^2$$

- Derive parameter d_{eq} by minimizing the error:

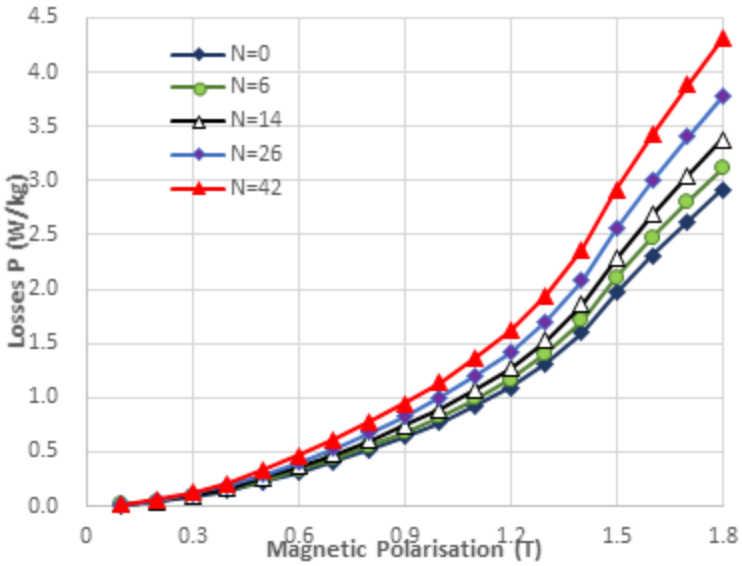
$$d_{eq} = b \frac{\sum_i \sum_j (P_{ij} - P_{i0}) P_{i0} N_j}{\sum_i \sum_j (P_{i0} N_j)^2}$$

For example, the following shows the *measured* core loss varying with magnetic polarization for several values of the number of cut edges;

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



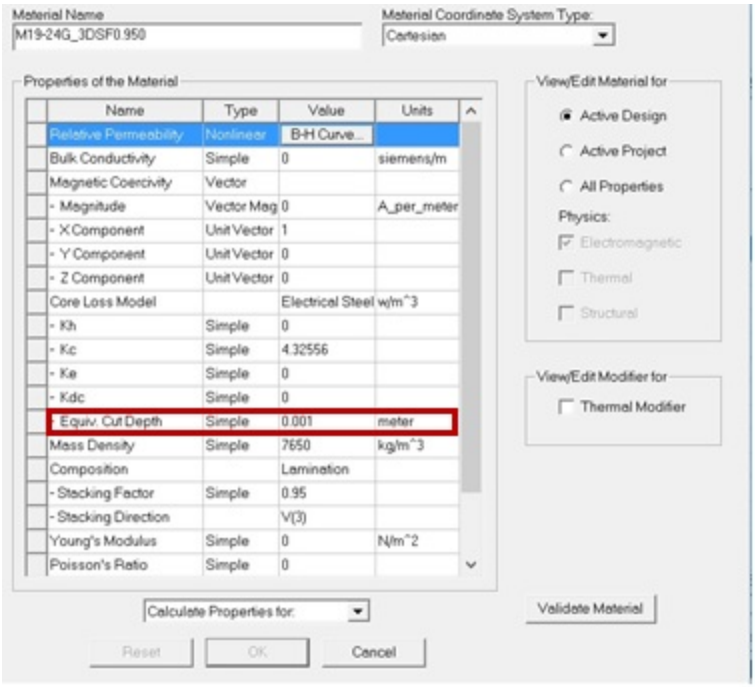
and the following shows the *simulated* core loss varying with magnetic polarization for several values of the number of cut edges



Maxwell Implementation of Cut Edge Effects

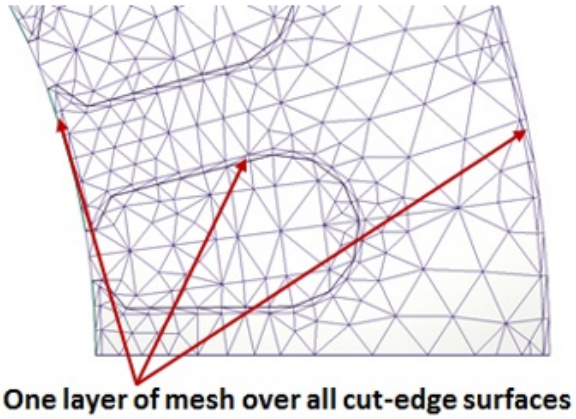
Maxwell uses the parameter d_{eq} described above as an additional input material property, **Equiv. Cut Depth**.

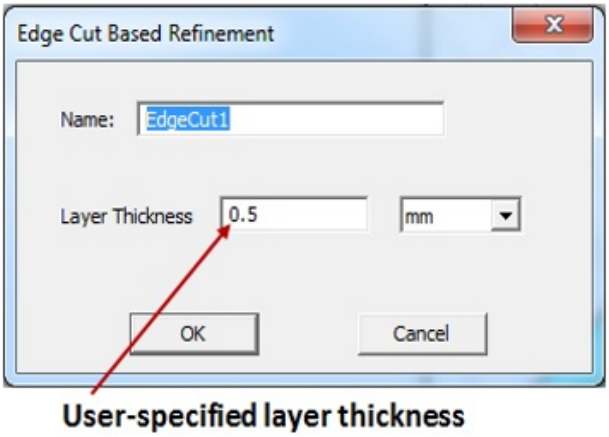
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To implement cut edge effects, the Maxwell solver:

1. Constructs one layer of mesh over all cut-edge surfaces with user-specified layer thickness d_{LT}





2. The solver then computes core loss considering cut-edge effects P_c on all elements inside

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the layer:

$$\begin{aligned} P_c &= P_0 + \Delta P_c \\ &= p_0 \, d_{LT} \, S_c + p_0 \, d_{eq} \, S_c \\ &= \left(1 + \frac{d_{eq}}{d_{LT}}\right) p_0 \, d_{LT} \, S_c \\ &= k_c \, P_0 \end{aligned}$$

Additional Considerations

It is not necessary for the layer thickness d_{LT} to be equal to the equivalent cut-edge depth d_{eq} since the above equation is able to ensure the derived core loss increase matches the measurements.

To take into account the cut-edge effects in core loss computation, the following three conditions must be fulfilled at the same time:

- The Edge Cut Based mesh operation must be specified.
- The **Composition** property for the assigned material must be set to **Lamination**.
- The **Equiv. Cut Depth** material property must be set to a value greater than zero (default is 1 mm).

Calculating Properties for Core Loss (BP Curve)

To be able to extract parameters from the loss characteristics (B-P Curve), you first set the **Core Loss Model** of the material to **Electrical Steel** or **Power Ferrite** as a material property in the **View / EditMaterial** window.

> **Note:**
>
> If the Core Loss Model is set to **Hysteresis Model**, refer to Calculating Properties Using a Hysteresis Loop in Maxwell.
>
> If the Core Loss Model is set to **B-P Curve,** refer to Calculating Properties for Temperature-Dependent Core Loss Curves in Maxwell.

To calculate core loss properties for an electrical steel material:

1. Click **Tools** > **Edit Libraries** > **Materials**.

   Alternatively, in the Project Manager tree, you can right-click **Materials**, under **Definitions**, and select **Edit Libraries**.

   The **Edit Libraries** dialog box appears.

2. Click **Add Material**.
   The **View / Edit Material** dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In the **Core Loss Model** row, select **Electrical Steel** or **Power Ferrite** from the **Value** drop-down menu.

   Additional parameters appear in the following table **Core Loss Model** (**Kh**, **Kc**, **Ke**, **Y**, **Kdc**, and **Equiv. Cut Depth** for electrical steel, and **Cm**, **X**, **Y**, **Kdc**, and **Equiv. Cut Depth** for power ferrite). For **Electrical Steel**, the **Core Loss Coefficient** selection in the **Calculate Properties for** drop-down menu at the bottom of the dialog box is also enabled. This allows the electrical steel coefficients **Kh**, **Kc**, **Ke**, **Y**, and **Kdc**, or the power ferrite coefficients **Cm**, **X**, **Y**, and **Kdc** to be derived from a manufacturer-provided core loss curve.

   > **Note:**
   >
   > Both electrical steel and power ferrite core loss models are unsuitable when there is a wide frequency range. If the fitting error is too large, consider duplicating the curve data and reducing the number of curves to just the relevant frequencies. As an example, for the loss curves of a material with frequencies ranged from 50Hz to 1kHz, it is better to have two parameter sets, one for 50Hz, 60Hz, and 100Hz curves, and the other for 400Hz and 1kHz curves.
   >
   > The fitting error can be used as a guide on whether the electrical steel or power ferrite model is more suitable.

   > **Note:**
   >
   > The **Kdc** core loss quantity is not used for the AC Magnetic solution type.

4. If you selected **Electrical Steel** and have one measured BP curve at a fixed frequency, do the following:
   a. Select **Core Loss at One Frequency** from the **Calculate Properties for** drop-down menu at the bottom of the dialog box.
      The **BP Curve** dialog box appears.
   b. Do one of the following to specify a BP curve:
      - Import the curve from a saved file.
      - Enter the coordinates manually.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> The accuracy in inputting the data for BP Curve for the electrical steel material has a significant effect on the correctness of analyses of electromagnetic devices. You should input the data for BP Curve according to accurate data provided by material manufacturers. Typically, core material suppliers provide the average loss over a cycle for a peak B, of sinusoidal nature. Therefore, for BP curve input in Maxwell, B(Tesla) should be peak, and P should be average.

> **Note:**
>
> Measured BP curves that are a linear function of $(Bm)^Y$ will provide a more accurate curve fit.

c. Select the units in which the B-P curve is measured from the **Core Loss Unit** drop-down menu.

d. Enter values and select units for the following:

- **Mass Density**
- **Frequency**
- **Thickness**
- **Conductivity**

As the input data changes, the following parameters are dynamically updated with both the specified unit and the standard unit (w/m^3):

- **Kh** - Hysteresis Coefficient
- **Kc** - Classical Eddy Coefficient
- **Ke** - Excess Coefficient
- **Y** – Power of Bm for hysteresis core loss: $Ph = Kh * f * Bm^Y$

e. Click **OK**. The **View / Edit Material** dialog box reappears. The property values for **Kh**, **Kc**, **Ke**, and **Y** are updated as new default values.

5. If you selected **Power Ferrite** or **Electrical Steel**, and have measured frequency-dependent BP curves, do the following:

a. Select **Core Loss versus Frequency** from the **Calculate Properties for** drop-down menu at the bottom of the dialog box. The Core Loss versus Frequency dialog box appears.

b. Specify a frequency value in **Edit** box and click the **Add** button to add a frequency in the Frequency field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Do one of the following to specify a BP curve for each specified frequency via **Edit Dataset** button in the **Dataset** field:
   - Import curve from a saved file
   - Enter coordinates manually

d. Select the units in which the BP curve is measured from the **Core Loss Unit** drop-down menu, and specify **Mass Density** when it is enabled.

   As the input data changes, the following parameters are dynamically updated with both the specified unit and the standard unit (w/m^3):

   - **Cm** – Coefficient of power-ferrite core loss
   - **X** – Power of frequency for power-ferrite core loss
   - **Y** – Power of Bm for power-ferrite core loss: $P = Cm * f^X * Bm^Y$

e. Click **OK**. The View / Edit Material dialog box reappears. The property values for **Cm**, **X**, and **Y** are updated as new default values.

> **Note:**
>
> The measured frequency-dependent BP curves can be used to extract parameters of the core loss model for either the Electrical Steel or the Power Ferrite material. Therefore, users can change the Loss Model inside the dialog box to see which model will have better regression curves. The regression error is displayed, and the displayed parameters change as the Loss Model changes. The smaller the regression error is, the better the regression curves match the measured curves.

f. Click **OK** to close the **View / Edit Material** dialog box.

g. Click **OK** to close the **Edit Libraries** dialog box.

Core Loss Coefficients for Electrical Steel

Under sinusoidal flux conditions, core loss is computed in the frequency domain as follows:

$$P_v = P_h + P_c + P_e = K_h f(B_m)^Y + K_c(fB_m)^2 + K_e(fB_m)^{1.5}$$

For a material with anisotropic core losses, the coefficients Kh, Kc, Ke, and Y have different values in different principal directions, that is, (Kh_x, Kh_y, Kh_z), (Kc_x, Kc_y, Kc_z), (Ke_x, Ke_y, Ke_z), and (Y_x, Y_y, Y_z) in (x, y, z) directions, respectively.

When a DC component exists in the flux density, the core loss is modified to the following:

$$P_v = C_{dc}K_h f(B_m)^Y + K_c(fB_m)^2 + K_e(fB_m)^{1.5}$$

where

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **B_m** is the amplitude of the AC flux component,
- **f** is the frequency,
- **K_h** is the hysteresis core loss coefficient,
- **K_c** is the eddy-current core loss coefficient, and
- **K_e** is the excess core loss coefficient.
- **Y** is the power of Bm for hysteresis core loss.

C_{dc} is computed from the following equation:

$$C_{dc} = 1 + K_{dc}{B_{dc}}^2$$

where

- **B_{dc}** is the DC flux component, and
- **K_{dc}** is the coefficient considering the DC flux bias effects.

Core Loss Coefficient Extraction from Single-Frequency Loss Curve

The principles of the computation algorithm for **K_h**, **K_c**, **K_e**, and **Y** are summarized as below.

> **Note:** Because the manufacturer-provided loss curve is obtained under sinusoidal flux conditions at a given frequency, these coefficients can be derived in the frequency domain.

The iron-core loss without DC flux bias is expressed as the following:

$$P_v = P_h + P_c + P_e = K_h f B_m^Y + K_c (f B_m)^2 + K_e (f B_m)^{1.5}$$

where f is the frequency, and B_m is the amplitude of the applied sinusoidal flux density waveform. The classical eddy-current loss coefficient K_c can be calculated analytically from

$$K_c = (\pi d)^2 \sigma / 6$$

where σ is the conductivity, and d is the thickness of one lamination sheet.

The remaining coefficients K_h and K_e, as well as Y, can be derived by minimizing the error between the measured and modeled core loss curves at different B_m expressed as

$$err = \sum_{i=0}^{n-1} \left(P_{mi} - P_{vi}\right)^2 = min$$

where P_{mi} and P_{vi} are the measured and modeled core losses at a fixed frequency f and varied flux density amplitudes B_{mi}, and n is the number of points of the measured core loss curve. K_c is known, so the modeled core loss P_{vi} depends on unknown parameters K_h, K_e, and Y. K_h and K_e

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

can be identified by linear regression, and *Y* by a numerical optimization algorithm. Assuming an initial value of *Y*, the error can be derived from

$$err\,(Y) = \sum_{i=0}^{n-1} \left(P_{mi} - \left(K_h f B_{mi}^{Y} + K_c\left(fB_{mi}\right)^2 + K_e(fB_{mi})^{1.5}\right)\right)^2 = min$$

After K_h and K_e have been determined by linear regression, the error is a function of *Y*. The optimal *Y* can be finally derived by applying a numerical optimization algorithm to minimize the error.

Core Loss Coefficient Extraction from Multi-Frequency Loss Curves

The principles of the computation algorithm for $\mathbf{K_h}$, $\mathbf{K_c}$, $\mathbf{K_e}$, and **Y** are summarized below.

> **Note:** Because the manufacturer-provided loss curve is obtained under sinusoidal flux conditions at a given frequency, these coefficients can be derived in the frequency domain.

The iron-core loss without DC flux bias is expressed as the following:

$$P_v = P_h + P_c + P_e = K_h f B_m^{Y} + K_c(fB_m)^2 + K_e(fB_m)^{1.5}$$

where *f* is the frequency and B_m is the amplitude of the applied sinusoidal flux density waveform. The coefficients K_h, K_c, and K_e, as well as *Y*, can be derived by minimizing the error between the measured and modeled core loss curves at different frequencies *f* and varied flux density amplitudes B_m expressed as

$$err = \sum_{j=0}^{m-1} \sum_{i=0}^{n-1} \left(P_{mij} - P_{vij}\right)^2 = min$$

where P_{mij} and P_{vij} are the measured and modeled core losses at different frequencies f_j and varied flux density amplitudes B_{mi}, *n* is the number of points of the measured core loss curve, and *m* is the number of curves of different frequencies.

The modeled core loss P_{vij} depends on unknown parameters K_h, K_c, K_e, and *Y*. Here K_h, K_c, and K_e can be identified by linear regression, and Y by a numerical optimization algorithm. Assuming an initial value of *Y*, the error can be expressed as

$$err\left(Y\right) = \sum_{j=0}^{m-1} \sum_{i=0}^{n-1} \left(P_{mi} - \left(K_h f_j {B_{mi}}^{Y} + K_c\left(f_j B_{mi}\right)^2 + K_e\left(f_j B_{mi}\right)^{1.5}\right)\right)^2 = min$$

After K_h, K_c, and K_e have been determined by linear regression, the error is a function of *Y*. The optimal *Y* can be finally derived by applying a numerical optimization algorithm to minimize the error.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Core Loss Parameter Extraction for Power Ferrite Materials

The principles of the computation algorithm for $\mathbf{C_m}$, **X**, and **Y** are summarized below.

> **Note:** Because the manufacturer-provided loss curve is obtained under sinusoidal flux conditions at a given frequency, these coefficients can be derived in the frequency domain.

The power-ferrite iron-core loss without DC flux bias is expressed as the following:

$$P_v = C_m f^X B_m^Y$$

or

$$log\,(P_v) = C + X log\,(f) + Y log\,(B_m)$$

where f is the frequency, and B_m is the amplitude of the applied sinusoidal flux density waveform, and $C = log(C_m)$.

The parameters C, X, and Y can be derived by minimizing the error between the measured and modeled core loss curves at different frequencies f and varied flux density amplitudes B_m expressed as

$$err = \sum_{j=0}^{m-1} \sum_{i=0}^{n-1} \left(\log(P_{mij}) - \log(P_{vij})\right)^2 = min$$

or

$$err = \sum_{j=0}^{m-1} \sum_{i=0}^{n-1} \left(\log(P_{mij}) - (C + X \log\,(f_j) + Y \log(B_{mi}))\right)^2 = min$$

where P_{mij} and P_{vij} are the measured and modeled core losses at different frequencies f_j and varied flux density amplitudes B_{mi}, n is the number of points of the measured core loss curve, and m is the number of curves of different frequencies.

After C, X, and Y have been determined by linear regression, parameter C_m can be obtained from $C_m = e^C$.

Importing a B-P Curve From a File

For the loss characteristics (B-P Curve), you first set the **Core Loss Model** of the material to **Electrical Steel** or **Power Ferrite** as a material property in the View/Edit Material window. When you do so, this enables the Calculate Properties selection for drop down at the bottom of the window. Select either **Core Loss at One Frequency** or **Core Loss versus Frequency** from the drop down menu to open the B-P Curve window.

For **Core Loss at One Frequency**:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the BP Curve dialog box, click **Import Dataset** to open the **Import Dataset** dialog box.
2. Find and select the tab separated points file containing the BP curve data points.
3. Click **Open**.
4. Optionally, click **Swap X-Y Data** to switch the B values and P values if they are in the wrong columns.

For **Core Loss versus Frequency**:

1. In the Core Loss versus Frequency dialog box, if needed, enter the desired **Frequency** and click **Add**.
2. Click **Edit Dataset** for the desired frequency to open the **Edit Dataset** dialog box.
3. Click **Import Dataset** to open **Import Dataset**dialog box.
4. Find and select the tab separated points file containing the BP curve data points.
5. Click **Open**.
6. Optionally, click **Swap X-Y Data** to switch the B values and P values if they are in the wrong columns.

Manually Specifying B-P Curve Coordinates

To manually specify the loss characteristics (B-P Curve):

1. In the **View/Edit Material** window, set the **Core Loss Model** of the material to **Electrical Steel** or **Power Ferrite**.

   When you do so, this enables the Calculate Properties selection for the drop-down menu at the bottom of the window.
2. Select either **Core Loss at One Frequency** or **Core Loss versus Frequency** from the drop-down menu to open the B-P Curve window.
3. Select **Core Loss Coefficient** from the **Calculate Properties for** drop-down menu to open the **BP Curve** dialog box.
4. In the **BP Curve** dialog box, enter **B** and **P** values in each row of the **Coordinates** table.

   > **Note:** The value of B must increase along the curve. Placing the cursor in the field in the **Coordinates** table enables the **Add Row** and **Delete Row** buttons.

   As you enter values, the graph is updated.
5. To add or edit rows, click the following buttons:
   - **Add Row Above**
   - **Add Row Below**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Append Rows** (In the dialog box that appears, specify the number of rows to append to the table, and click **OK**.)
- **Delete Rows**

Temperature-Dependent Core Loss Curves

To be able to extract parameters from the temperature-dependent core loss characteristic curves, you first set the **Core Loss Model** of the material to **B-P Curve** as a material property in the **View / Edit Material** window.

> **Note:** When a material has temperature-dependent core loss curves defined, neither the Thermal Modifier nor the Spatial Modifier are applicable to the material.

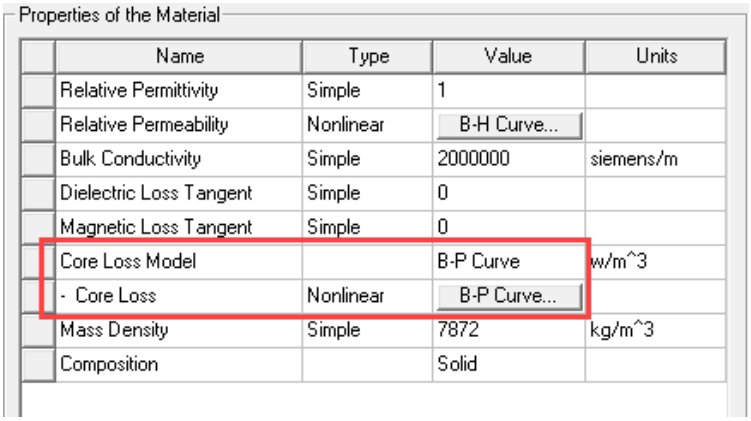
To calculate temperature-dependent core loss properties for a material:

1. Click **Tools** > **Edit Libraries** > **Materials**.

   Alternatively, in the Project Manager tree, right-click **Materials**, under **Definitions**, and select **Edit All Libraries**.

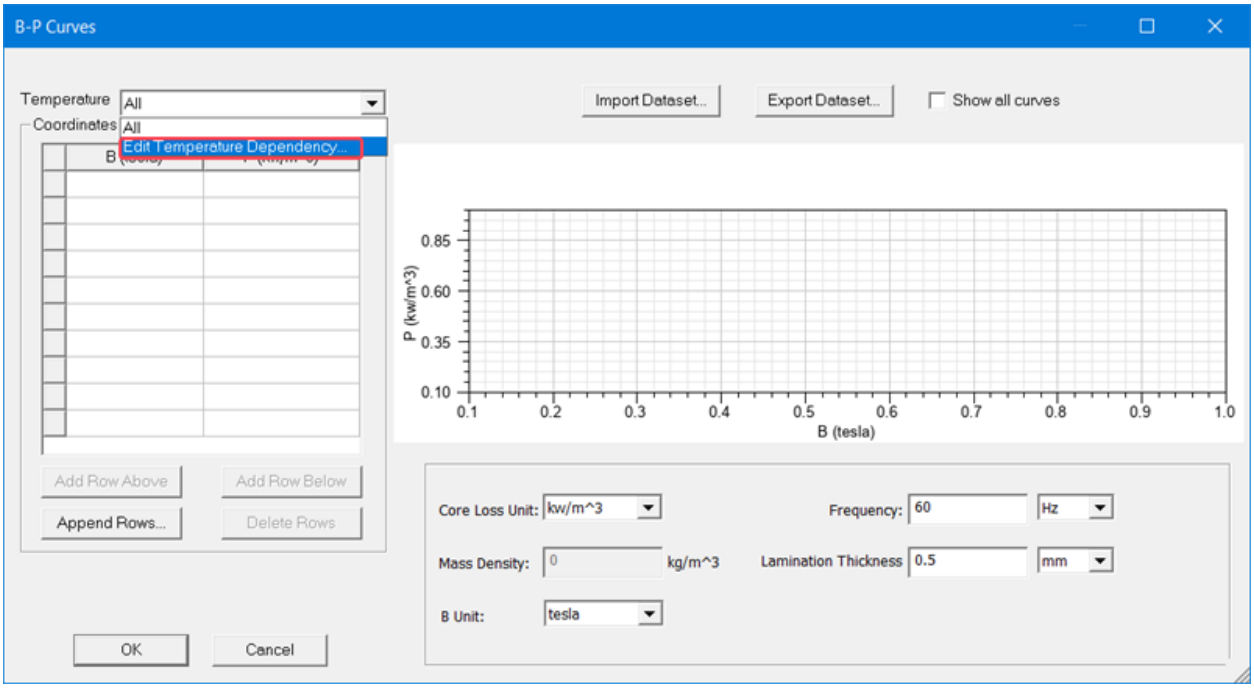
   The **Edit Libraries** window appears.

2. Click **Add Material**.
   The **View / Edit Material** window appears.
3. In the **Core Loss Model** row, select **B-P Curve** from the **Value** drop-down menu.

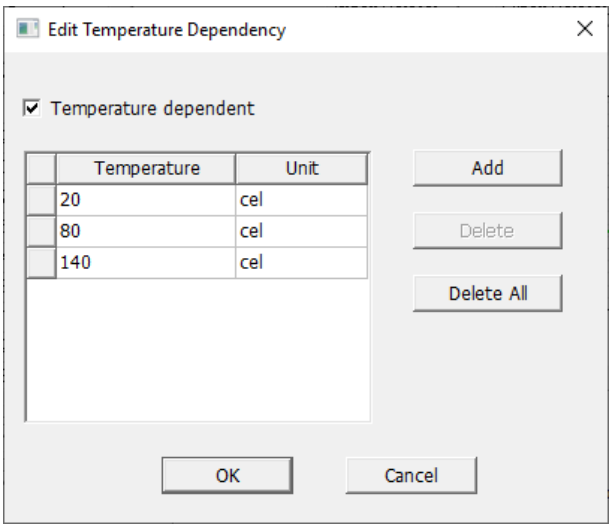


For **Core Loss**, click the **B-P Curve** button. The **B-P Curve** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. From the **Temperature** drop-down menu you can do either of the following:
   a. Select **All** to specify a single core loss curve.
   b. To create (or edit) multiple core loss curves, select **Edit Temperature Dependence**.



   - Click **Add** to add temperatures; then select the desired units. You can also select rows and **Delete** them, or **Delete All**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Check **Temperature dependent** to enable use of the temperature-dependent B-P curves by the solver.
- Click **OK** to close the dialog box and return to the **B-P Curves** window.

5. Do one of the following to specify a B-P curve:
   a. Click **Import Dataset** to open the **Import Dataset** dialog box. Find and select the tab separated file containing the B-P curve data points. (Data points must start at the origin.)
   b. Enter the coordinates manually. (Data points must start at the origin.)
6. Select the units in which the B-P curve is measured from the **Core Loss Unit** and **B Unit** drop-down menus.
7. Type values and select units for the following parameters:
   - **Mass Density**
   - **Frequency**
   - **Lamination Thickness**
8. Click **OK**.The **View / Edit Material** window reappears.
9. Click **OK** to close the **View / Edit Material** window.
10. Click **OK** to close the **Edit Libraries** window.

Extracting Equivalent Cut Depth Based on Multiple Curves

The manufacturing effects on core loss computation can be considered in Maxwell transient simulation. To consider the manufacturing effects, you need to input a parameter called **Equivalent Cut Depth**. The default value for this parameter is 1 mm, which is close to the average results extracted from some measurements.

A tool is featured to extract this parameter. If you have multiple core loss curves varying with flux density under different number of cut edges (two cut edges are produced by each cut) measured by the single sheet tester method, you can use this tool to extract the parameter.

To be able to use this feature, you must first set the **Core Loss Model** of the material.

To extract the equivalent cut depth:

1. Click **Tools** > **Edit Configured Libraries** > **Materials**. The **Edit Libraries** dialog box appears.
   - Alternatively, in the project tree, you can right-click **Materials**, under **Definitions**, and select **Edit Libraries**. The **Edit Libraries** dialog box appears.
   - Or select an object, and click **Modeler** > **Assign Materials**. The **Select Definition** dialog box appears.
2. Click **View / Edit Material**.
   The **View / Edit Material** dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select **Electrical Steel**, or **Power Ferrite**, or **Hysteresis Model**, from the **Core Loss Model** drop-down menu. The **Equiv. Cut Depth** property is available under **Core Loss Model**, and the **Extract Equiv. Cut Depth** is added in the **Calculate Properties for** drop-down menu.
4. Select **Extract Equiv. Cut Depth** from the **Calculate Properties for** drop-down menu. The **Extract Equiv. Cut Depth** dialog box appears. Click **Add** to add a new row for a loss curve input. Enter a value for number of cut-edges in the **Number** column, and enter the loss curve data in the **Edit Dataset** column.
5. Click the **Edit Dataset…** button. The **Edit Dataset** dialog box appears. Enter values for loss curve **P** and the sinusoidal flux density **B**. Click **OK** to return to the **Extract Equiv. Cut Depth** dialog box .
6. Select the type of curve you want to plot. You can plot for **Selected Number** or **All Numbers**, and plot the **Original Curve** or the **Regression Curve**. To identify the equivalent cut depth, you need to input various measured loss curves with different number of cut edges.
7. Select the **Core Loss Unit**.
8. Type a value for **Mass Density**. This field can only be edited when **Core Loss Unit** is w/lb or w/kg.
9. Enter a value for the **Band Width**. As the input data changes, the parameters are dynamically updated with both the specified unit and the standard unit (w/kg).
10. Click **OK**.

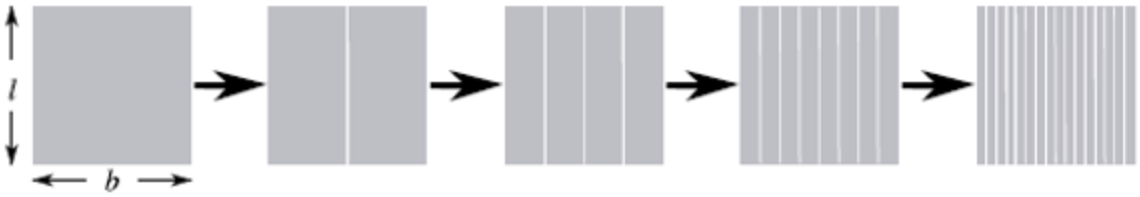
    The **View/ Edit Material** dialog box reappears. The extracted values are updated as new default values.
11. Click **OK** to close the **View / Edit Material** dialog box.
12. Click **OK** to close the **Select Definition** or **Edit Libraries** dialog box.

Parameter Identification Based on Single-Sheet Tester

The device for single-sheet tester is shown below:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The single-sheet specimen to be measured with different numbers of cut edges is shown below:



At each cut-edge number N_j, we can measure a loss curve P_{ij} varying with the amplitudes of sinusoidal flux density B_i. Then equivalent cut depth is identified by minimizing the total error between the predicted and measured losses:

$$\varepsilon = \sum_{i=1}^{n} \sum_{j=1}^{m} \left(P'_{ij} - P_{ij} \right)^2$$

where

- P'_{ij} and P_{ij} are predicted and measured losses for all points, respectively,
- m is the number of measured loss curves,
- n is the number of points for each loss curve.

In a single-sheet tester, the predicted losses can be derived from

$$P'_{ij} = P_{i0} + \Delta P_{ij} = P_{i0} \cdot \left(1 + w_e N_j / b \right)$$

where:

- P_{i0} are the intact losses, the loss curve without cut-edge effects,
- ΔP_{ij} are the increased losses due to the cut-edge effects,

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- w_e is Equiv. Cut Depth to be identified, and
- b is the original band width of the single sheet.

In general, the intact losses P_{i0} are unknown, and can be determined by

$$\frac{\partial \varepsilon}{\partial P_{i0}} = \frac{\partial \varepsilon}{\partial P_{i0}} \left(\sum_{i=1}^{n} \sum_{j=1}^{m} \left[P_{i0} \left(1 + w_e N_j / b \right) - P_{ij} \right]^2 \right) = 0$$

based on which we derive

$$P_{i0} = \frac{\sum_{j=1}^{m} P_{ij} (1 + w_e N_j / b)}{\sum_{j=1}^{m} (1 + w_e N_j / b)^2}$$

Similarly, based on

$$\frac{\partial \varepsilon}{\partial w_e} = 0$$

we get

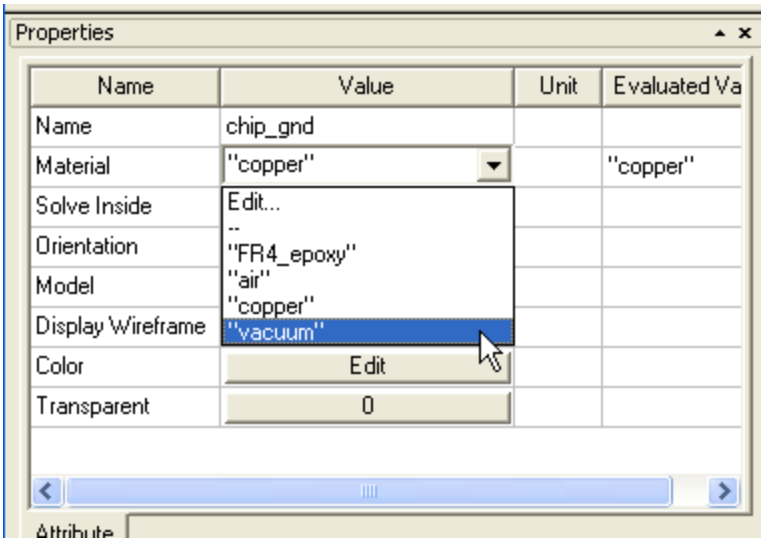
$$w_e = b \frac{\sum_{i=1}^{n} \sum_{j=1}^{m} (P_{ij} - P_{i0}) P_{i0} N_j}{\sum_{i=1}^{n} \sum_{j=1}^{m} (P_{i0} N_j)^2}$$

In above equations, intact losses P_{i0} depend on parameter w_e, which in turn depends on the intact losses. An iteration process is required to get the solution. Therefore, to identify the equivalent cut depth, and show the regression curves, we need to input various measured loss curves P_{ij} at different number of cut edges N_j, as well as the band width of the single sheet b.

Assigning Materials from the Object Properties Window

The Properties dialog for each object includes a materials property. If you click on the current material property you see a drop down list that includes an *Edit* command and a list of materials in the current project. You can select from the list of current materials to assign the selected material to that object.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

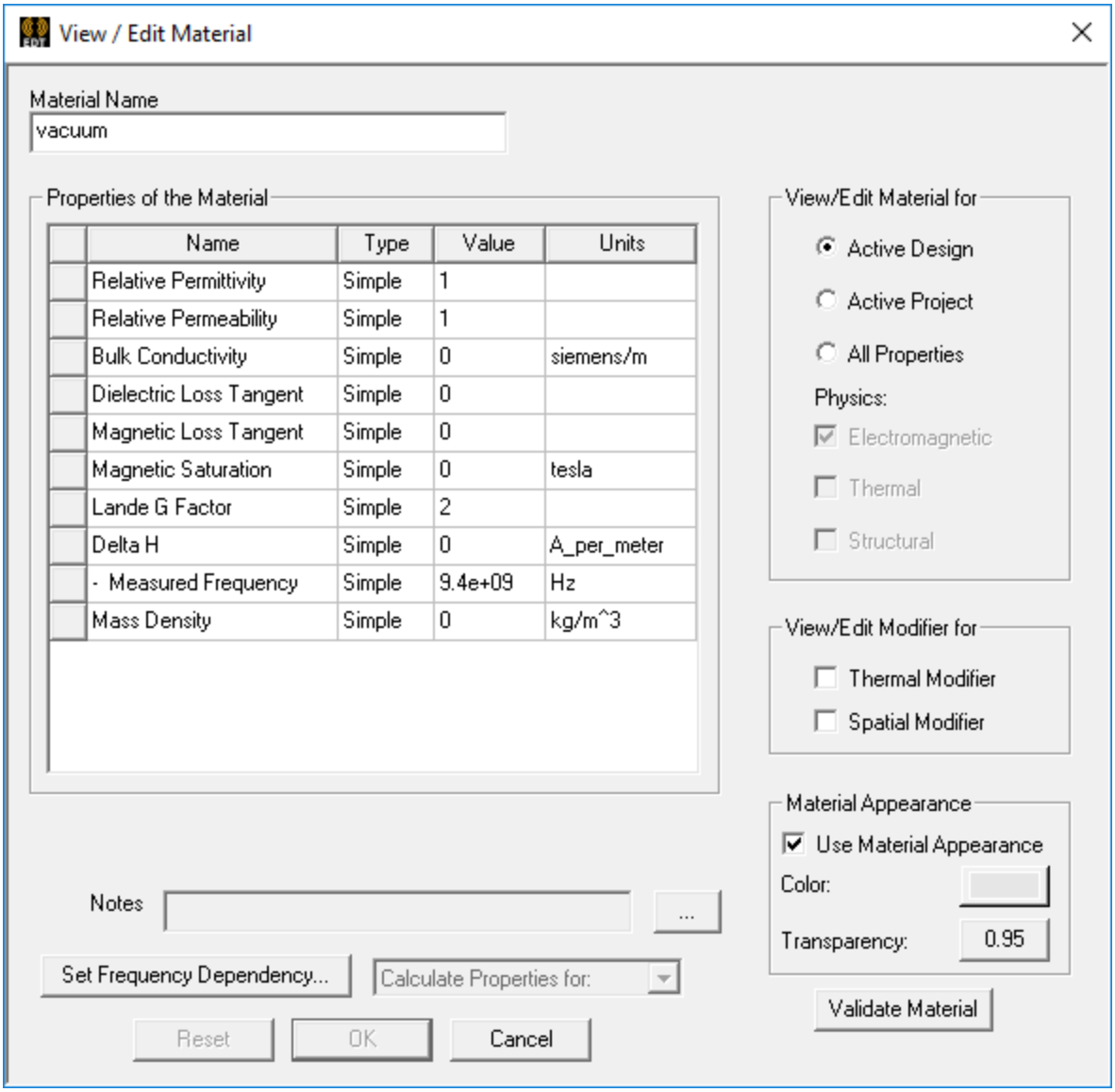


Viewing and Modifying Material Attributes

1. From the **Select Definition** window, select a material to view or modify, and click **View/Edit Materials**.

   The **View/Edit Material** window appears. The material name and its property values are listed.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

By default, not all of the available properties are displayed in the materials table. Only the properties commonly used by the product are displayed.

To view the complete table of properties, see Filtering Materials.

2. Under **View/Edit Material for**, select:
   - **Active Design** to display properties used in the active design.
   - **Active Project** to display properties used in the active project.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **All Properties** to display all properties available. This enlarges the table of properties to show all properties possible. You can use the scroll bars or size the dialog to see all properties. When **All Properties** is selected, the following **Physics** classification options are enabled to show or hide properties based on simulation type:
  - **Electromagnetic**
  - **Thermal**
  - **Structural**

> **Note:**
>
> If a material is edited in a design type for which the **Physics** type has not been set (e.g., an HFSS design but Electromagnetic physics type was not set), the **Physics** type will be automatically set in the material.

3. You can modify the material as follows:
   a. Provide a new name for the material in the **Material Name** text box.
   b. Depending on material property selected, the types available may be **Simple**, **Anisotropic**, **Vector**, or **Nonlinear**, as required for that property. Magnetostriction and Inverse Magnetostriction take custom properties, see Calculating the Properties for Magnetostriction and Calculating the Properties for Inverse Magentostriction for more details.

      For **Simple**, you provide a value or variable.

      For **Anisotropic**, you provide tensor values.

      For **Vector**, you provide a **Vector Mag**.

      For **Nonlinear**, you provide a Data Set.

   c. Provide new material property values in the **Value** boxes.
   d. Change the units for a material property.
   e. Enable the **Use Material Appearance** check box to specify a color and transparency.

> **Note:**
>
> Materials stored in the global material library cannot be modified.

4. If you want to add descriptive notes for the new material, click the ellipsis button [...] to the right of the **Notes** field. This opens a dialog box in which you can enter text. Pressing **Enter** or clicking **OK** saves the note. To enter multiple lines of notes, use **Ctrl+Enter** at the start of each new line.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click **OK** on the **View/Edit Materials** dialog to add the new material to the material library.
6. Click **OK** to save the changes and return to the **Select Definition** window.

> **Warning:**
>
> If you modify a material that is assigned in the active project after generating a solution, the solution will be invalid.

Setting the Material Threshold

The material thresholds tell the **Maxwell 2D** and **Maxwell 3D** solvers how to deal with conductors and insulators. Materials with conductivities above the **Perfect Conductor** threshold are treated as having infinite conductivity and surface current only. Materials with conductivity values below the **Insulator/Conductor** threshold are treated as insulators with no conductivity and no current carrying capability. Materials that fall between the two thresholds are treated as normal conductors that can carry current throughout the volume of the material.

1. Click **Maxwell** > **Design Settings.** The **Design Settings** dialog box appears.

   The material thresholds are set in the **Material Thresholds** tab.
2. Type a value in the **Perfect Conductor** text box.
3. Type a value in the **Insulator/Conductor** text box.
4. Click **OK**.

Validating Materials

The Ansys Electronics Desktop can validate a material's property parameters for an Ansys EM software product. For example, the software will check if the range of values specified for each material property is reasonable.

If a material's property parameters are invalid, an error message will appear in the lower-right corner of the **View/Edit Material** window. If the parameters are valid, a green check mark will appear there.

To validate the material attributes listed in the **View/Edit Material** window:

- Select **Active Design**, **Active Project**, or **All Properties**, and then click **Validate Material**.

Copying Materials

1. In the **Select Definition** window, select the material you want to copy, and then click **Clone Material** or right-click the selected material and select **Clone** from the shortcut

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

menu.

2. To modify the material's attributes, follow the directions for modifying materials.
3. Click **OK** to save the copy in the active project's material library.

Removing Materials

1. In the **Select Definition** window, select one or more materials you want to remove from the active project's material library.
2. Click the **Remove Material (s)** button or right-click the selected material and click **Remove** on the short-cut menu.

The material is deleted from the project material library.

> **Note:**
>
> The following materials cannot be deleted:
>
> - Materials stored in Ansoft's global material library.
> - Materials that have been assigned to objects in the active project.

In a project library, you may want to use the **Tools** > **Project Tools** > **Remove Unused Definitions** command to remove selected materials definitions that your project does not require.

Exporting Materials to a Library

1. In the **Select Definition** window, select the material you want to export.
2. Click the **Export Material to Library** button, or right-click the selected material and click **Export** from the short cut menu.

   The **Export to material library** file browser appears.

3. Click **PersonalLib** to export the material to a local project directory, accessible only to the user that created it.

   Click **UserLib** to export the material to a library that is shared by more than one user, usually in a central location.

4. Type the library's file name and then click **Save**.

Sorting Materials

You can change the order of the materials listed in the **Select Definition** window. You can sort the list of materials by name, library location, color, or material property value. To change the order of the listed materials:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

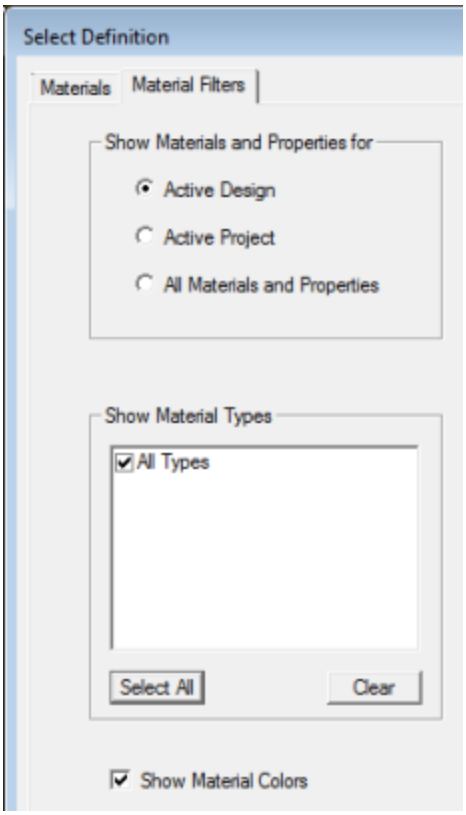
- Click the column heading by which you want to order the materials.

If the arrow in the column heading points up, the material data will be listed in ascending order (1 to 9, A to Z) based on the values in the column you chose. If you want the material data to be listed in descending order (9 to 1, Z to A), click the column heading again. The arrow will point down.

> **Note:**
>
> By default, not all of the available properties are displayed in the materials table. Only the properties commonly used by the product are displayed. To view the complete table of properties, see Filtering Materials.

Filtering Materials

If you want to remove certain materials or material properties from the list in the **Select Definition** window, use the filter options under the **Material Filters** tab. You can filter out materials based on the product or library with which they are associated. You can also filter out material properties and types of material properties. And you can remove the filtering in order to see all available material properties.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To filter materials or material properties listed in the **Select Definition** window, using the choices in the **Materials** tab:

1. The text field under Libraries lists the libraries for the project. Selecting the listed library highlights it and cause the table to display the materials in that library.
2. Above the Libraries area, you can check or uncheck boxes to show or hide Project Definitions and All Libraries.

- With both unchecked, nothing appears in the materials table. With both checked, the table shows all materials and highlights those used in the project.
- With only Project Definitions checked, the materials table shows only the materials used in the project.
- With All Libraries checked, the table displays all materials, but may not show all available properties.

To filter out or show additional materials/properties in the **Materials** tab:

1. Click the **Material Filters** tab.
2. Under **Show Materials and Properties for**, select one of the radio buttons:

- **Active Design** to display materials/properties used in the active design.
- **Active Project** to display materials/properties used by the active project.
- **All Materials and Properties** to display all materials and properties available. Selecting this enlarges the table of materials shown under the **Materials** tab to show all materials possible. You can use the scroll bars or size the dialog to see all materials.

3. Under **Show Material Types**, select materials types to display on the **Materials** tab. Click **Select All** to select all of the types listed. Click **Clear** to clear all selections.
4. If you check Show Material colors, the Materials tab of the **Select Definition** window includes a color for material colors. See Viewing and Modifying Material Attributes.
5. Click the **Materials** tab to save your selections.

Click **Cancel** to revert back to the last saved selections.

Working with Material Libraries

There are three different materials libraries in Maxwell: a *system library*, a *user library*, and a *personal library*.

The library files that ship with Electronics Desktop are stored under the **`syslib`** directory. These libraries are intended to be read-only and should not be modified. They are available for any material assignment in any project.

In addition to the system libraries, Electronics Desktop recognizes two user-configurable library structures, called the *User Library* and the *Personal Library*. These are used to add user (or company)-defined materials. Customarily, **`userlib`** is a network repository for proprietary or

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

corporate definitions available to all seats in an enterprise, while **personalLib** contains project and design-specific libraries as needed by individual designs.

A root library directory is set up at installation. If none is specified, the default is the root Electronics Desktop directory.

Materials from all libraries are available for use in projects.

Editing Libraries

There are two different methods of editing libraries:

- Right-click **Materials** in the project window to display the **Edit Libraries** shortcut menu. Clicking displays the **Edit Libraries** window.

  Editing definitions from the project window does not modify the configured libraries for any particular design, since this is editing in general.

- Using **Tools** > **Edit Libraries** > **Materials** from the menu bar takes the current design into account and adds any new libraries to the configured list for the design.

Editing Material Definitions

When editing material definitions or adding a material to the user/personal library, use the menu command (**Tools** > **Edit Libraries** > **Materials**) rather than the project tree. Editing or defining definitions by right-clicking in the project tree window does not modify the configured libraries for any particular design – it only edits the material in general. When you use the menu command, the current design is considered, and any new materials are made available to assign in that current design. New or modified materials can also be exported to a library in the User Library or Personal Library from the **Edit Libraries** dialog.

To edit a material's definition:

1. Click **Tools** > **Edit Libraries** > **Materials**.

   The **Edit Libraries** window appears.

2. In the **Search Parameters** section under **Libraries**, select the library database. By default the **[sys] Materials** system library is selected.

3. Select the material you want to edit, and click **View/Edit Materials**.

   The **View/Edit Material** window appears.

4. Make the desired changes to the **Material Name**, the **Material Coordinate System Type**, and the properties listed in the **Properties of the Material** section.
5. From the **View/Edit Material for** section, select whether the changes shall apply to the (**Active Design**, **Active Project**, or **All Properties**).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. You may optionally set a *Thermal Modifier* to specify that certain material properties have characteristics that vary with temperature.
7. Click **Validate Material** to make sure your changes are valid.

   If the material properties are valid, a green check mark appears below the **Validate Material** button.
8. Optionally, use the **Calculate Properties for...** drop-down menu to calculate properties for a permanent magnet or a Non-Linear Permanent Magnet.
9. Optionally, to calculate properties for core loss, select **Core Loss Coefficient** from the **Calculate Properties for...** drop down list.
10. When you are done making changes to the material properties, click **OK**.
    The **Edit Libraries** window reappears.
11. If you are finished in the **View/Edit Material** window, click **OK** to close it.

Setting a Thermal Modifier

Materials used in 3D Designs, and for sheet objects in 2D Designs, may have a thermal modifier assigned to change specific material properties with temperature, primarily for use when coupling to thermal analysis.

To specify Thermal modifiers for a material:

1. In the View/ Edit materials window, you must enable the View/ Edit check box for **Thermal Modifier**.

   This causes the properties of the Material table to expand to include a **Thermal Modifier** column. By default, the **Thermal Modifier** property is set to **None**.
2. Selecting **Edit...** from the drop-down menu, rather than **None** causes the **Edit Thermal Modifier** dialog box to appear.
3. Select the **Expression** radio button to display the **Parameters Modifier** text field or the **Quadratic** radio button to display the tabs for **Basic Coefficient Set** and **Advanced Coefficient Set**.

   - With **Expression** selected, you can write an equation for a thermal modifier in the Parameters **Modifier** text field.

     Checking **Use temperature dependent data set** disables the Modifier text field. You can then use the drop-down menu to select Add/Import Dataset. This lets you define the thermal modifier as a data set.
   - With the **Quadratic** radio button selected, in the **Basic Coefficient Set** tab, you can edit fields for the TempRef and units, and fields for C1 and C2 for the following

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

equation:

$P(Temp) = Pref[1+ C1(Temp - TempRef) + C2(Temp - TempRef)^2]$

where the Pref is defined as the reference relative permittivity.

- With the **Quadratic** radio button selected, in the **Advanced Coefficient Set** tab, you can edit fields for lower and upper temperature limits (TL and TU respectively) and select their units from the drop down.

  You can also edit the constant value limit for the thermal modifier values outside the limits. By default, these are automatically calculated. Uncheck **Auto Calculate TML, TMU** to specify new values for thermal modifier lower (TML) and thermal modifier upper (TMU) limits.

4. Click **OK** to accept the edits and return to the View/ Edit materials window.

> **Note:**
>
> Thermal modifier for permeability is disabled when temperature-dependent BH curves are specified.

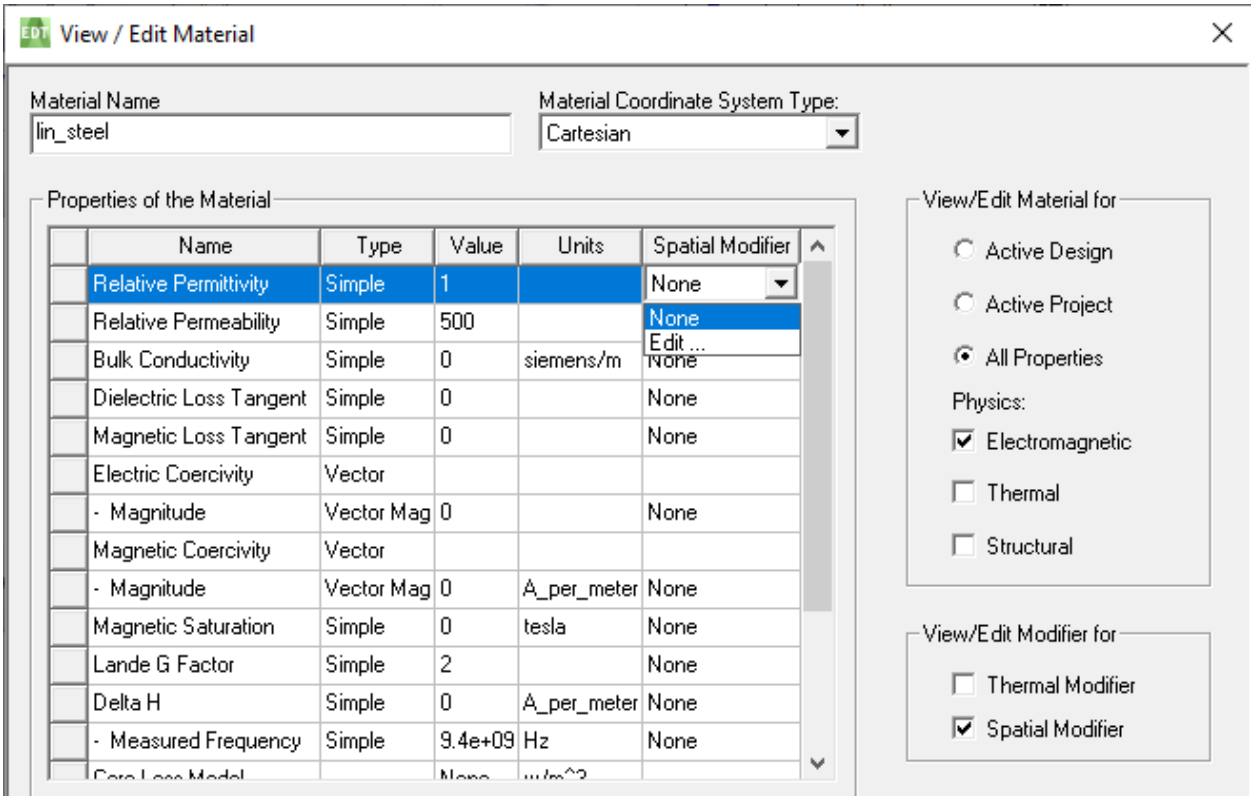
Setting a Spatial Modifier

Maxwell magnetic transient solvers support use of a 3D dataset as a **Spatial Modifier** to modify spatially varying properties of conductivity, permeability, and permittivity. In general, Thermal Modifier and **Spatial Modifier** cannot be applied to one material property simultaneously.

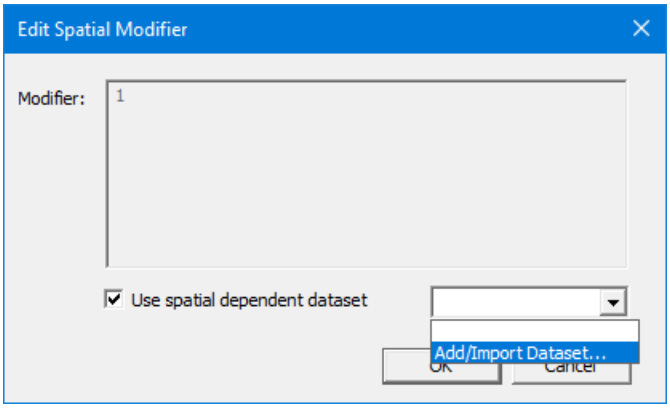
To specify a 3D dataset as spatial modifiers for a material:

1. In the View/Edit materials dialog you must enable the **Spatial Modifier** check box in the **View/Edit Modifier for** panel. This causes the properties of the Material table to expand to include a **Spatial Modifier** column. By default, the **Spatial Modifier** property is set to **None**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



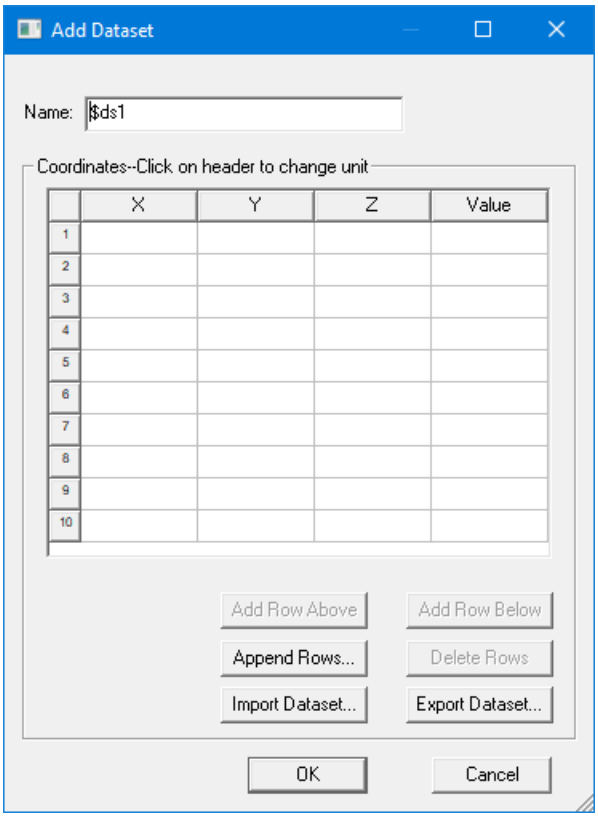
2. Selecting **Edit...** from the drop-down menu causes the **Edit Spatial Modifier** dialog box to appear.



3. Check **Use spatial dependent dataset**. If a predefined dataset is available, you can select it. Or you can select **Add/Import Dataset**. This opens the **Add Dataset** dialog box for specifying 3D object coordinates and associated values of the material property (conductivity, permeability, or permittivity) to be used in the design. (You can click the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

column heads to select the units.)



You can also import a tab delimited text file containing predefined coordinates and property values.

Imported datasets must be tab-delimited files (*.tab) containing sets of X, Y, and Z coordinates and associated property Values. For example:

```
2 4 4 68

2 5 5 70

3 5 5 71
```

You can also edit existing datasets as needed.

4. The spatial modifier expression models the material property change with respect to location along the object. The expression appears in the Spatial Modifier column for the associated property.

   The expression is of the form: `clp(`*`dataset_name`*`,X,Y,Z)`

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where `clp` is the closest point interpolation function, and `dataset_name` is the name of the design dataset containing the set of 3D object coordinates and associated property values to be used in the design. For example: `clp(ds2,X,Y,Z)` is the expression that uses the `ds2` dataset.

Importing and Converting Materials

To preserve custom materials, you need to import them from a .mat file into the latest version of the user library.

> **Note:**
>
> The **.mat** extension shows up as a shortcut in the file browser.

To import these materials from previous versions, you must have at least one design loaded in the project window.

To translate custom materials to the latest version:

1. Click **Maxwell** > **Translate Material Database**.

   The **Translate Legacy Material Database** appears.

2. Enter the name of the old database in the **Legacy Material DB Name** box. You can also click the **...** button to locate the database.
3. Enter a name for the new database in your current project in the **User Material DB Name** box.
4. Click **Translate**.

Setting the Temperature of Objects

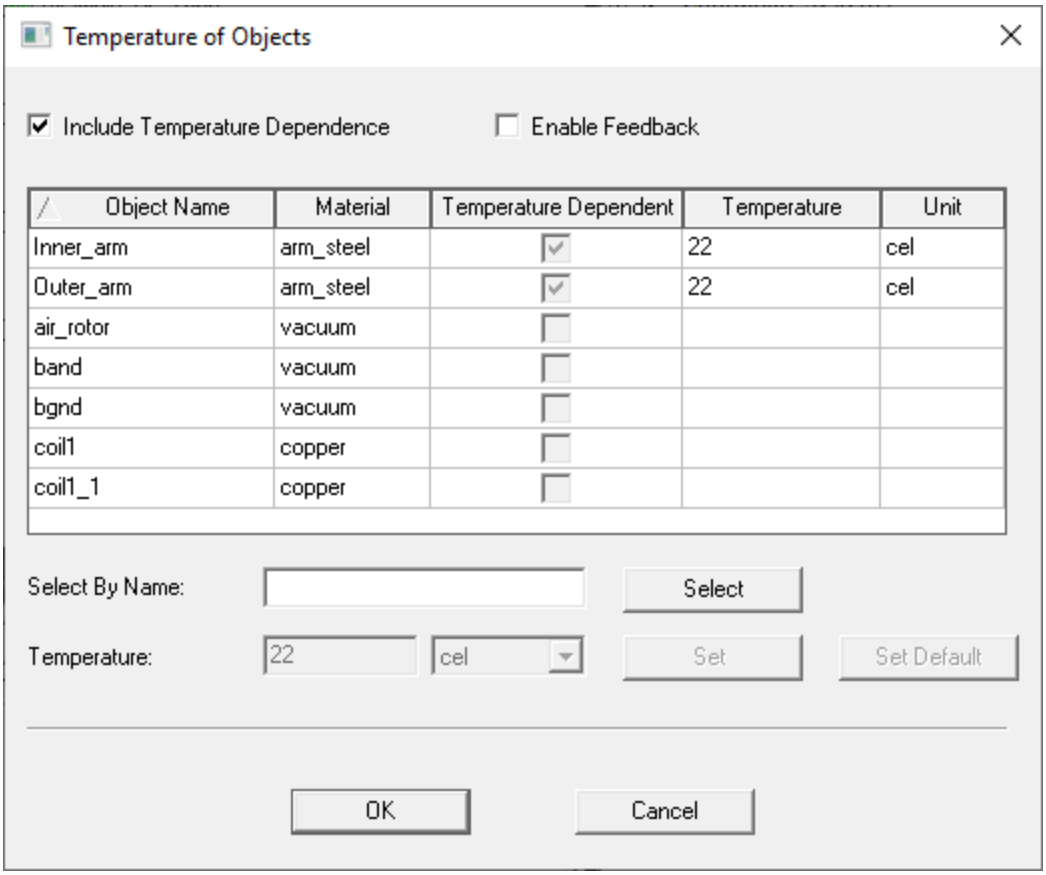
To set the temperature of temperature-dependent objects:

1. Use the **Maxwell** > **Set Object Temperature** command to display the **Temperature of Objects** dialog box.

   - For all design types, the dialog box includes a table of the objects in the design. It lists the **Object Name**, the **Material** assigned to the object, a non-editable **Temperature Dependent** check box to show whether that object has temperature-dependent features (such as those specified by Setting a Thermal Modifier). For temperature-dependent items, the remaining columns give the associated

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Temperature and **Unit** of measure.



2. Maxwell allows you to enable/disable temperature dependency independently for each design in the same project. To include temperature dependency and enable the editing features, check **Include Temperature Dependence**. This makes the table rows editable.

    - The headers for each of the columns – except the **Temperature Dependent** column – include sort direction capability. You can invert the sort direction in each column by clicking the header. If the list is longer than the display, you can use a scroll bar on the right of the table.
    - In the **Select By Name** field, you can enter the name of the object (or boundary for 3D AC magnetic designs) you want to select, then click **Select** to highlight the selected row. You can make multiple selections.

3. To set the initial temperature for selected **Temperature Dependent** objects (or boundaries):

    a. Type a value, an existing variable name, or a spatial modifier expression in the **Temperature** text field; then select the unit of measure from the drop down menu.

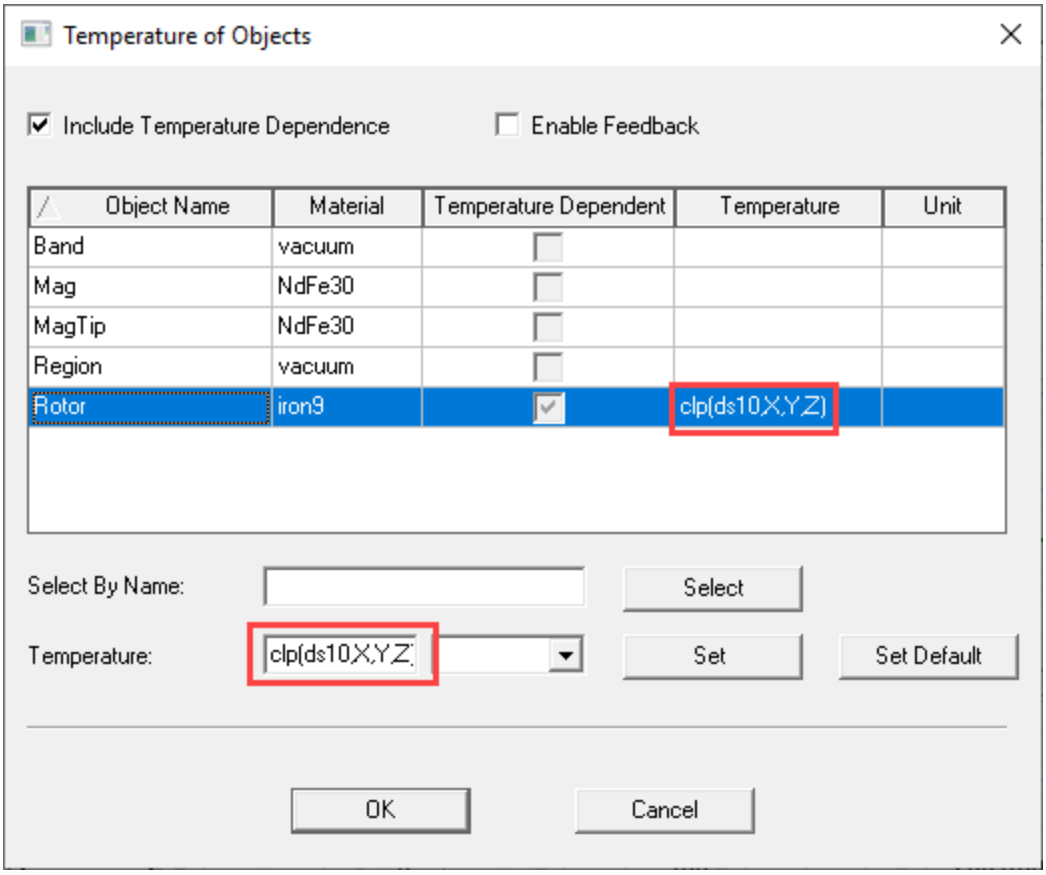
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Refer to step 4 for detailed information on using a *spatial modifier expression*.

b. Click **Set** to apply the value to the selected temperature dependent objects (or boundaries); or click **Set Default** to make the specified values the default. (If you click **Set Default**, the rows for the selected objects (or boundaries) display the Temperature value and units.)

> **Note:** You can also edit the **Temperature** and **Unit** for temperature dependent objects (or boundaries) directly in the table.

4. *Optionally*, for 2D and 3D transient designs, you can enter a *spatial modifier* expression in the **Temperature** field in the **Temperature of Objects** window as shown in the figure below. The spatial modifier expression models the temperature material property change with respect to location along the object. You can enter the expression either in the Temperature column at the top, or the Temperature text edit field at the bottom.

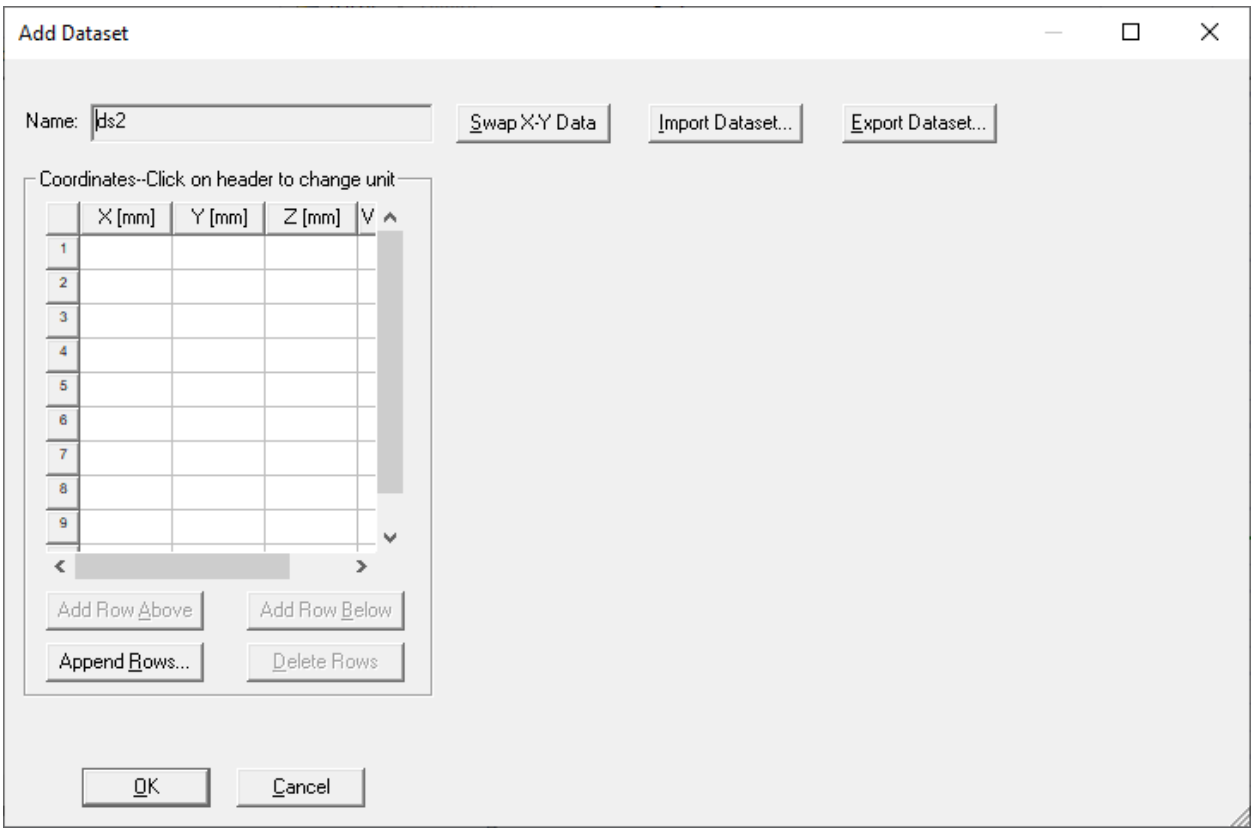


The expression is of the form: `clp(dataset_name,X,Y,Z)`

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where `clp` is the closest point interpolation function, and `dataset_name` is the name of the design dataset containing the set of 3D object coordinates and associated temperatures to be used in the design. For example: `clp(ds2,X,Y,Z)` is the expression that uses the `ds2` dataset.

If the expression contains an undefined dataset, when you click **OK** and close the **Temperature of Objects** dialog box, an **Add Dataset** dialog box for creating a new design dataset will pop up for inputting or importing the spatial data. You can click the column heads to select the units.

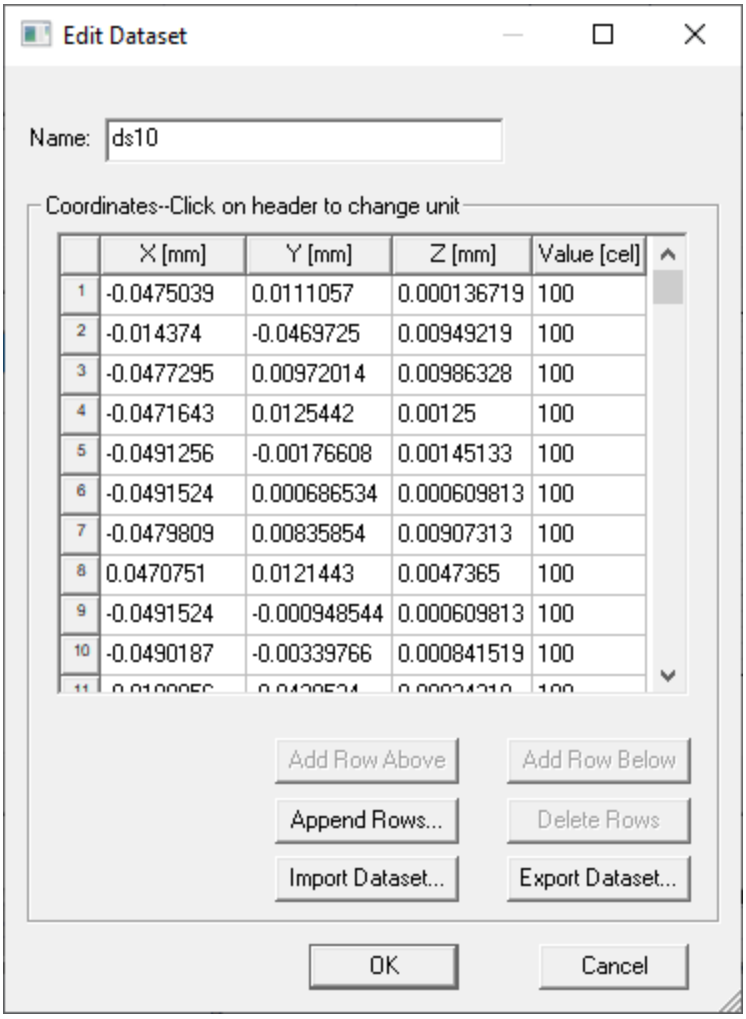


Imported datasets must be tab-delimited files (*.tab) containing sets of X, Y, and Z coordinates and associated temperature Values. For example:

```
2 4 4 68

2 5 5 70

3 5 5 71
```

You can edit existing datasets as needed.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. When **Include Temperature Dependence** is enabled, you can edit the material for an object (or boundary) by clicking on the material in the row to display a drop-down menu listing the current material, and an **Edit...** menu item.

   Select **Edit**... to display the Materials window in which you can edit the current material – or select a different material for the object (or boundary). The material editor also allows you to set a thermal modifier. When finished, close the dialog to return to the Temperature of Objects window. If you set a thermal modifier, the Temperature Dependent column for that object (or boundary) will be checked.

6. If the **Enable Feedback** option is checked, you can perform thermal analysis based on a Maxwell solution.
7. To close the **Temperature of Objects** window and accept the changes, click **OK**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reverting Objects to Initial Temperature

When a design is enabled for thermal feedback, a **Revert to Initial Temperature** menu item is added to the context menu of the solution setup item and analysis setup folder in the project tree. This menu item is enabled when there are exported thermal data from Mechanical or when thermal data has been incorporated into the current Maxwell solution. Choosing **Revert to Initial Temperature** reverts the temperatures of objects to their initial temperatures.

Setting Deformation of Objects for Stress Feedback

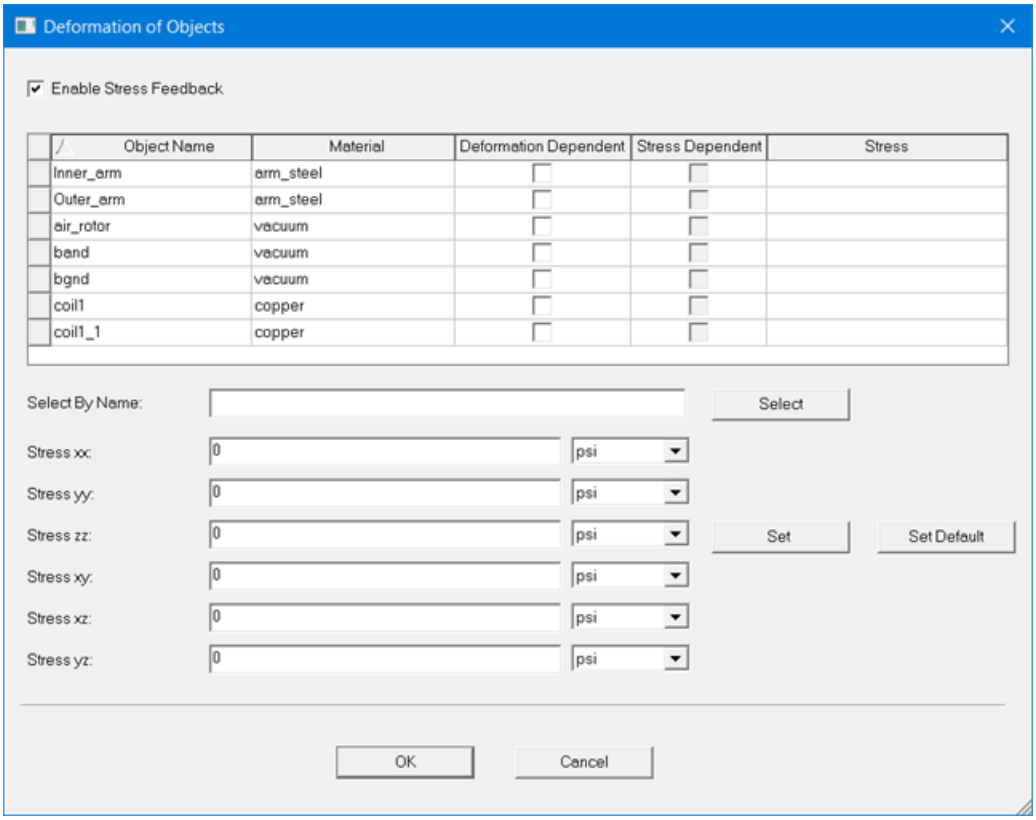
For Maxwell standalone 2D and 3D magnetostatic and transient designs, and for Maxwell designs that are coupled to Ansys Structural via Workbench, you can enable objects for stress feedback in the **Deformation of Objects** dialog box.

Additionally, for 2D and 3D magnetostatic and transient designs that include objects with nonlinear Magnetostriction and/or Inverse Magnetostriction properties, you can optionally enter initial stress values. Note that you cannot include an initial stress value for the Magnetostriction\Custom\Matrix property type.

To set the deformation of stress-dependent objects:

1. Use the **Maxwell** > **Set Object Deformation** command to display the **Deformation of Objects** dialog box.
    - For AC magnetic and electrostatic designs, this dialog box includes a table of objects in the design, and a check box to **Enable Stress Feedback**. The first column lists the **Object Name**. The second column lists the **Material**. The **Deformation Dependent** column displays a check box to show whether that object has deformation-dependency enabled.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



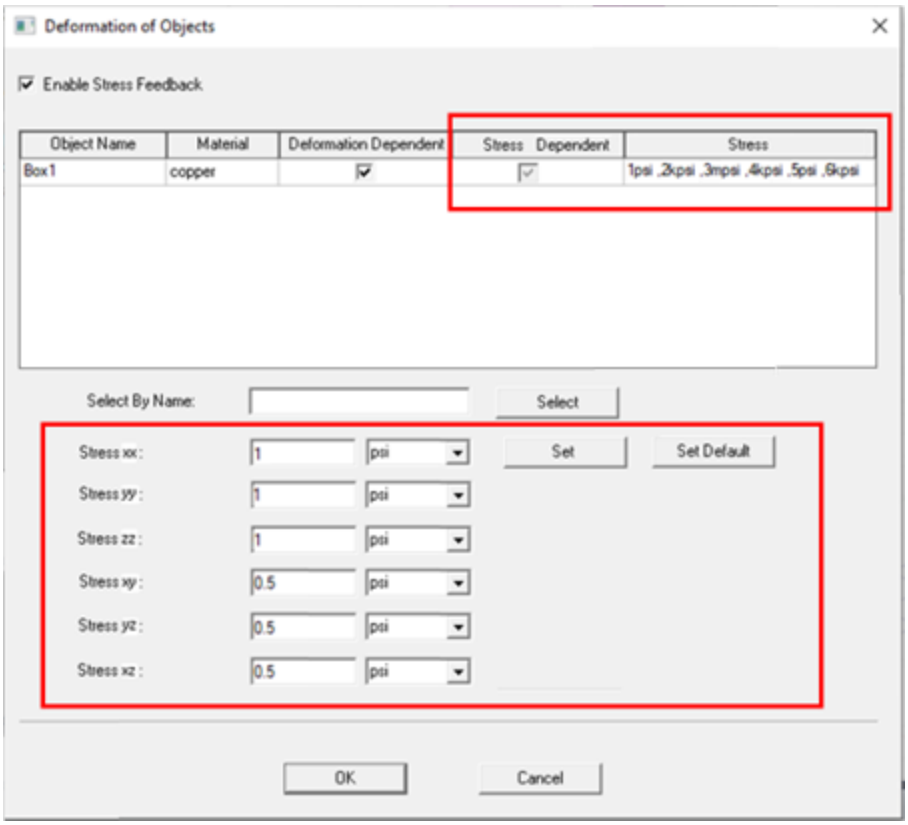
> **Note:**
>
> Objects without an assigned material, non-model objects, non-3D objects in Maxwell 3D designs, and non-2D objects in Maxwell 2D designs cannot be handled by the solver and thus do not appear in the table.

2. To include deformation dependency, check **Enable Stress Feedback**.
   This makes the table objects selectable. The headers for the **Object Name** column and the **Material** column include sort direction arrows. You can invert the sort direction in each column by clicking the header. If the list is longer than the display, you can use a scroll bar on the right of the table.

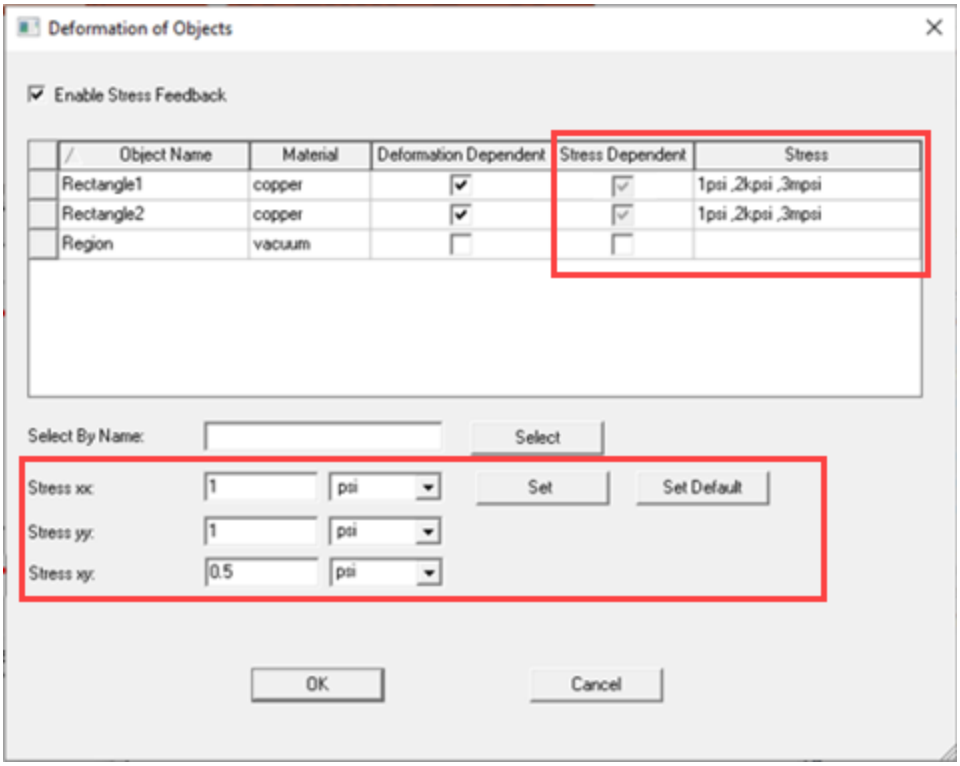
3. Check the **Deformation Dependent** check box for those objects you wish to include in the stress feedback analysis.
   For 2D and 3D magnetostatic and transient designs that include objects with nonlinear Magnetostriction and/or Inverse Magnetostriction properties, in addition to the **Enable Stress Feedback** check box, and the **Object Name**, **Material**, and **Deformation Dependent** columns, the table includes **Stress Dependent** and **Stress** values columns, which are read only. The dialog box also includes a section for entering initial stress

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

values and units.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

Switching the state of **Enable Stress Feedback** or **Deformation Dependent** for any object invalidates the existing solution. Existing solutions include all field and non-field solutions, such as force and torque. When these options are switched from on to off, existing field solutions include previously incorporated displacement data. On the next simulation, the design will resolve the last solved pass.

4. To close the **Deformation of Objects** dialog box and accept the changes, click **OK**.

Reverting Objects to Zero Displacement

When a design is enabled for stress feedback, a **Revert to Zero Displacement** menu item is added to the context menu of the solution setup item and analysis setup folder in the project tree. This menu item is enabled when there are exported displacement data from structural or when displacement data has been incorporated into the current Maxwell solution. Choosing **Revert to Zero Displacement** reverts the displacement values of objects to zero.

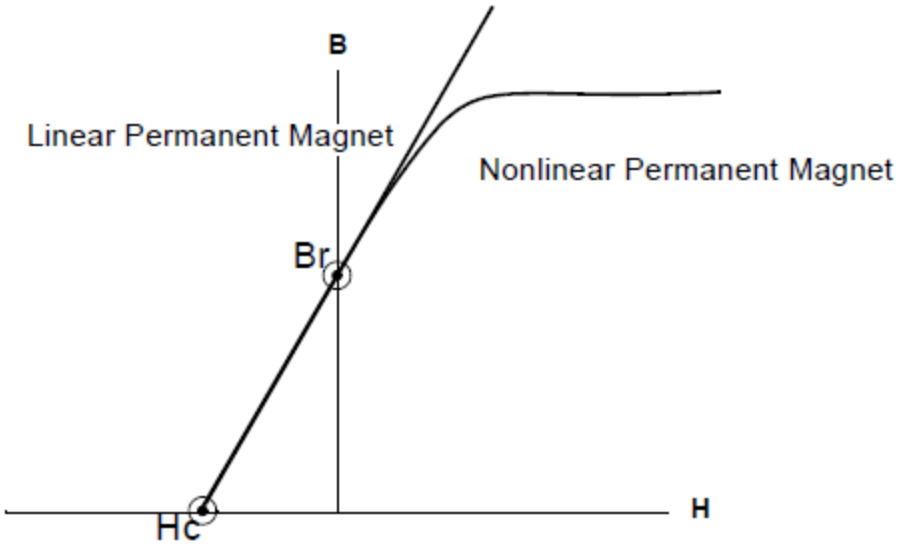
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Permanent Magnets

A permanent magnet is defined as a material that generates a magnetic flux due to permanent magnetic dipoles in that material.

Nonlinear vs. Linear Permanent Magnets

In general, permanent magnets are nonlinear and should be modeled via a B-H curve as shown below. The magnetic coercivity, H_c, is defined as the B-H curve's H-axis intercept, and the magnetic remanence, B_r, as its B-axis intercept.

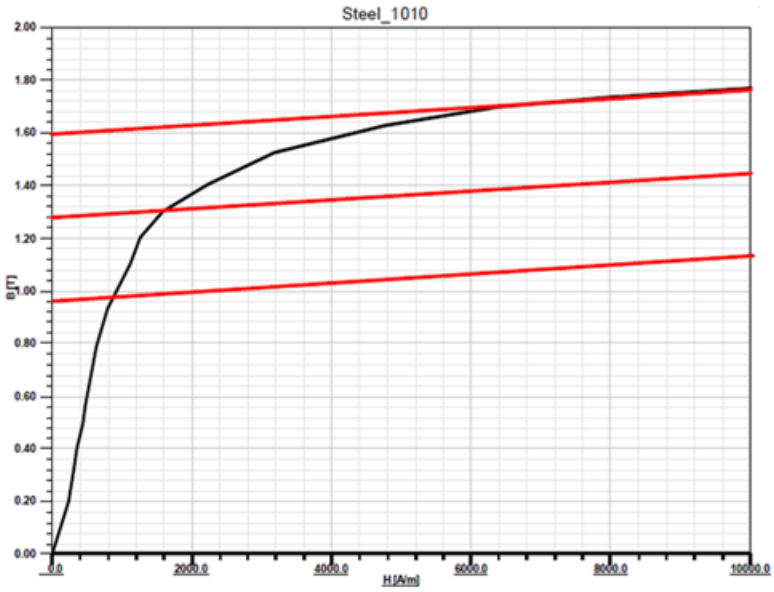


In many applications, however, the permanent magnet's behavior can be approximated using a linear relationship between **B** and **H**. In these cases, there is no need to create a nonlinear material. Enter the appropriate values of B_r or H_c for the material when defining its properties.

Compute Remanent B_r from B-H Curve

The value of the remanent B_r of the individual element after the magnetization field is computed is determined in such a way: after having located the operating point on the original non-remanent B-H curve, draw a line which is parallel to the original recoil curve with the slope of $\mu_0\mu_r$ and passes the operating point, the intersection of this line with B-axis is the remanent B_r as the result of the applied magnetization field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Temperature-Dependent Nonlinear Permanent Magnets

Temperature-dependent nonlinear permanent magnet properties may be specified in one of two ways:

- Scaling from a demagnetizing curve at a reference temperature
- Interpolating from multiple demagnetizing curves at different temperatures

Scaling from a Demagnetizing Curve

To consider the temperature dependence of the demagnetization behavior, the demagnetization curve is described by temperature-dependent parameters, which can be derived from supplier datasheets. For a better representation of any types of magnets by a function, it is advantageous to work with an intrinsic flux density $\boldsymbol{B_i}$ versus $\boldsymbol{H}$ curve, instead of a flux density $\boldsymbol{B}$ vs $\boldsymbol{H}$ curve. The relationship between $\boldsymbol{B_i}$ and $\boldsymbol{B}$ is

$$B = B_i + \mu_0 H \quad (1)$$

The temperature dependency of an intrinsic BH curve can be specified by two temperature dependent parameters: remanent flux density B_r and intrinsic coercivity H_{ci}. Here B_r is the value of B_i (or B) when H=0 and H_{ci} is the value of H at B_i=0. Both B_r and H_{ci} at temperature T can be expressed as

$$B_r(T) = B_r(T_0)\, P(T), \text{ and } H_{ci}(T) = H_{ci}(T_0)\, Q(T).$$

Here $P(T)$ and $Q(T)$ can be described using second order polynomials of temperature T as

$$P(T) = 1 + \alpha_1(T-T_0) + \alpha_2(T-T_0)^2 \quad (2)$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and

$$Q(T) = 1 + \beta_1(T-T_0) + \beta_2(T-T_0)^2 \quad (3)$$

where T_0 is the reference temperature, and α_1, α_2, β_1, and β_2 are coefficients which can be identified from supplier datasheets, and can be specified via the thermal modifier. In general, the thermal modifiers *P(T)* and *Q(T)* also can be described by any user-defined function of temperature *T*, in addition to the above second order polynomials.

Consequently, for an intrinsic demagnetization curve at reference temperature T_0 expressed by a data set (h_i, b_i), the demagnetization curve at temperature *T* can be derived by scaling *h* and *b* independently as (h_i•*Q(T)*, b_i•*P(T)*), where temperature-dependent scaling factors *P(T)* and *Q(T)* are the same as those for B_r and H_{ci}, as shown in Equations (2) and (3), respectively. This algorithm using the same scaling factors for all points of an intrinsic demagnetization curve is called the "shape preserving algorithm".

The shape preserving algorithm is applicable for shape-preserved intrinsic BH curves, in which all normalized intrinsic BH curves coincide with each other. **Figure 1** shows N4517 intrinsic and normal BH curves at different temperatures, and **Figure 2** shows all normalized intrinsic BH curves.

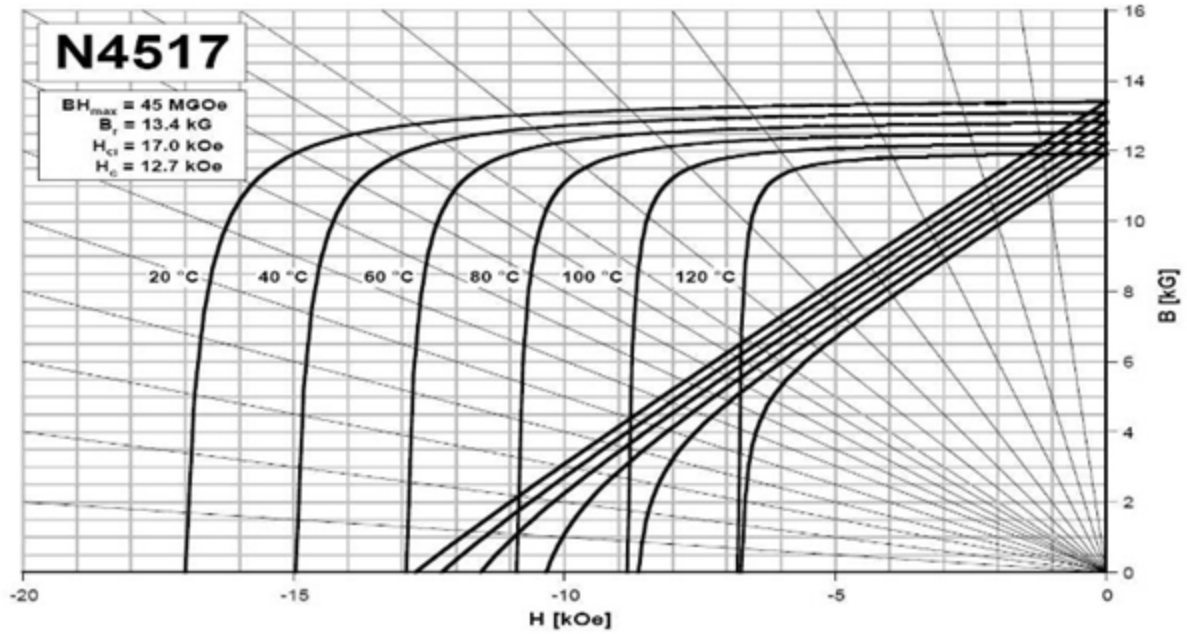


Figure 1 N4517 intrinsic and normal BH curves at different temperatures.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

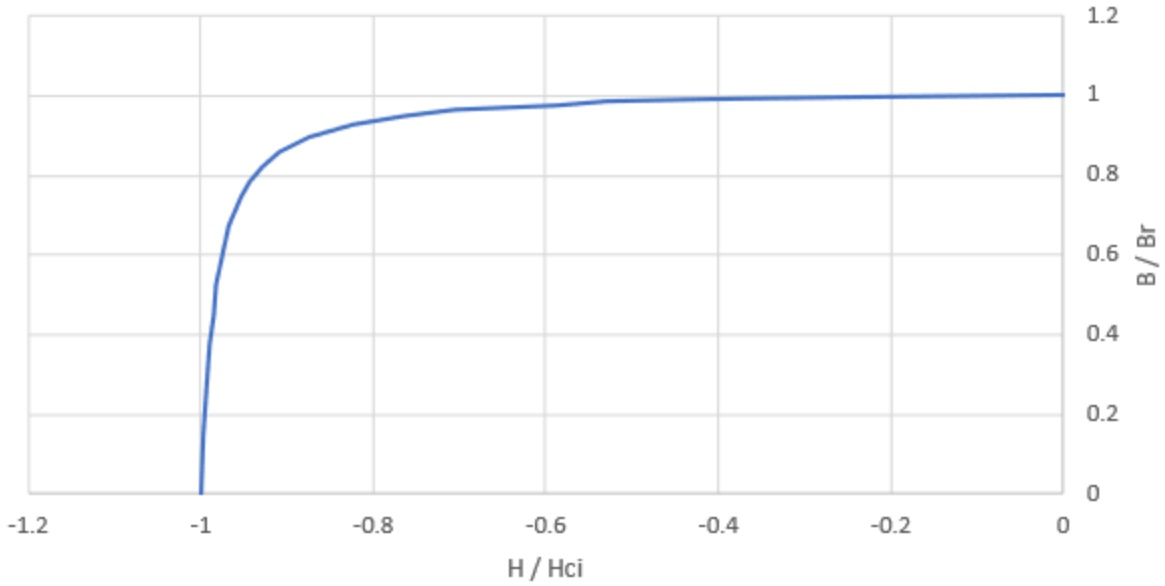


Figure 2 All normalized intrinsic BH curves for N4517 coincide each other.

Finally, the normal BH curve in the second and the third quadrants can be further derived through the conversion using equation (1).

Parameters T_0, α_1, and α_2 for temperature dependent factor *P(T)* can be specified via Thermal Modifier of the Relative Permeability field, and parameters T_0, β_1, and β_2 for temperature dependent factor *Q(T)* can be specified via Thermal Modifier of the Magnitude field, as shown in **Figure 3**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

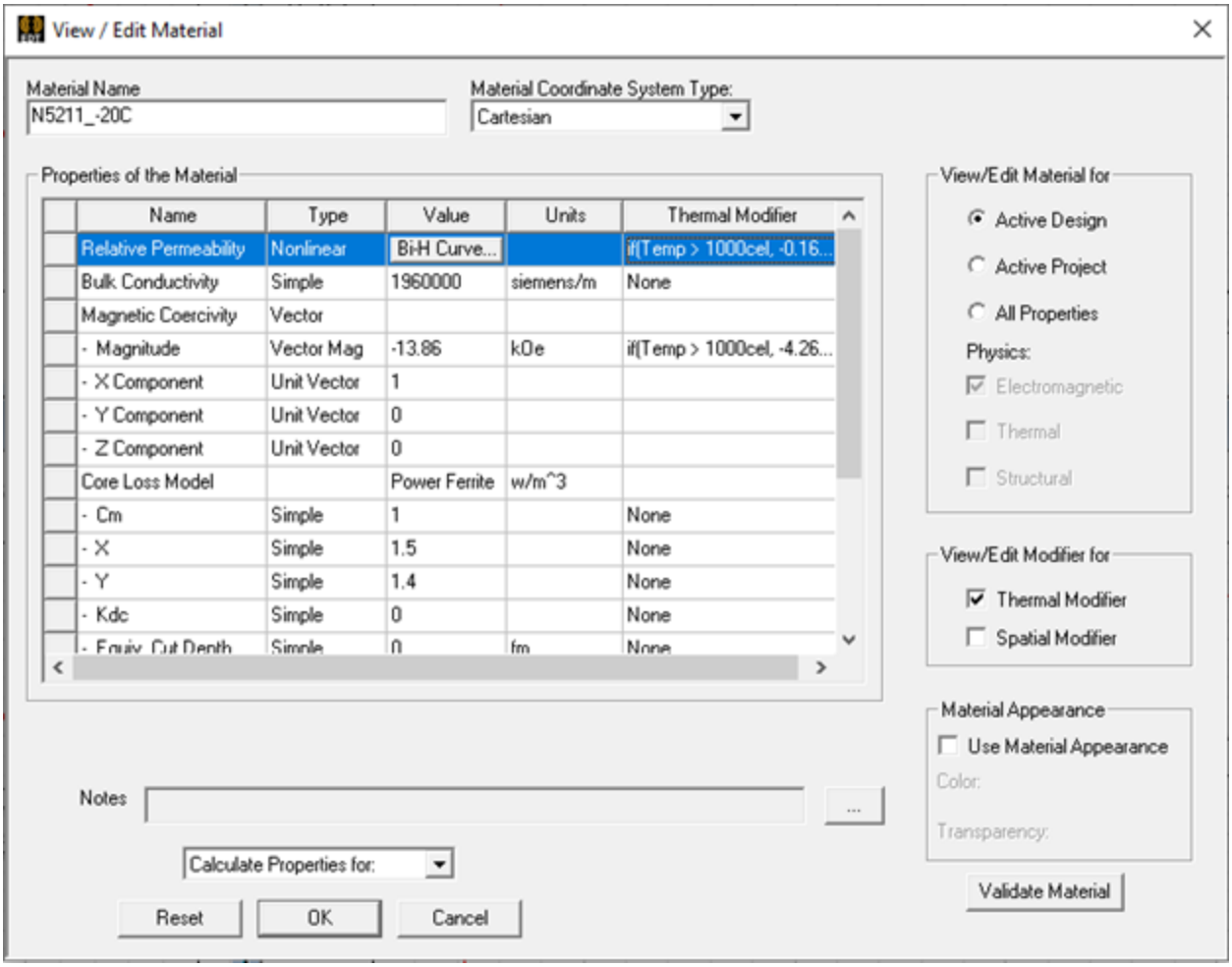


Figure 3: Thermal Modifiers for temperature dependent factors of B_r via Relative Permeability field and H_{ci} via Magnitude field, respectively.

> **Note:** Thermal Modifiers for both Relative Permeability and Magnitude fields must be input to correctly specify the temperature dependent factors for a shape preserved intrinsic BH curve.

Interpolating from Multiple Demagnetizing Curves

If the intrinsic BH curves are not shape-preserved, or temperature-dependent parameters are not available, two or more intrinsic BH curves at various reference temperatures are required. Suppose there are *m* inputs of temperature-dependent intrinsic demagnetization curves in the form of

$$T_0 :: (h_{ci_0},\ 0),(h_{01},\ b_{01})\cdots\cdots(h_{0i},\ b_{0i})\cdots\cdots(0,\ b_{0N_0})$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

.....

$$T_k :: (h_{ci_k},\ 0), (h_{k1},\ b_{k1}) \cdots\cdots \left(h_{ki},\ b_{ki}\right) \cdots\cdots \left(0,\ b_{kN_k}\right)$$

.....

$$T_m :: (h_{ci_m},\ 0), (h_{m1},\ b_{m1}) \cdots\cdots \left(h_{mi},\ b_{mi}\right) \cdots\cdots \left(0,\ b_{mN_m}\right)$$

where T_0, T_k, and T_m ($T_0 < T_k < T_m$) are reference temperatures at which demagnetization curves were measured. The flux density of the end point b_{kNk} is the residual b_{r_k} for curve k.

The number of (h_{ki}, b_{ki}) points, N_k +1, in each demagnetization curve can be different.

In Maxwell, if the local temperature of temperature-dependent nonlinear magnetic material is T, the local demagnetization curve can be obtained by interpolation:

If $T \le T_0$, the demagnetization curve at temperature T_0 is returned as the interpolated demagnetization curve.

If $T \ge T_m$, the demagnetization curve at temperature T_m is returned as the interpolated demagnetization curve.

If $T_k < T \le T_{k+1}$, the interpolated demagnetization curve at temperature T can be derived by following steps:

1. Get temperature interpolating coefficient: $cT = (T - T_k) / (T_{k+1} - T_k)$.
2. Get interpolated intrinsic coercivity: $h_{ci} = (1 - cT)\ h_{ci_k} + cT\ h_{ci_k+1}$.
3. Get interpolated residual: $b_r = (1 - cT)\ b_{r_k} + cT\ b_{r_k+1}$.
4. Derive demagnetization curve at temperature T using the shape preserving algorithm based on the curve at T_k, which is expressed as $(h_{ki} \bullet Q_0,\ b_{ki} \bullet P_0)$, where $Q_0 = h_{ci} / h_{ci_k}$, and $P_0 = b_r / b_{r_k}$.
5. Derive demagnetization curve at temperature T using the shape preserving algorithm based on the curve at T_{k+1}. which is expressed as $(h_{k+1}i \bullet Q_1,\ b_{k+1}i \bullet P_1)$, where $Q_1 = h_{ci} / h_{ci_k+1}$, and $P_1 = b_r / b_{r_k+1}$.
6. Derive the weighted average curve from the curve expressed by $(h_{ki} \bullet Q_0,\ b_{ki} \bullet P_0)$ and the curve expressed by $(h_{k+1\ i} \bullet Q_1,\ b_{k+1\ i} \bullet P_1)$ with weighting factors of $(1 - c_T)$ and c_T, respectively.

We can input temperature-dependent demagnetization curves in both normal and intrinsic formats. When normal demagnetization curves are input, we will transfer them to intrinsic curves and extend them with additional estimated data sets. Since the extended curve parts are approximated, it is suggested to directly input intrinsic curves with different temperatures as basic curves for interpolation, if they are available.

For example, the material Ferrite Y30 has BH curves with B_r decreasing and H_{ci} increasing as temperature increasing, as shown in **Figure 4**. If we input two intrinsic curves of the minimum

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and maximum temperatures, we can interpolate the intrinsic curves of other temperatures, and derived normal curves, as shown in **Figure 5**.

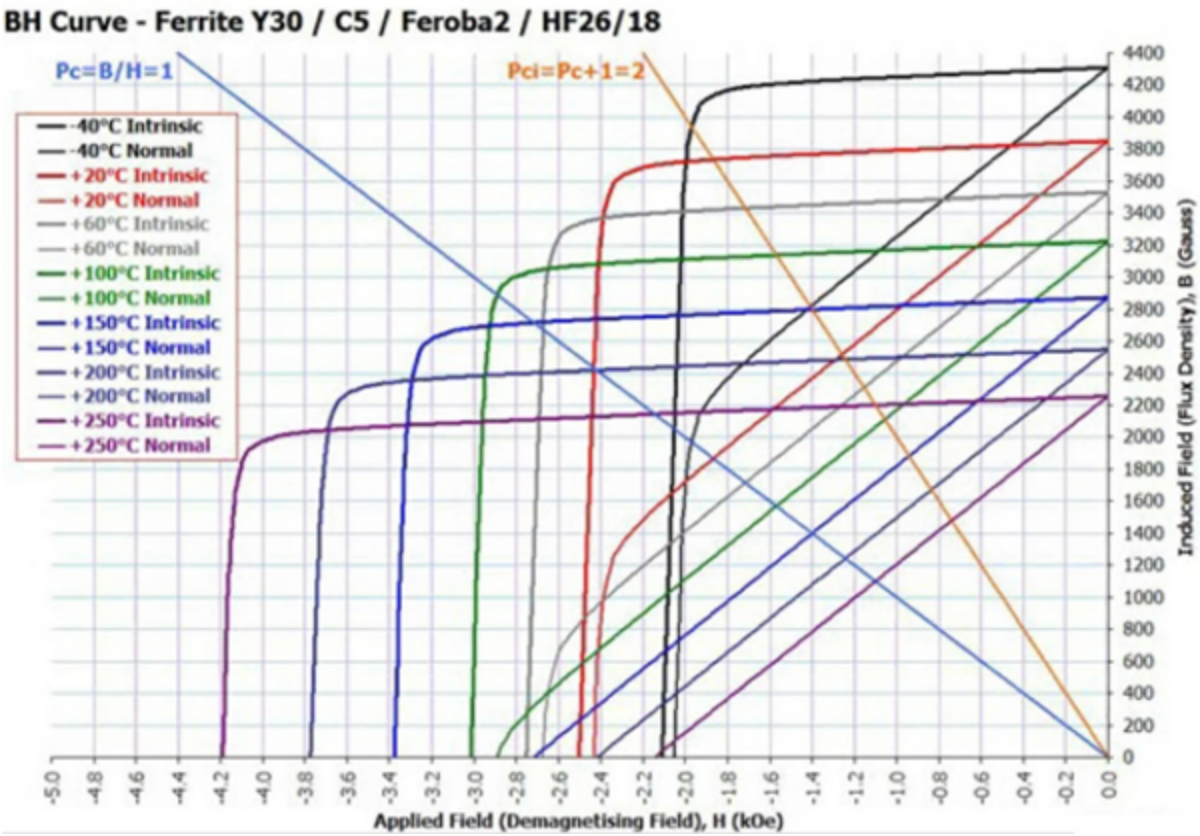


Figure 4: Temperature-dependent BH curves of Ferrite Y30

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

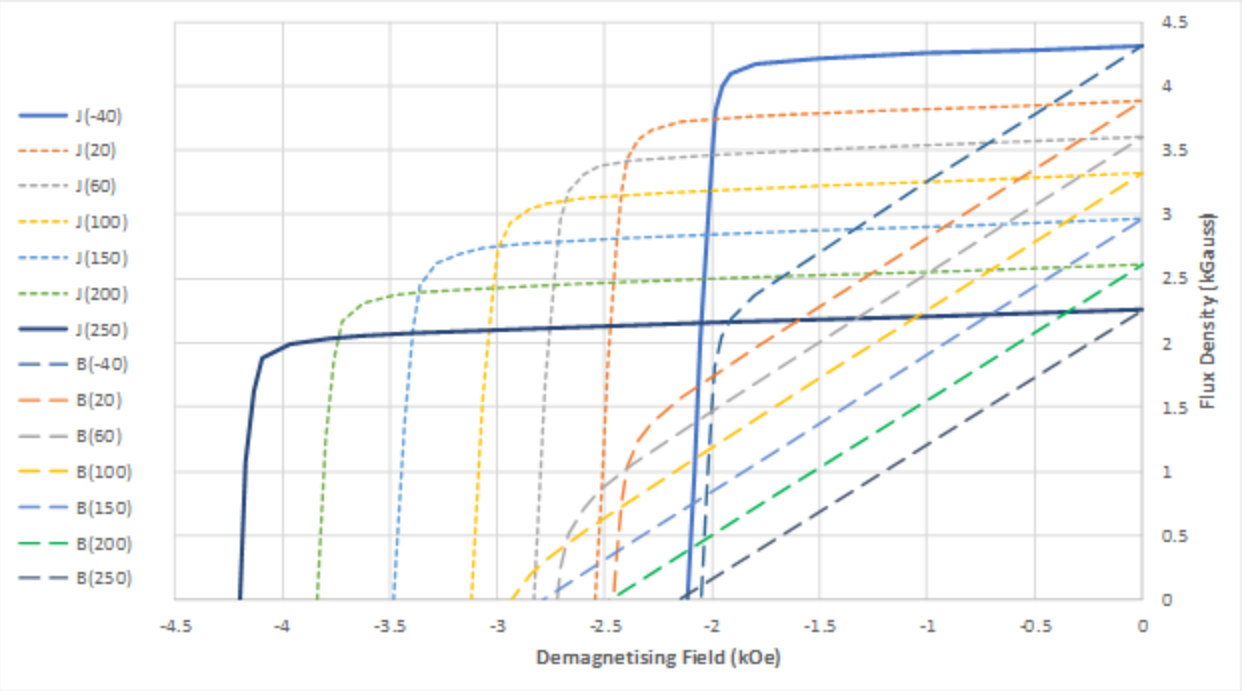


Figure 5: Input two intrinsic curves in solid lines, interpolated intrinsic curves in short-dashed lines, and derived normal curves in long-dashed lines.

If intrinsic curves are not available, we need to input three normal curves with minimum and maximum temperatures, as well as the curve with maximum Hc. In such cases, the estimated and interpolated intrinsic curves, and derived normal curves, are shown in **Figure 6**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

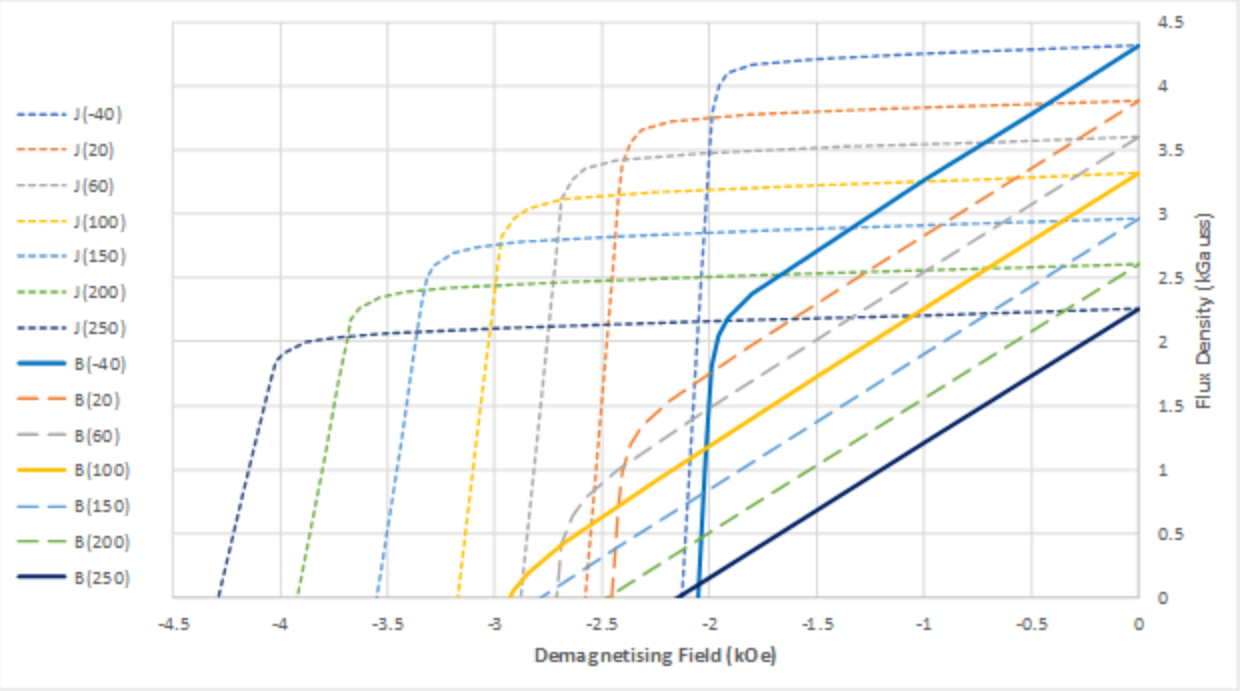


Figure 6: Input three normal curves in solid lines, estimated and interpolated intrinsic curves in short-dashed lines, and derived normal curves in long-dashed lines.

Multiple temperature-dependent normal or intrinsic demagnetization curves can be specified via BH Curve editing panel, as shown in **Figure 7**. In the drop-down menu menu of the Temperature field, select **Edit Temperature Dependency** to launch Edit Temperature Dependency dialog box, in which all temperature references for all magnetization curves can be specified. Then, each demagnetization curve related to each reference temperature can be input in the BH Curve panel.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

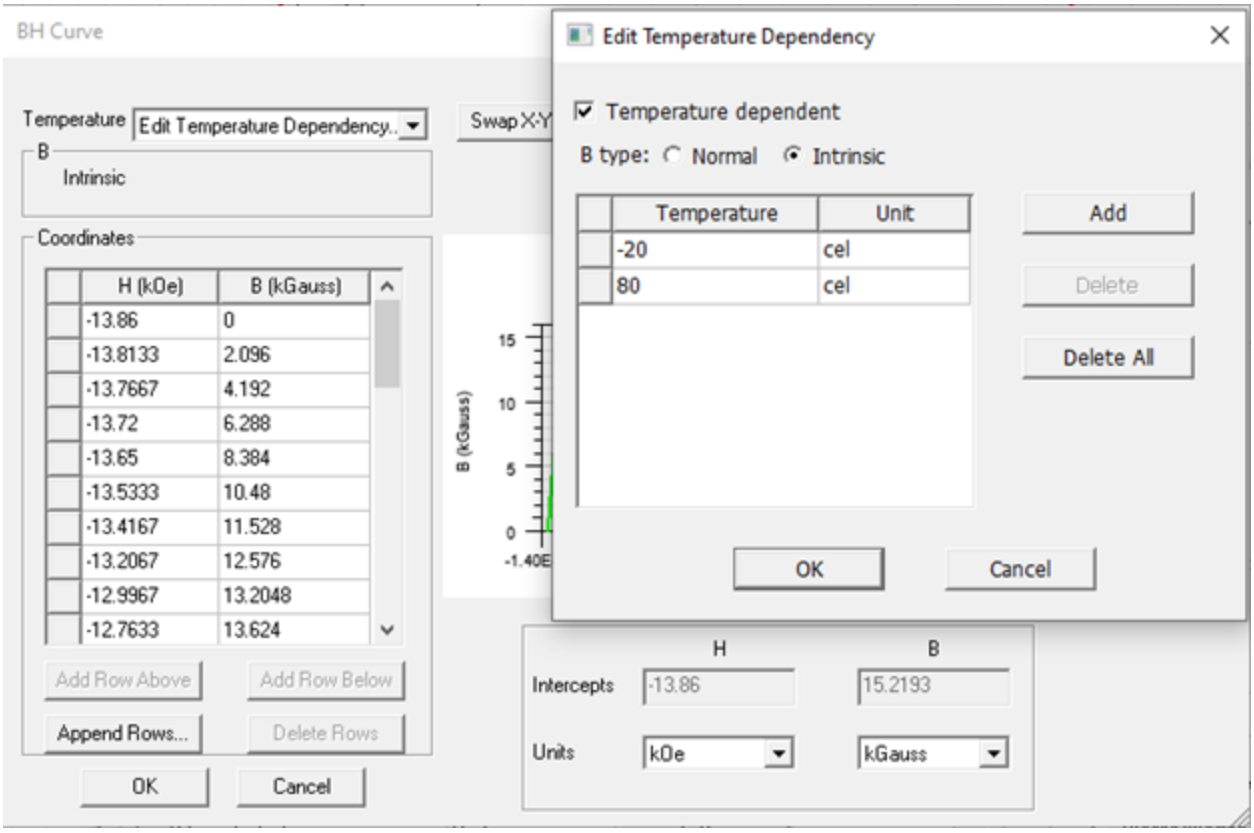


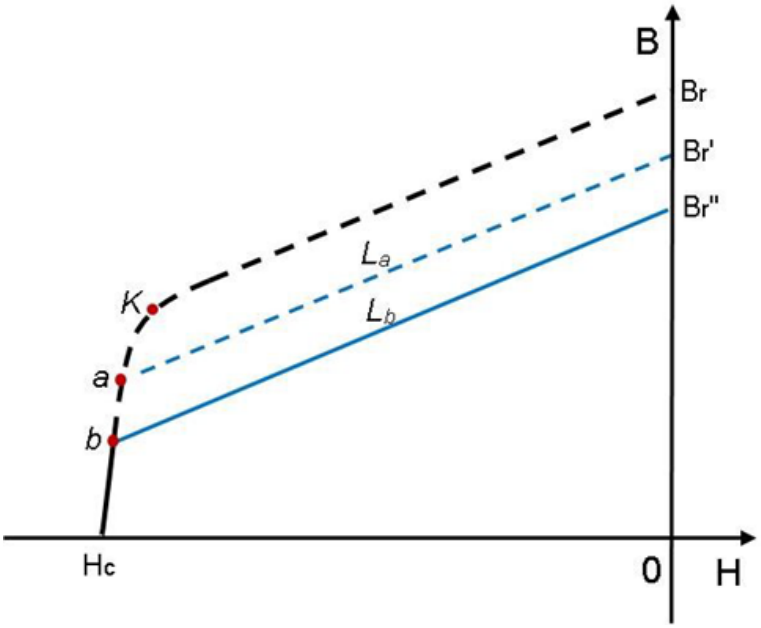
Figure 7: BH Curve panel to specify multiple temperature-dependent demagnetization curves

Irreversible Demagnetization Due to Temperature Change

Permanent magnets exposed to heat can lose strength that cannot be recovered. For example, if a permanent magnet motor is operated under a torque overload condition, the motor will draw more current to produce the required torque. The increased current causes some demagnetization, further increasing the current draw. Over a period of time, this overload may cause the motor to overheat. The resulting rise in temperature can lower the magnet material's intrinsic coercivity, thus making the magnet prone to demagnetization. If the temperature exceeds the magnet's operating point, some of the magnet's domains become permanently demagnetized. Consequently, when the magnet's temperature decreases to its initial value, the operating point does not return to its original value. This effect is irreversible.

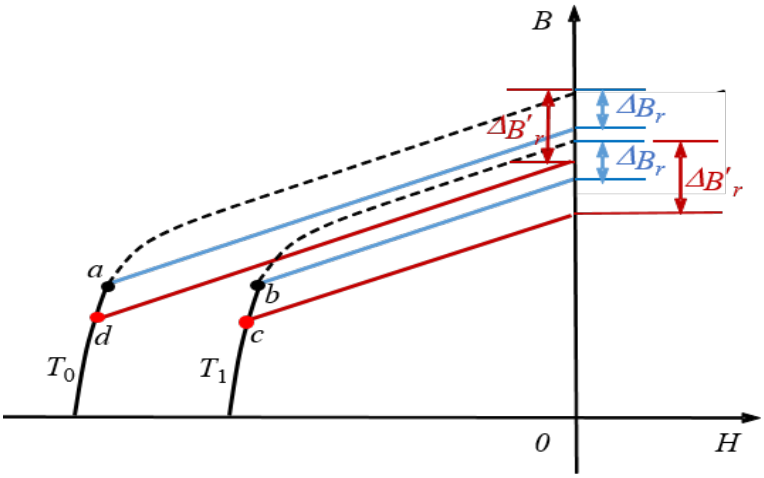
The following image shows that when a new working point ***a*** goes below the knee point ***K***, the subsequent working point will change along a new constructed B-H curve, which consists of two curve segments: the recoil line $\boldsymbol{L_a}$ and the demagnetizing curve part below point ***a***. The residual flux density of the recoil line is B_r', and the difference between B_r and B_r' is the irreversible B_r loss, which is denoted as $\Delta B_r = B_r - B_r'$.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



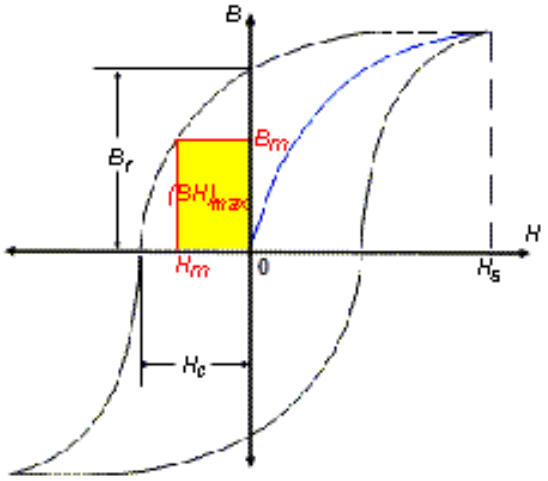
When the temperature of a permanent magnet with operating point ***a*** (as shown below) increases from T_0 to T_1, based on user-defined thermal modifiers, a new demagnetization curve can be constructed element by element at the new temperature, T_1. Because the initial irreversible B_r loss ΔBr remains unchanged, the initial operating point on the new demagnetization curve can be identified as point ***b*** (the intersection of the new demagnetization curve and the recoil line which passes ($B_r - \Delta B_r$). It follows that the worst-case operating point at temperature T_1 can be further evaluated as indicated by point ***c*** from the field solution. This new worst-case operating point, *c*, causes the irreversible B_r loss to increase from ΔB_r to $\Delta B'_r$. If the temperature then changes from T_1 back to T_0, the operating point will not return to point ***a***. Instead, it will be located at ***d*** – associated with temperature T_0 based on the new irreversible loss $\Delta B'r$. The Maxwell solution is able to store the design's irreversible demagnetization values for different working temperatures.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Characteristics and Main Parameters of Permanent-Magnetic Materials

The permanent-magnetic material belongs to hard-magnetic material. It is characterized with "fat" hysteresis loop, which encloses large area as shown in the figure. When magnetized, it keeps high magnetic property with the external magnetic field removed, therefore is used in the permanent-magnet electric machine to produce magnetic field. The characteristics of the permanent-magnetic material are represented with its main parameters: residual flux density B_r, coercive field force H_c and maximum magnetic energy product $(BH)_{max}$.

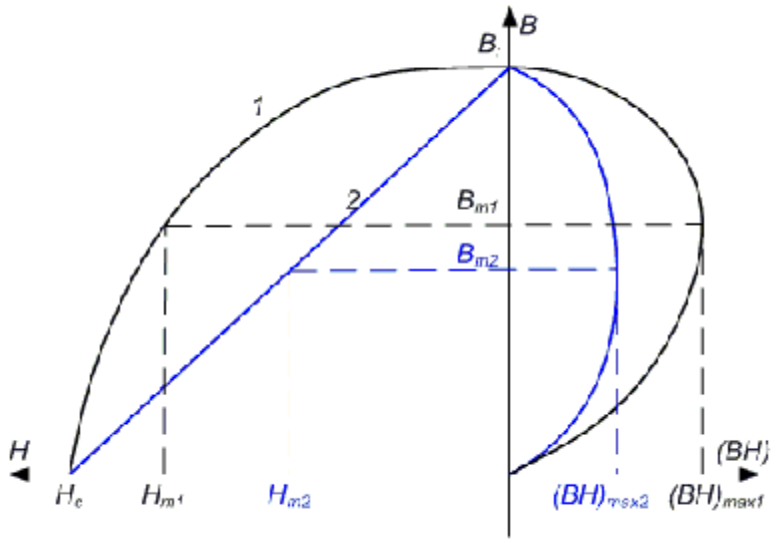


For materials with nonlinear B-H curve, the hysteresis core loss can be modeled by a hysteresis loop. In Maxwell, the hysteresis loop can be created from a normal B-H curve (starts at (0,0)) and hysteresis parameters (H_c, B_r).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Demagnetization Curve

The part of the maximum hysteresis loop of the permanent-magnetic material in the second quadrant is called the demagnetization curve as shown in the next figure. It is the basic characteristics curve of the permanent-magnetic material. On the demagnetization curve, the magnetic flux density has positive values, but the magnetic field intensity has negative values. It means that the permanent-magnetic material is applied with demagnetization magnetic field intensity. Because H_m and B_m are in opposite directions, when the magnetic flux goes through the permanent-magnetic material, the magnetic potential difference along the direction of the magnetic flux does not drop, but rises. Therefore, the permanent-magnetic material is a magnetic source, similar to the electric source in the electric circuit.



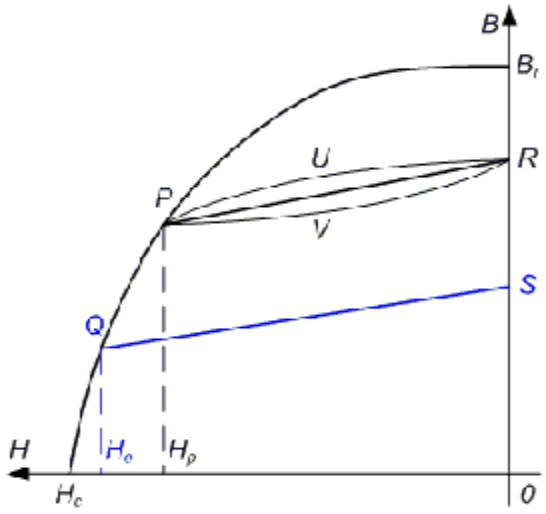
The two extreme positions on the demagnetization curve are the two significant parameters to represent the magnetic characteristics of the permanent-magnetic material. On the demagnetization curve, the value of the magnetic flux density corresponding to zero magnetic field intensity H is termed residual flux density B_r, on the other hand, the value of the magnetic field intensity corresponding to zero magnetic flux density B is termed coercive field force H_c. The produce of the magnetic flux density and the magnetic field intensity at any point on the demagnetization curve is termed magnetic energy product (BH), which is proportional to the magnetic energy density possessed by the permanent magnet at the given operating situation. At the two extreme positions ($B = B_r$, $H = 0$) and ($B = 0$, $H = H_c$), the magnetic energy product is equal to zero. Somewhere at an intermediate position, the magnetic energy reaches its maximum value and is termed maximum magnetic energy product $(BH)_{max}$, which is another significant parameter to represent the magnetic characteristics of the permanent-magnetic material. To some permanent-magnetic materials with linear demagnetization curve, it is

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

obvious that at ($B = B_r / 2$, $H = H_c / 2$), the magnetic energy product reaches its maximum value, i.e. $(BH)_{max} = B_r H_c / 4$.

Recoil Lines

The relationship between the magnetic flux density and the magnetic field intensity represented by the demagnetization curve only exists when the magnetic field intensity varies in the same direction. In fact, when the permanent magnet electric machine is working, the demagnetization field intensity varies repeatedly in both directions. When demagnetization field is applied to the magnetized permanent magnet, the magnetic flux density decreases along the curved segment B_rP on the demagnetization curve as shown in the figure:



If the external demagnetization field intensity H_p is removed when the magnetic flux density reaches the point *P*, the magnetic flux density will increase not along the original demagnetization curve, but along another curve *PVR*. If the external demagnetization field intensity is reapplied, the magnetic flux density will decrease along the new curve *RUP*. By repeatedly applying the demagnetization field intensity, a localized loop is formed and is termed local hysteresis loop. On the local hysteresis loop, the rising segment and the dropping segment are quite close to each other, therefore can be approximated by the straight line *PR*, which is termed recoil line with *P* as the starting point. If demagnetization field with intensity H_q not exceeding the original value H_p is applied thereafter, the magnetic flux density will vary reversibly along the recoil line *PR*. If $H_q > H_p$, the magnetic flux density drops to a new starting point *Q*. It will vary along the new recoil line *QS*, but not the previous one *PR*. This sort of irreversible variation in magnetic flux density causes instability in the characteristics of electric machines and complicates the design computation of permanent magnet electric machines, therefore should be avoided as possible.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Recoil Magnetic Permeability

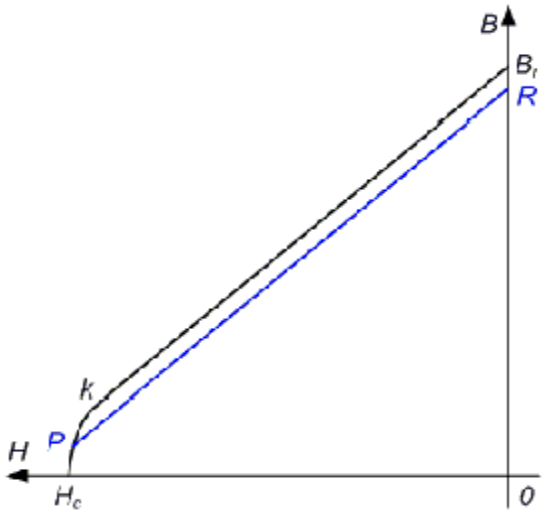
The ratio of the average slope of the recoil line to the magnetic permeability in vacuum μ_0 (μ_0= 4 x 10^{-7} H/m) is termed relative recoil magnetic permeability or recoil magnetic permeability for short, μ_r:

$$\mu_r = \frac{1}{\mu_0} \left| \frac{\Delta B}{\Delta H} \right|$$

If the demagnetization curve is curly, the value of μ_r depends on the location of the starting point and is a variable, but typically varies within a small range. Therefore, it is approximated as a constant and is taken as the slop of the tangent to the point (B_r, 0) on the demagnetization curve. In other words, the recoil lines at different starting points are approximated as a family of parallel lines, which are all parallel to the tangent to the point (B_r, 0) on the demagnetization curve.

Inflection Point

Some permanent-magnetic materials, such as some ferrite permanent-magnetic materials, show straight upper segment on the demagnetization curve. When the demagnetization field intensity drops to a given value, the demagnetization curve turns to decrease rapidly. The turning point is termed inflection point. If the demagnetization field intensity does not exceed the inflection point *k*, the recoil line coincides with the straight segment on the demagnetization curve. If the demagnetization field intensity exceeds the inflection point *k*, the new recoil line *RP* no longer coincides with the straight segment on the demagnetization curve.



Some permanent-magnetic materials, such as most of the rare-earth permanent-magnetic materials, show straight demagnetization curve in the whole range. In those cases, the recoil line coincides with the demagnetization curve. This makes the magnetic property keep stable while

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the permanent-magnet electric machine is working. This is the best ideal demagnetization curve for electric machine applications.

Calculating the Properties for a Permanent Magnet

From the **View/Edit Materials** window:

1. Set the **Relative Permeability** to **Simple**.

   This enables the **Calculate Properties for...** drop down menu at the bottom of the window.

2. Click **Permanent Magnet** from the drop-down menu.

   This displays the **Properties for Permanent Magnet** dialog box. This contains the following fields.

Mu (enabled by default)	Provide a value for relative permeability.
Hc (enabled by default)	Coercive field force H_c in the units specified. Provide a value and select units from the drop-down menu.
Br/Mp (disabled by default)	Checking this enables the radio buttons that let you specify either Br or Mp.
Br	Residual flux density B_r, in Tesla. If enabled, provide a value and select units from the drop-down menu.
Mp	Permanent Magnetization M_p, in A/m. If enabled, provide a value and select units from the drop-down menu.

3. Click **OK** to close the dialog and return to the **View/Edit Materials** window.

The values for **Relative Permeability** and **Magnitude** under **Magnetic Coercivity** are updated as new default values.

Calculating the Properties for a Nonlinear Permanent Magnet

Nonlinear permanent magnet properties may be specified in one of two ways:

- Input a B-H Curve directly
- Using the Properties for Nonlinear Permanent Magnet window

Input a B-H curve directly as follows:

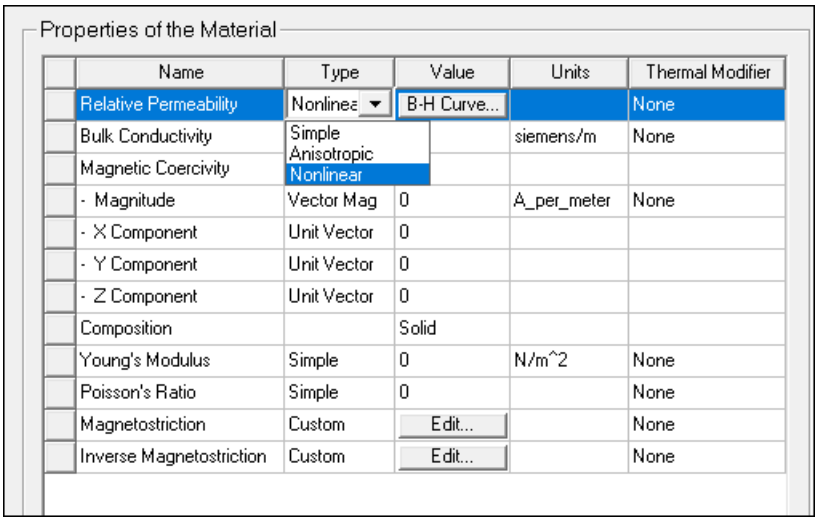
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: You can model multiple temperature dependencies of a nonlinear material property by providing two or more temperature-dependent B-H curves. Refer to Specifying B-H Curves for Nonlinear Relative Permeability for detailed information.

1. Click the **View/Edit Materials…** button in the **Edit Libraries** window.

   The **View/Edit Material** window appears.

2. To define the nonlinear B-H curve, set the **Relative Permeability** **Type** to **Nonlinear:**

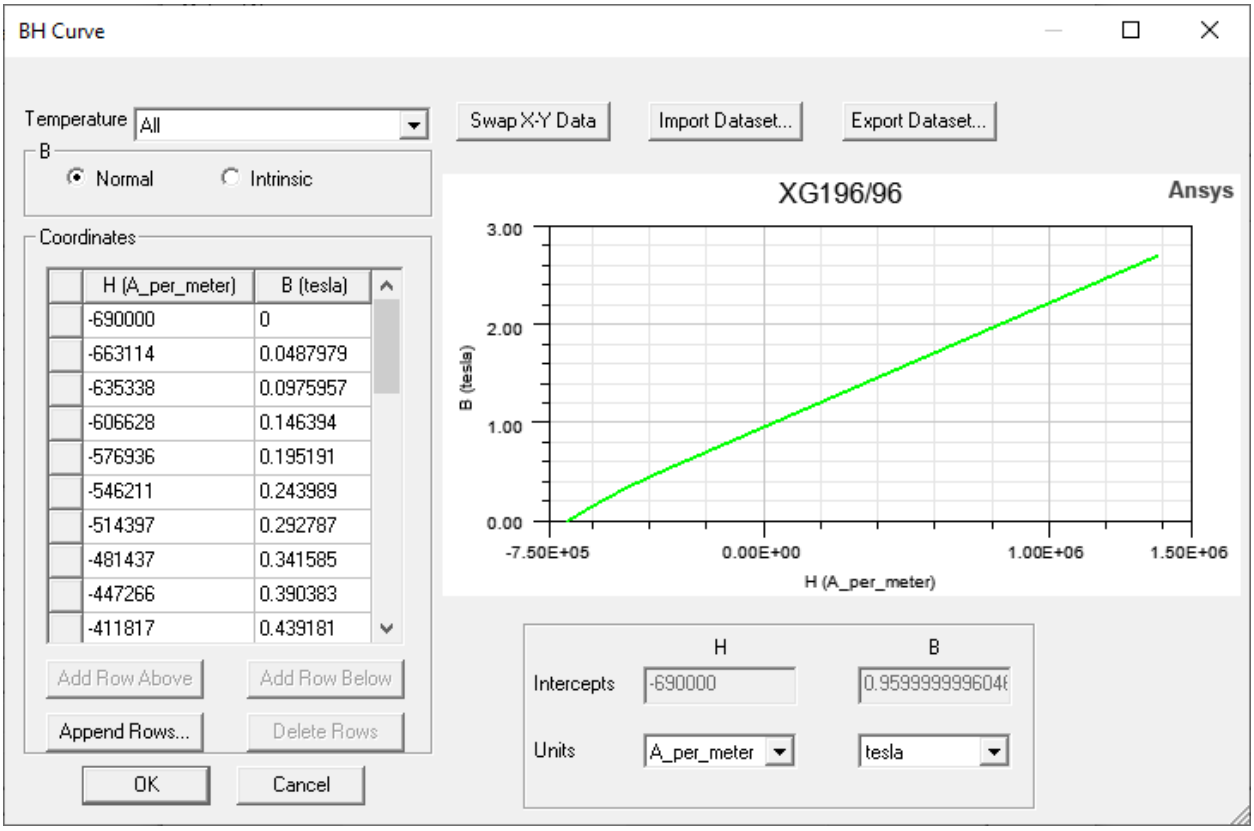


   A **B-H Curve** button appears in the nonlinear property's **Value** column.

3. Input the B-H curve by clicking the **B-H Curve** button in the property **Value** column.

   This opens the **BH Curve** dialog box in which you can input (or modify) curve data. (Refer to *Adding Datasets* for general information on working with datasets.)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- For a **Normal** B-H curve, the slope of the curve can not be less than that of free space anywhere along the curve.
- For an **Intrinsic** B-H curve, the slope of the curve can not be less than 0.

Note:

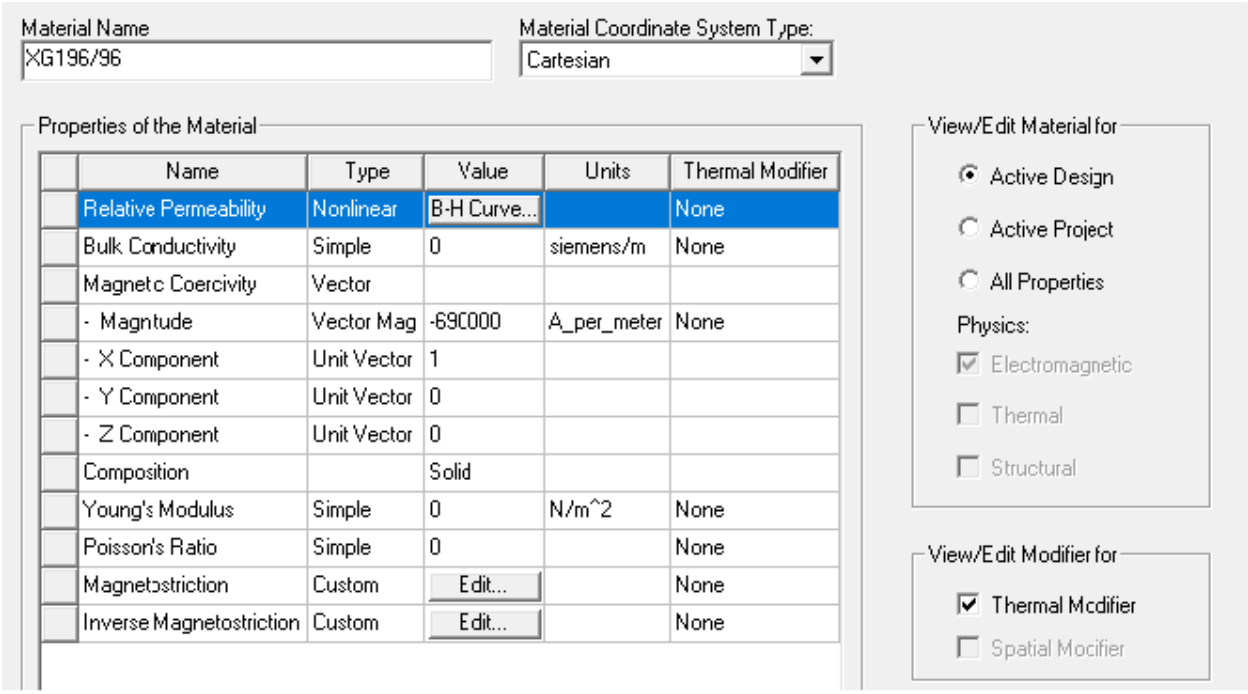
- The **Intrinsic** B-H curve is supported only in Maxwell 2D/3D magnetostatic and transient design types. A material property defined using an Intrinsic BH curve will fail validation check in all the other product/design types.
- When an **Intrinsic** B-H curve is added, the **Relative Permeability Value** button label changes to **Bi-H Curve** as visual indication of the type of curve currently defined for the material.

When you click **OK** in the dialog box, an error message displays if a slope is out of tolerance, identifying the data points between which the slope is less than that of free space.

The operations to input a nonlinear demagnetization curve are the same as entering a B-H curve for Steel material. When a B-H curve goes through the second quadrant or third quadrant, the curve is treated as a demagnetization curve.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. To model a single *temperature dependency* for a nonlinear permanent magnet you must
   - Use an **Intrinsic** B-H curve to model the **Relative Permeability**
   - Specify a thermal modifier for both **Relative Permeability** and the **Magnitude** of **Magnetic Coercivity**. Apply a thermal modifier by selecting the **Thermal Modifier** check box. Checking this box causes the **Thermal Modifier** column to display at the right side of the **Properties of the Material** table. Selecting **Edit** rather than **None** causes display of the **Edit Thermal Modifier** dialog box.



> **Note:** You can model multiple temperature dependencies of a nonlinear material property by providing two or more temperature-dependent BH curves. Refer to *Specifying BH Curves for Nonlinear Relative Permeability* for detailed information.

Creating a Nonlinear B-H Curve with Four Parameters

Alternatively, a nonlinear B-H curve can be modeled by the following four parameters:

- residual flux density B_r
- coercive field force H_c
- maximum energy product $(BH)_{max}$
- relative recoil permeability μ_r

From the **View/Edit Materials** window:

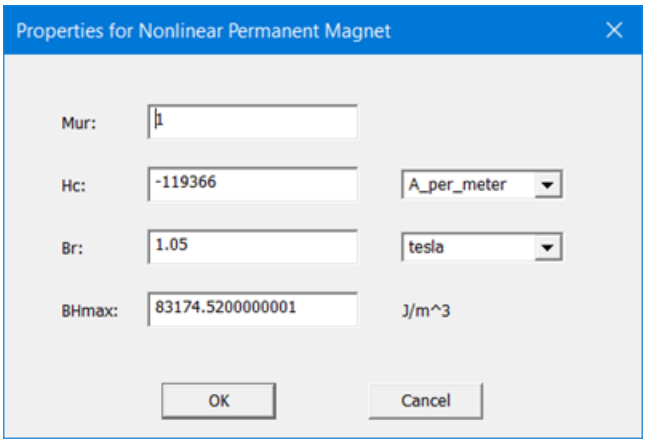
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Set the **Relative Permeability** to **Nonlinear**.

   This enables the **Calculate Properties for...** drop-down menu at the bottom of the window.

2. Select **Nonlinear Permanent Magnet** from the drop-down menu.

   This displays the **Properties for Nonlinear Permanent Magnet** dialog box, which contains the following fields into which you enter the appropriate values.



Mur	Provide a value for relative permeability.
Hc	Coercive field force H_c in the units specified. Provide a value and select units from the drop-down menu.
Br	Residual flux density B_r in Tesla. If enabled, provide a value and select units from the drop-down menu.
BH max	Maximum magnetic energy product $(BH)_{max}$ If enabled, provide a value and select units form the drop-down menu.

3. Click **OK** to close the dialog box and return to the **View/Edit Materials** window.

The values for **Relative Permeability** and **Magnitude** under **Magnetic Coercivity** are updated as new default values, and the existing B-H curve is updated based on the new values of these parameters.

Calculating Properties Using a Hysteresis Loop in Maxwell

For a material whose Relative Permeability is Nonlinear (refer to Calculating the Properties for a Nonlinear Permanent Magnet), and whose **Magnitude** of **Magnetic Coercivity** is zero, you can calculate core loss properties using a hysteresis loop as follows:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Tools** > **Edit Libraries** > **Materials** to open the **Edit Libraries** window.

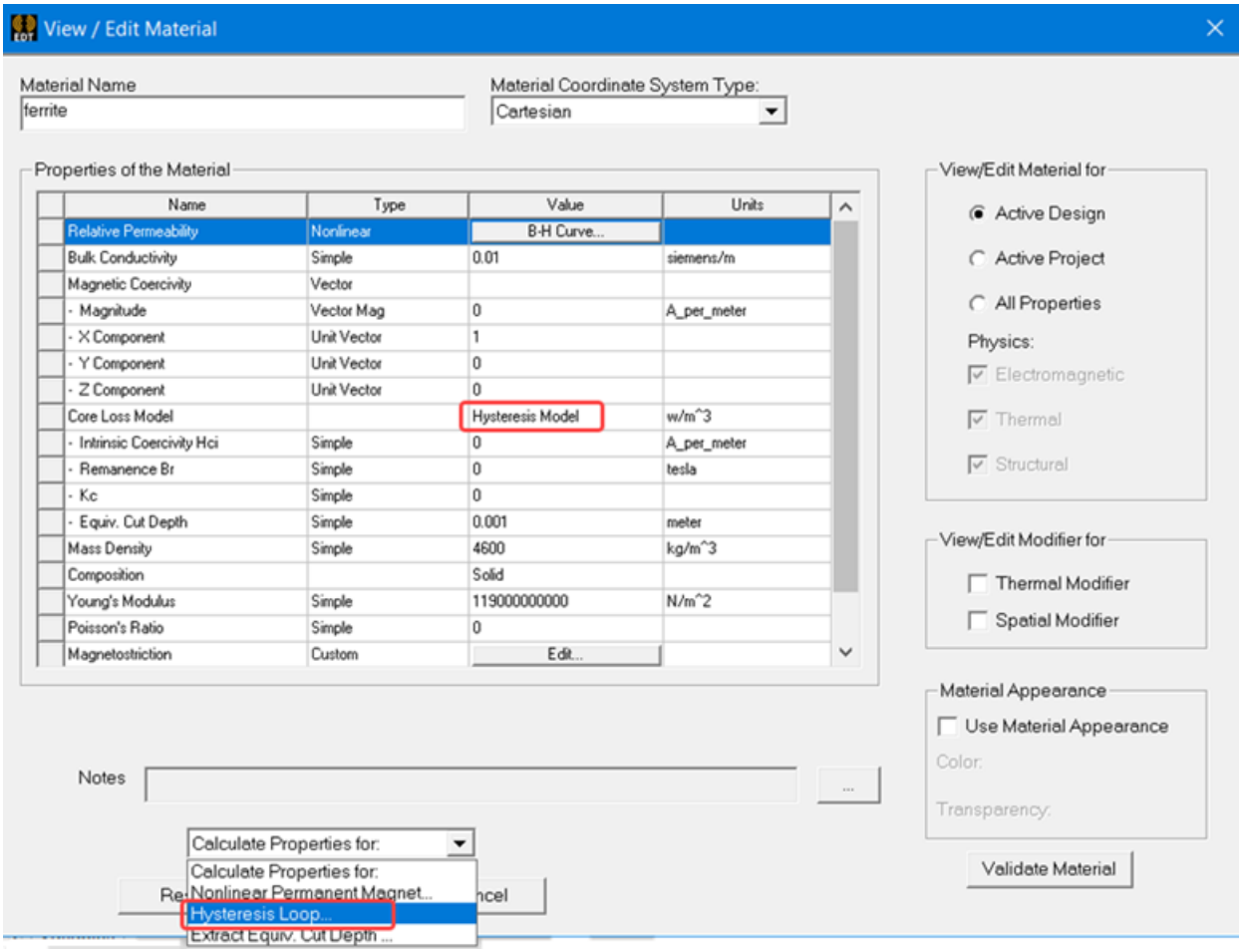
   Alternatively, in the Project Manager tree, you can right-click **Materials**, under **Definitions**, and select **Edit All Libraries**.

2. Click **Add Material** to open the **View/ Edit Material** window.
3. In the **Core Loss Model** row, select **Hysteresis Model** from the **Value** drop-down menu.

   If you select either **View/Edit Material for: This Product** or **All Products** radio button, additional parameters appear in the table below **Core Loss Model** (**Intrinsic Coercivity Hci**, **Remanence Br**, **Kc**, and **Equiv. Cut Depth**). You can input these values directly. You can also supply the **Hci** and **Br** values in the **Properties for Hysteresis Loop** dialog box as described below.

   The **Hysteresis Loop** selection in the **Calculate Properties for** drop-down menu at the bottom of the window is also enabled. This allows the coefficients **Hci**, and **Br** to be derived from a manufacturer-provided core loss curve.

4. Select **Hysteresis Loop** from the **Calculate Properties for** drop-down menu at the bottom of the window to open the **Properties for Hysteresis Loop** dialog box.



The current BH Curve coordinates are loaded by default. **Hci** and **Br** values from the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

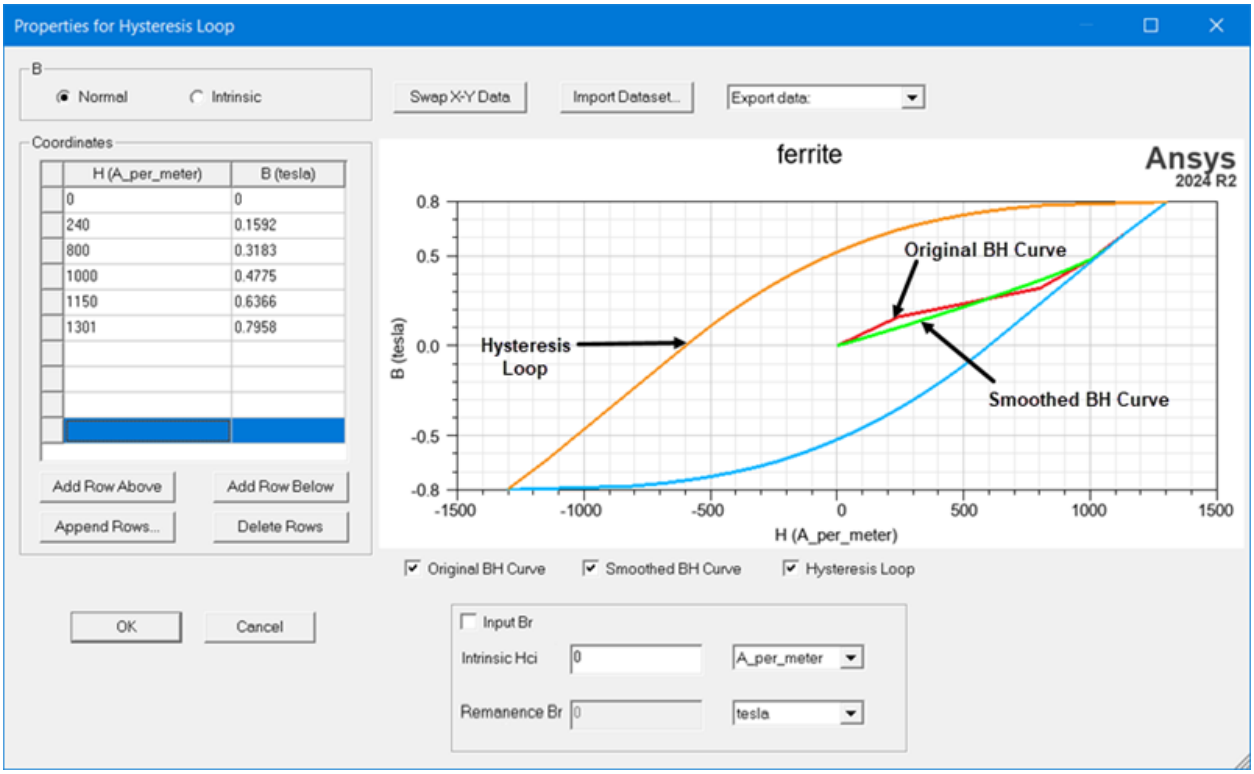
Intrinsic Coercivity Hci, **Remanence Br** fields in the **View / Edit Material** window are also displayed.

5. Do one of the following to specify the hysteresis loop curve:
   - Enter the **B** and **H** values in each row of the **Coordinates** table manually. The graph updates as you enter values. Buttons allow you **Add**, **Append**, and **Delete** table rows as needed.

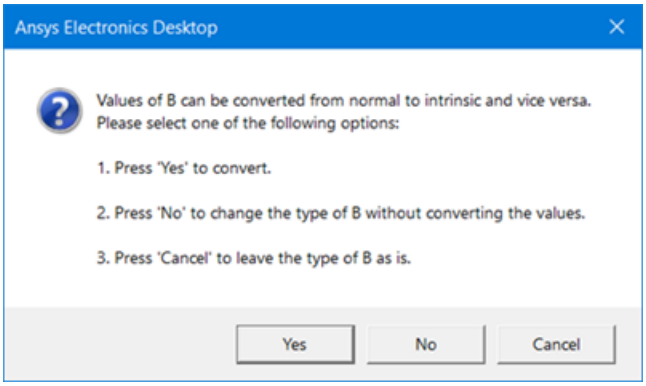
     Note:
     - The value of B must increase along the curve.
     - The accuracy of input data for the curve has significant effect on the correctness of the analyses of the electromagnetic devices. You should ensure curve data provided by material manufacturers is accurate.
     - You can also import the hysteresis loop coordinates from a saved file.

   - Click **Import Dataset** to open the **Import Dataset** dialog box. Find and select either the BH Data.bh file, or the tab-delimited .tab file containing the curve data points. Click **Open** to import the data.
   - Optionally, click **Swap X-Y Data** to switch the B and H values if they are in the wrong columns.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. Optionally, enter the Hci and Remanence Br values and choose appropriate units for them. **Input Br** must be enabled to enable the Br entry box; otherwise, the default zero value indicates that **Br** will be determined automatically by the software. (These fields may already have values in them if previously supplied in the **View / Edit Material** window.) Disabling **Input Br** resets the Br value to zero.
7. Optionally, select either **Normal** (the default) or **Intrinsic** for BH curve input and plot display. When changing from **Normal** to **Intrinsic** (or vice-versa) the following dialog box displays, allowing you to choose whether or not to convert B values.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Optionally, you can export data for the descending and ascending curve of the hysteresis loop. You can also export the BH curve as well as the smoothed BH curve.
9. Click **OK** to close the Properties for Hysteresis Loop dialog box.

   The values of Magnetic Coercivity Magnitude, **Hci**, and **Br** are updated in the **View/Edit Material** window.

   > **Note:** In the **View/Edit Material** dialog box, if the **Magnetic Coercivity Magnitude** is changed to a non-zero value by modifying the **Relative Permeability B-H Curve** property, the **Core Loss Model** property is reset from **Hysteresis Model** to **None**.

10. Click **OK** to close the **View / Edit Material** window.
11. Click **OK** to close the **Edit Libraries** window.

Magnetostriction and Inverse Magnetostriction

Magnetostriction is a property of ferromagnetic materials that causes their deformation during the process of magnetization. The structure of such materials is divided into *domains*, each of which is a region of uniform magnetic polarization. When a magnetic field is applied, the domains rotate causing a change in the material's dimensions. This change in the material's dimensions is due to the anisotropic crystalline structure of the material – it takes more energy to magnetize a crystalline material in one direction than another. For example, the steel used for the thin laminations in transformers is manufactured by passing it through a series of rollers to achieve the desired thickness. Consequently its crystalline structure – and associated magnetic domains – differs along the direction in which it was rolled (the *rolling* direction) versus the *transverse* direction.

Magnetostriction – and Inverse Magnetostriction – can be regarded as an energy transduction (or transformation) from magnetic to mechanical and vice-versa. It can also be described as a bidirectional magneto-mechanical coupling between the mechanical and magnetic fields in the magnetostrictive material. It is magnetostriction that causes the low-pitched humming sound that can be heard coming from transformers – caused by the changing magnetic field produced by oscillating AC currents. For detailed information on these properties, refer to Magnetostriction Modeling of Magnetic Materials in the technical notes section.

- Magnetostriction/inverse magnetostriction properties can be applied to Maxwell 2D/3D magnetostatic and transient solutions through customizable Magnetostriction and Inverse Magnetostriction material properties.
- Maxwell Desktop supplies the magnetostriction and inverse magnetostriction material properties to the Maxwell solver. The solver then calculates force density with magnetostriction force components included.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For Maxwell magnetostatic solutions, if any material used has the Inverse Magnetostriction property defined, Maxwell Desktop saves the initial H field, which is generated by the solver without exterior stress/strain applied, in the project results.
- Magnetostriction and inverse magnetostriction are applicable to Maxwell 2D/3D Magnetostatic design one-way and two-way coupling with Ansys Mechanical system.

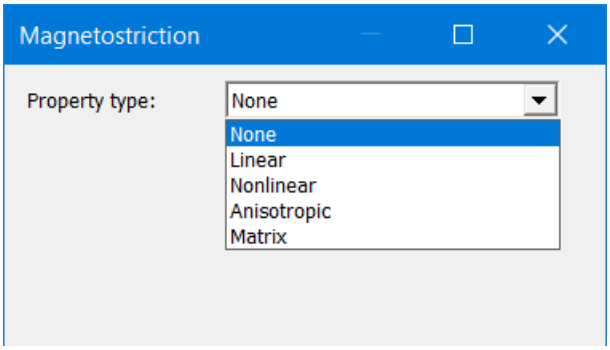
> **Note:**
>
> For inverse magnetostriction:
>
> - Materials having cylindrical and spherical coordinate systems are not supported.
> - In Maxwell 3D, the inverse magnetostriction effect on a BH curve only responds to normal stress components; shear components are ignored.

Calculating the Properties for Magnetostriction

The magnetostriction property supports four models: Linear, Nonlinear, Anisotropic, and Matrix. The property values you enter typically come from manufacturer data sheets.

To define these model properties, from the **View/Edit Material** window:

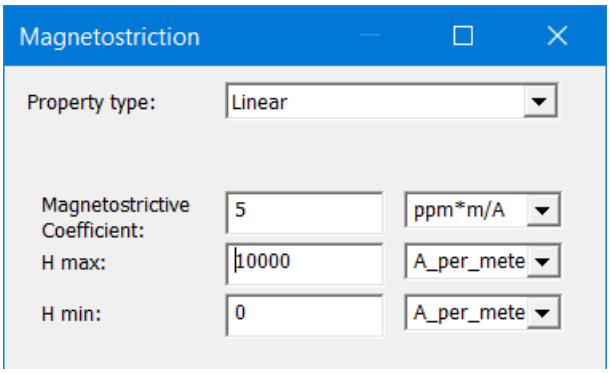
1. Click the **Edit** button to the right of **Magnetostriction Custom** to open the Magnetostriction properties dialog box:



2. Choose a **Property type** from the drop-down menu: **Linear**, **Nonlinear**, **Anisotropic**, or **Matrix**, and do one of the following:

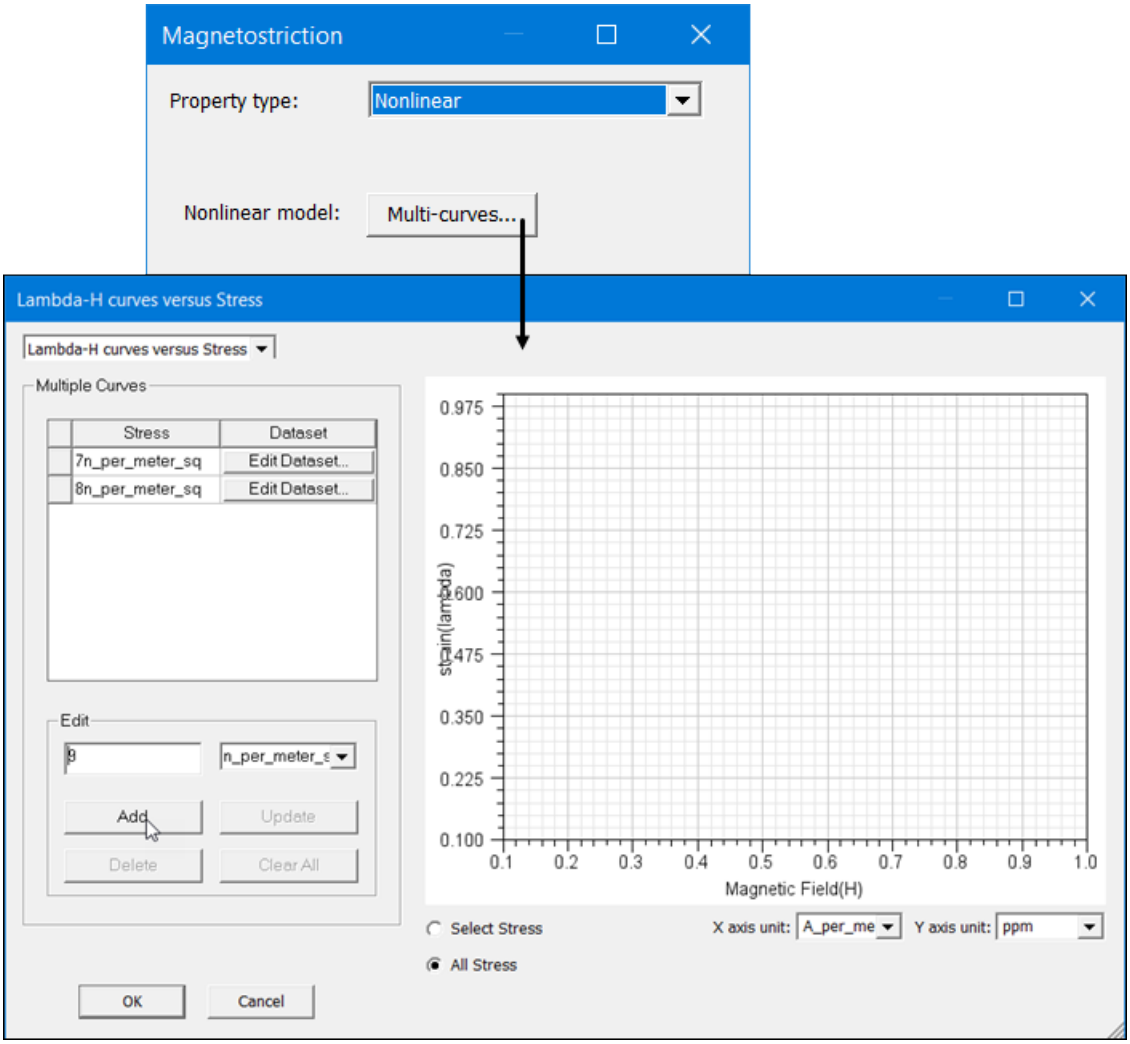
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. For **Linear**, enter values for **Magnetostrictive Coefficient**, **H max**, and **H min**.



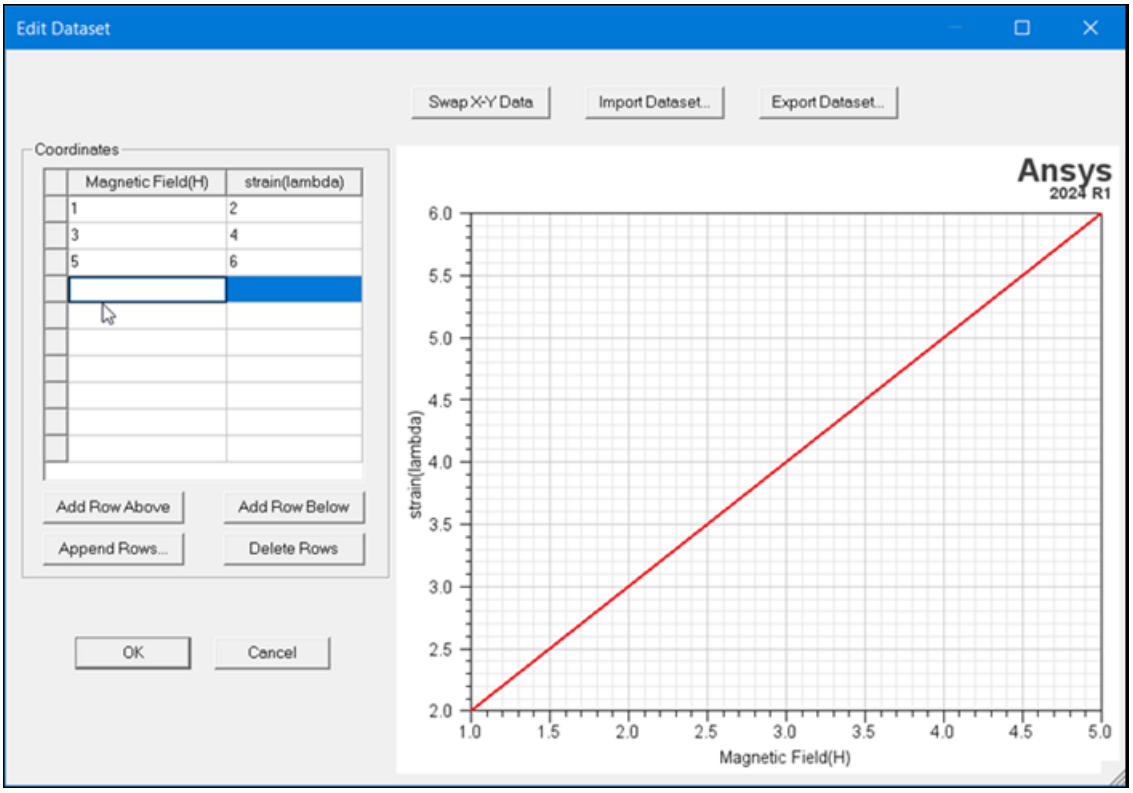
b. For **Nonlinear**, click the **Multi-curves** button to add one or more stress-dependent Lambda-H curves.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Enter a stress value in the **Edit** text box and click **Add**, then click **Edit Dataset** to open the **Edit Dataset** dialog box, in which you enter coordinate values for the **Magnetic Field (H)** and **strain (Lambda)**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

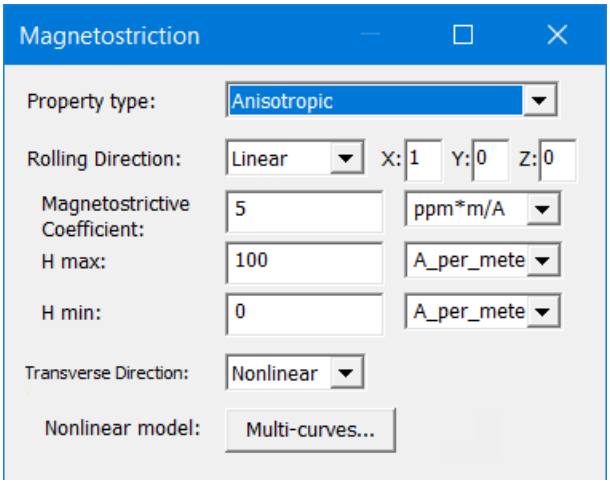


When finished, click **OK** to close the dialogs and return to the **Magnetostriction** dialog box.

c. For **Anisotropic**, select either **Linear** or **Nonlinear** from the **Rolling Direction** and **Transverse Direction** drop-down menus. Rolling Direction refers to the alignment of the crystalline structure (and associated magnetic domains) in the direction along which the material (such as the steel laminations used in transformers) passed through the rollers during manufacture. Transverse Direction is at right angle to the Rolling Direction. Magnetostriction property values typically differ in each direction.

> **Note:**
>
> The type you select for each of the directions need not be the same. For example, you could select Linear for Rolling Direction and Nonlinear for the Transverse Direction, and vice versa.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

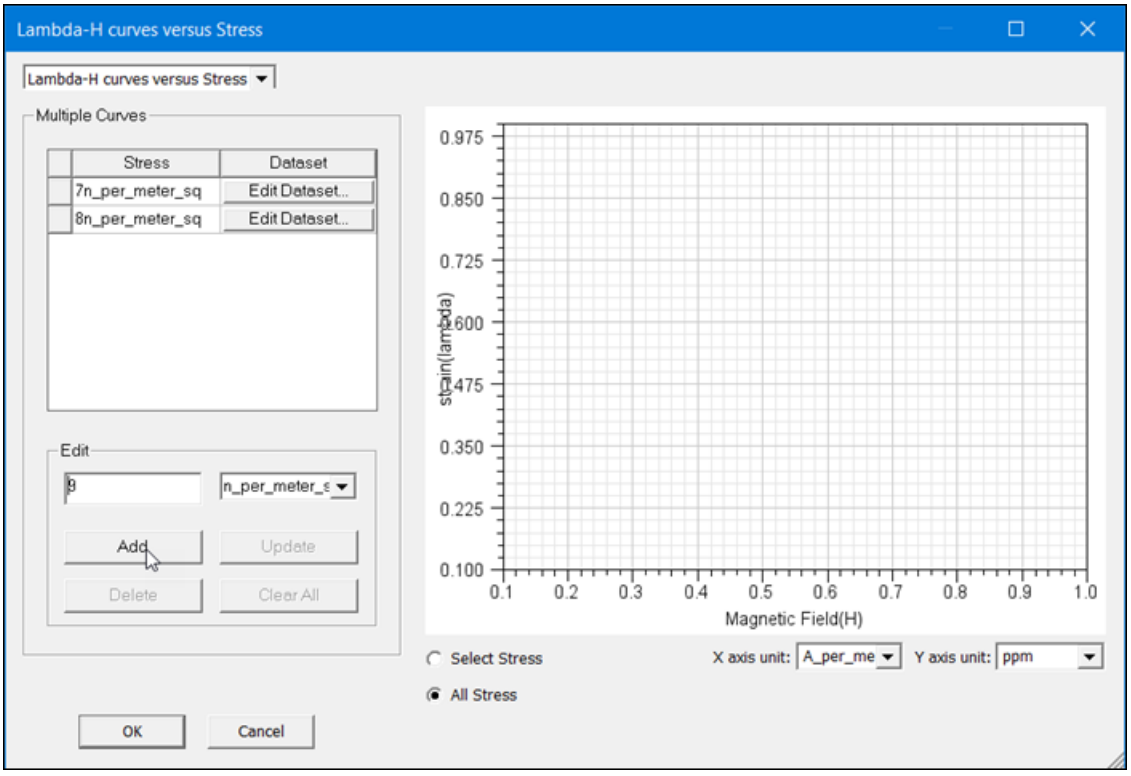


- For **Linear Rolling Direction** only, enter the **X**, **Y**, and **Z** coordinate values. For **Linear Transverse Direction**, enter values for **Magnetostrictive Coefficient**, **H max**, and **H min**.

  > **Note:**
  >
  > The X, Y, and Z components of the Rolling Direction correspond to the **Material Coordinate System Type** set in the **View/Edit Material** window.
  >
  > - For **Cartesian**: X=X, Y=Y, Z=Z
  > - For **Cylindrical**: X=R, Y=Phi, Z=Z
  > - For **Spherical**: X=Rho, Y=Theta, Z=Phi

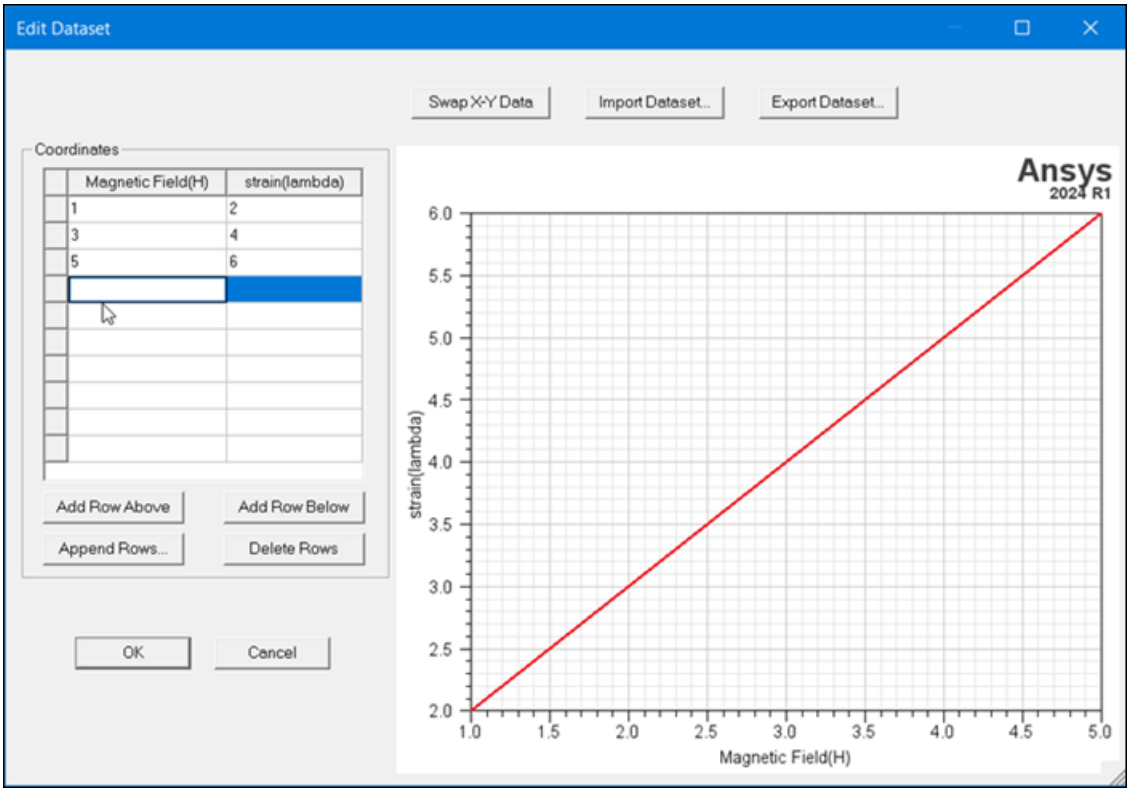
- For **Nonlinear**, click the **Multi-curves** button to add one or more stress-dependent Lambda-H Curves.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Enter a stress value in the **Edit** text box and click **Add**, then click **Edit Dataset** to open the **Edit Dataset** dialog box, in which you enter coordinate values for the **Magnetic Field (H)** and **strain (Lambda)**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



d. The **Matrix** type can be used to simulate magnetostrictive stress and deformation with full magnetostriction matrix [3x3]; with this type, a simulation can consider both isotropic and anisotropic magnetostriction effects. This property type is applicable to Maxwell 2D and 3D magnetostatic and transient designs.

   The magnetostriction deformations and magnetic fields are linked via the following expression:

$$\lambda_i = C_{ij} H_j, (i, j = 1,\ 2,\ or\ 3)$$

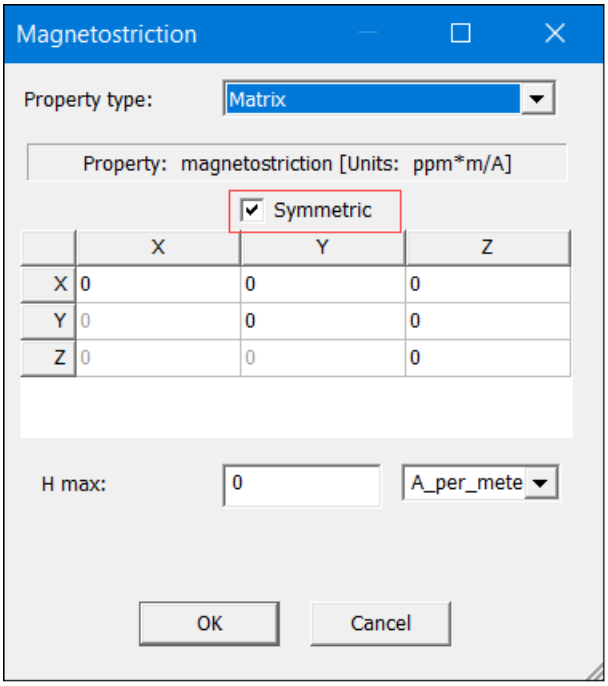
   where λ is LambdaVector,

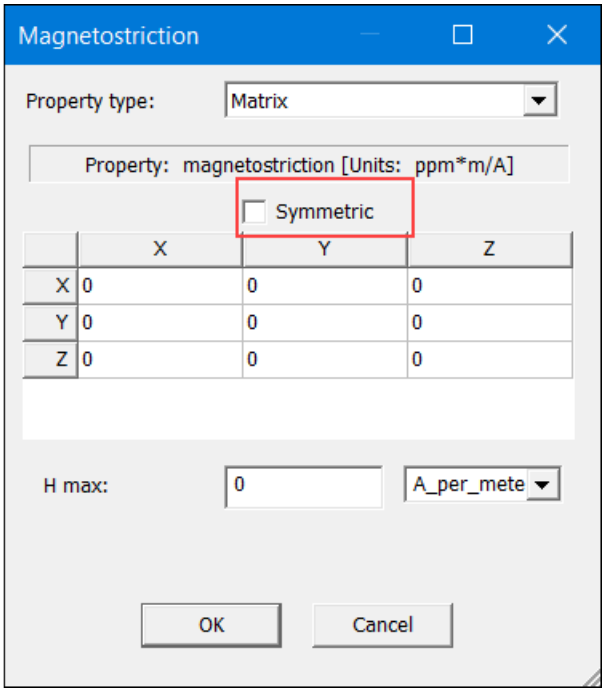
   C is the CoefficientMatrix, and

   H is HFieldVector.

   H max (magnitude of H vector) setting seen below the matrix is also needed as saturation control.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





> **Note:**
>
> If you want to use a variable to define any of the X, Y, and Z fields, the variable name must begin with $ because material properties only support project-level variables. For example, **$a_value** would be a valid name, but **a_value** would not.

3. When finished, click **OK** to close the dialog boxes and return to the **Magnetostriction** window.
4. Click **OK** to close the **Magnetostriction** dialog box and return to the **View/Edit Materials** window.

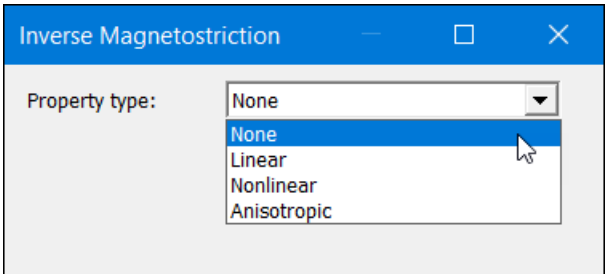
Calculating the Properties for Inverse Magnetostriction

The inverse magnetostriction property supports three models: Linear, Nonlinear, and Anisotropic. The property values you enter typically come from manufacturer data sheets.

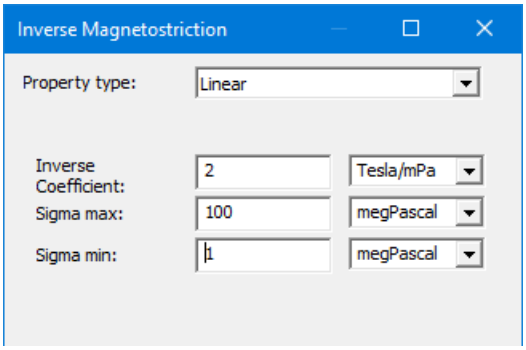
To define these model properties, from the **View/Edit Material** window:

1. Click the **Edit** button to the right of **Inverse Magnetostriction Custom** to open the Inverse Magnetostriction properties dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

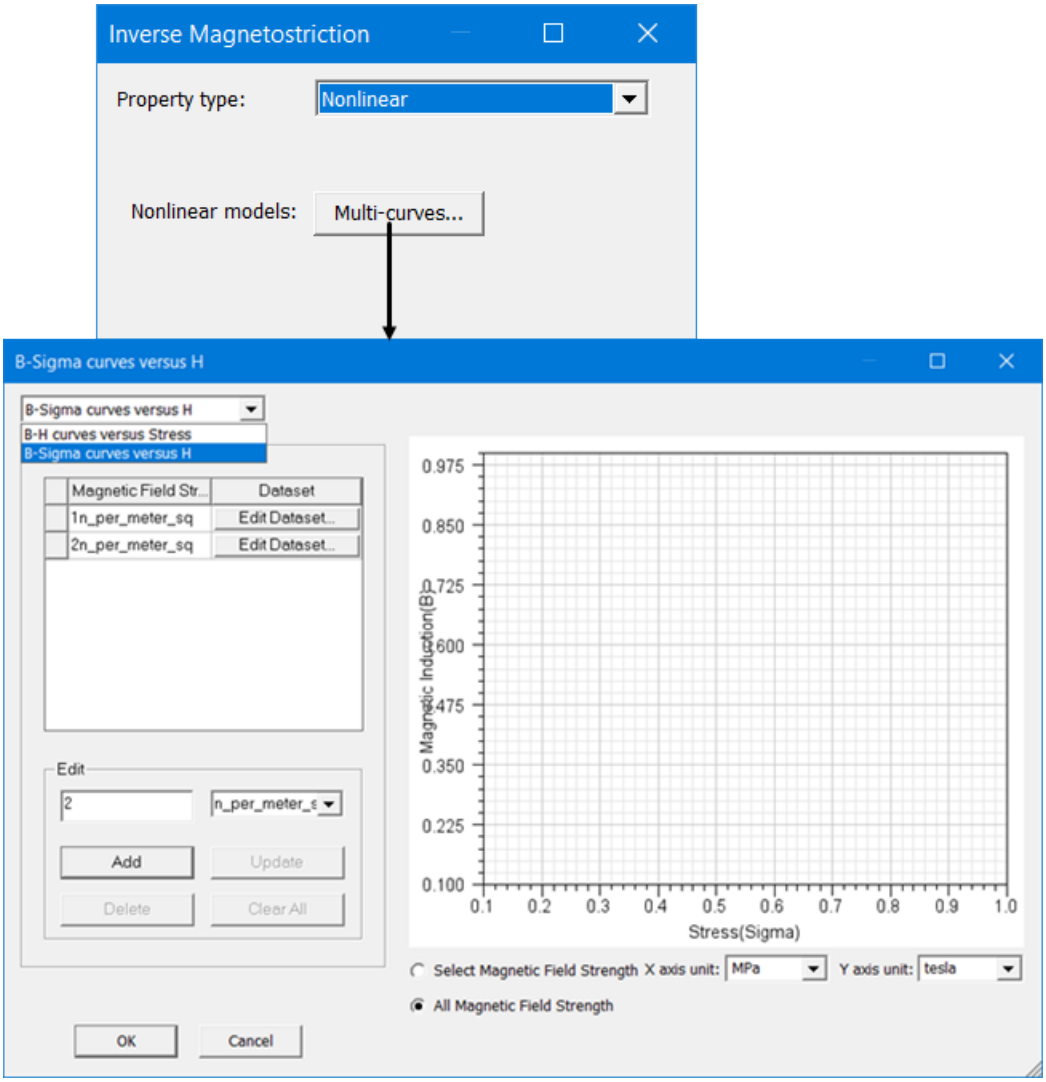


2. Choose a **Property type** from the drop-down menu: **Linear**, **Nonlinear**, or **Anisotropic**, and do one of the following:
    a. For **Linear**, enter values for **Inverse Coefficient**, **Sigma max**, and **Sigma min**, where sigma represents mechanical stress.



    b. For **Nonlinear**, click the **Multi-curves** button to add one or more stress-dependent B-H curves, or H-dependent B-Sigma curves.

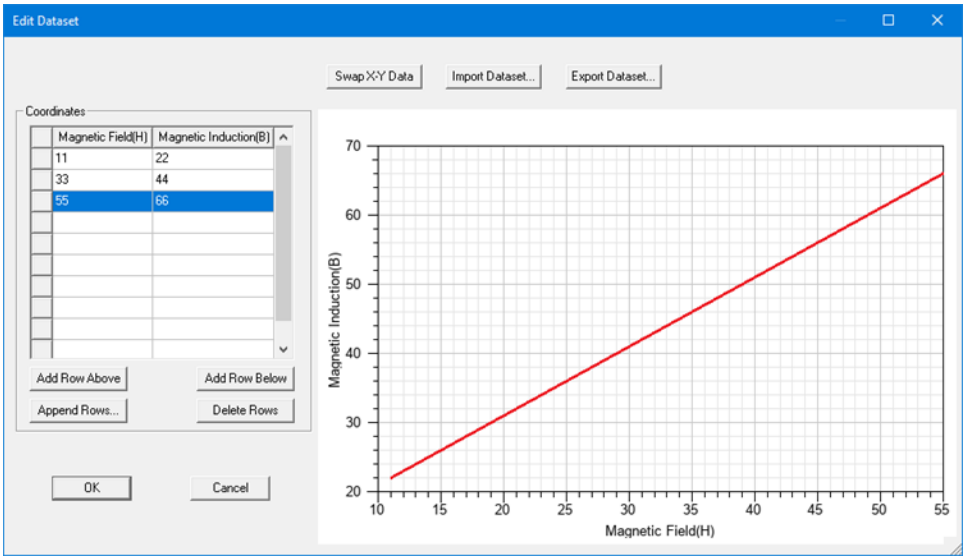
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



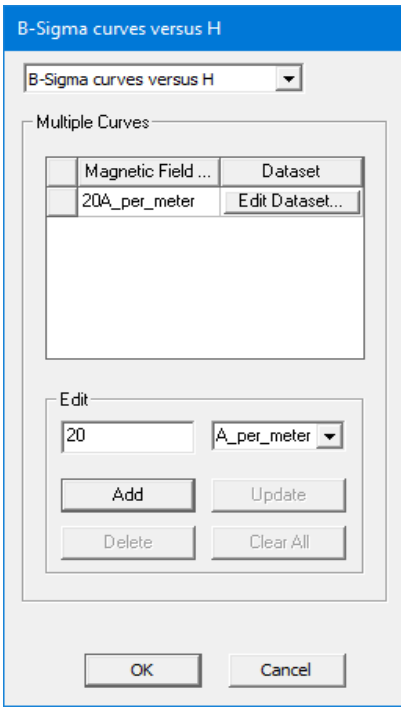
- For **B-H curves versus Stress** – Enter a stress value in the **Edit** text box and click **Add**; then click **Edit Dataset** to open the **Edit Dataset** dialog box, in which you enter coordinate values for the **Magnetic Field (H)** and **Magnetic Induction(B)**.

> **Important:** In the Maxwell 3D application, the inverse magnetostriction effect on a BH curve only responds to normal stress components; shear components are ignored.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

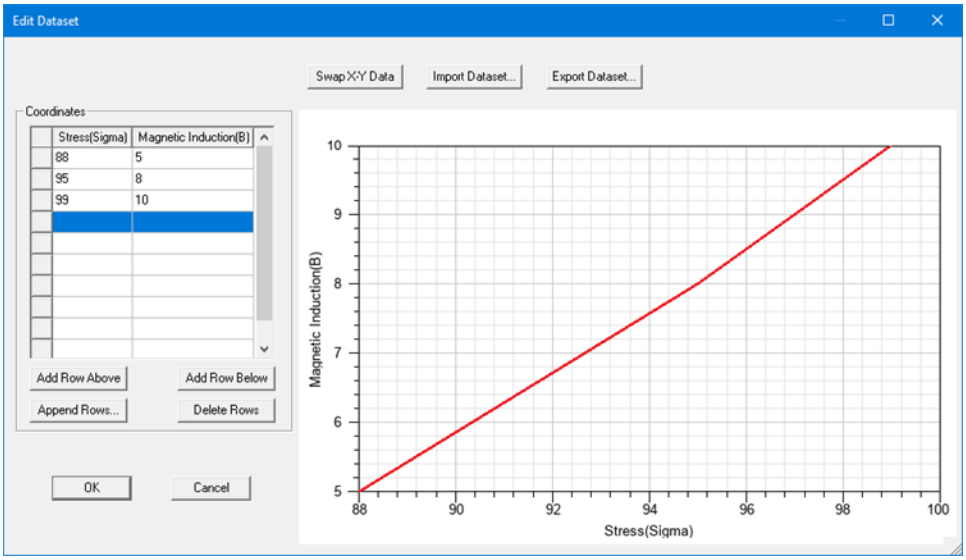


- For **B-Sigma curves versus H** (as shown below) – Enter a magnetic field strength value in the **Edit** text box and click **Add**,



then click **Edit Dataset** to open the **Edit Dataset** dialog box, in which you enter coordinate values for the **Stress (Sigma)** and **Magnetic Induction(B)**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



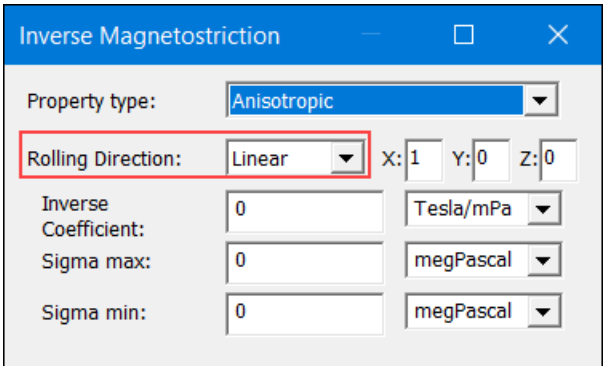
> **Note:** Multiple B-H curves vs stress are interpolated, but not extrapolated. Therefore, B-H curves over the entire range of expected stress levels must be entered for positive stress (tensile stress pointing out of the object) and if anticipated negative stress (compressive stress pointing into the object).

When finished, click **OK** to close the dialog boxes and return to the **Inverse Magnetostriction** dialog box.

c. For **Anisotropic**, select either **Linear** or **Nonlinear** from the **Rolling Direction** drop-down menus. Rolling Direction refers to the alignment of the crystalline structure (and associated magnetic domains) in the direction along which the material (such as the steel laminations used in transformers) passed through the rollers during manufacture. Magnetostriction property values typically differ in each direction.

   - For **LinearRolling Direction**, enter the **X**, **Y**, and **Z** coordinate values, as well as **Inverse Coefficient**, **Sigma max**, and **Sigma min**, where sigma represents mechanical stress.

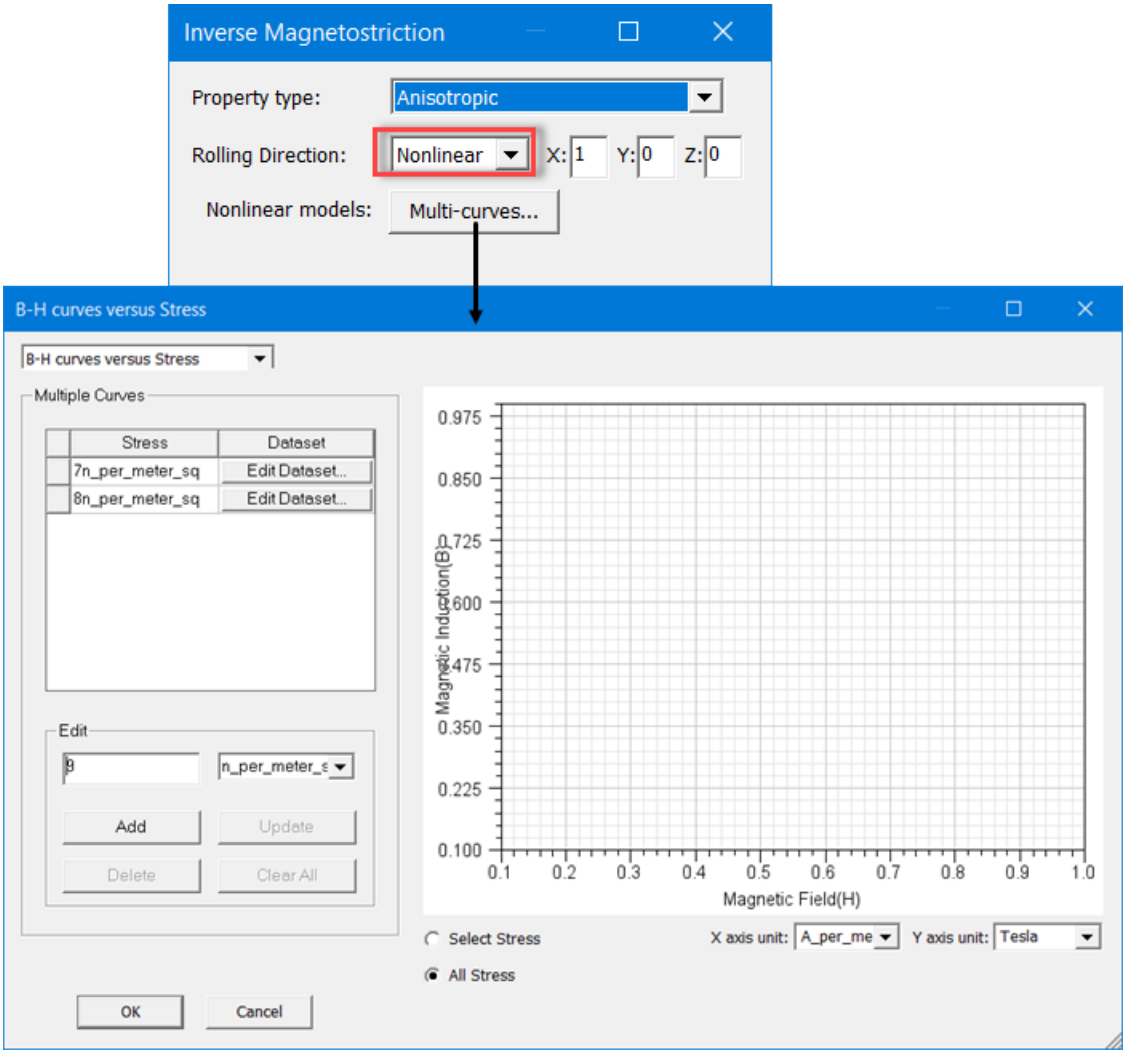
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> The X, Y, and Z components of the Rolling Direction correspond to the **Material Coordinate System Type** set in the **View/Edit Material** window.
>
> - For **Cartesian**: X=X, Y=Y, Z=Z
> - For **Cylindrical**: X=R, Y=Phi, Z=Z
> - For **Spherical**: X=Rho, Y=Theta, Z=Phi

- For **Nonlinear**, click the **Multi-curves** button to add one or more stress-dependent Lambda-H Curves or H-dependent B-Sigma curves.

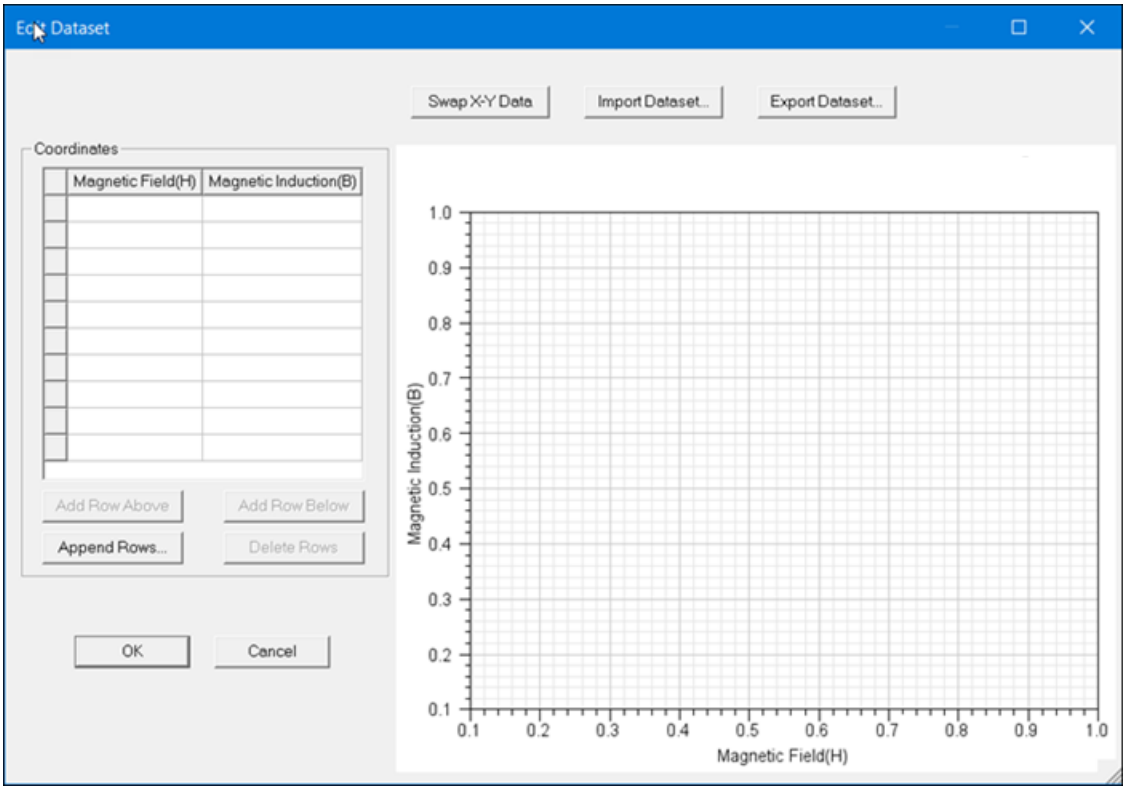
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Important:** In the Maxwell 3D application, the inverse magnetostriction effect on a BH curve only responds to normal stress components; shear components are ignored.

For example, for **B-H curves vs Stress**, enter a stress value in the **Edit** text box and click **Add**, then click **Edit Dataset** to open the **Edit Dataset** dialog box, in which you enter coordinate values for the **Magnetic Field (H)** and **Magnetic Induction (B)**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Click **OK** to close the dialog box and return to the **View/Edit Materials** window.

Perfect Conductors

Perfect conductors are used in Maxwell to approximate the following:

- Magnetic field effects at a conductor surface when current flow is restricted to the conductor surface. This condition can exist in conductors with very high conductivity.
- The magnetic field at the surface of a conductor carrying high frequency current, modeled in the AC magnetic solver when the frequency is very high. Perfect conductors are used in conjunction with Impedance Boundaries in the AC magnetic solvers to handle the following conditions:
    - The skin depth in the conductor is less than two orders of magnitude smaller than the dimensions of the structure. In models like this, the mesher may not be able to create a fine enough mesh in the conductor to compute eddy currents.
    - The magnetic field decays much more rapidly inside the conductor in the direction that's normal to the surface than it does in directions that are tangential to the surface.
    - The AC current source is relatively far away from the surface where eddy currents occur, compared to the size of the skin depth.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Conductors in the Electrostatic solver where the voltage is constant in the conductor and there is no electric field penetration in the conductor.
- In the DC Conduction solver where a material contains a high conductivity.
- In conjunction with boundaries requiring the material on one side of the boundary to be the background region such as impedance or resistance boundaries.

Because of the special treatment of perfect conductors, no field solution is performed in these objects, saving time and computer resources when the approximation does not significantly change the field solution outside the conductor.

In order to define an object as a perfect conductor, do one of the following:

- Assign the Perfect Conductor material to the object in the Material Manager.
- Set the object to have a material with a conductivity greater than the conductivity defined in the *Material Threshold* section of the Design Settings dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7 - Drawing a Model

After you insert a design into the current project, you can draw a model of the electromagnetic structure. The general strategy is to build the model as a collection of 3D objects. You can assign any single material to each 3D object.

You can create 3D objects by using the modeler's **Draw** commands or you can draw 1D and 2D objects, and then manipulate them to create 3D objects. Objects are drawn in the **3D Modeler** window. You can also import objects from other systems, and use the 3D component libraries.

To open a new **3D Modeler** window, do one of the following:

- Insert a new design into the current project.
- Double-click an Ansys Electronics Desktop design that uses the 3D Modeler (HFSS, Q3D, Maxwell (2D and 3D), or Icepak) in the project tree.

If a **3D Modeler** window for an existing design is not open, do one of the following:

- Click **[solverName]** > **3D Model Editor**.
- Right-click the design name in the project tree, and then click **3D Model Editor** on the shortcut menu.

The model you draw is saved with the current project when you click **File** > **Save**.

> **Note:**
>
> If you access your machine via Remote Desktop, if Ansys Electronics Desktop is running and one or more modeler windows are open, those modeler windows automatically close. The message manager window displays a message indicating that Ansys Electronics Desktop closed the modeler windows.

When working with multiple projects, or when a project has multiple designs, you may have multiple **Modeler** windows available. To switch to the modeler window associated with a specific design:

1. In the Project Manager window, select the **Design** of interest.
2. Click **[solverName]** > **3D Model Editor** to focus the modeling window on the selected design.

If the menu command is unavailable, then the selected design is already in the modeler window.

Setting Units of Measurement for the Model

You can specify the units of measurement for drawing geometric models. After the units of measurement have been specified, they are assigned to the objects in the **3D Modeler** window.

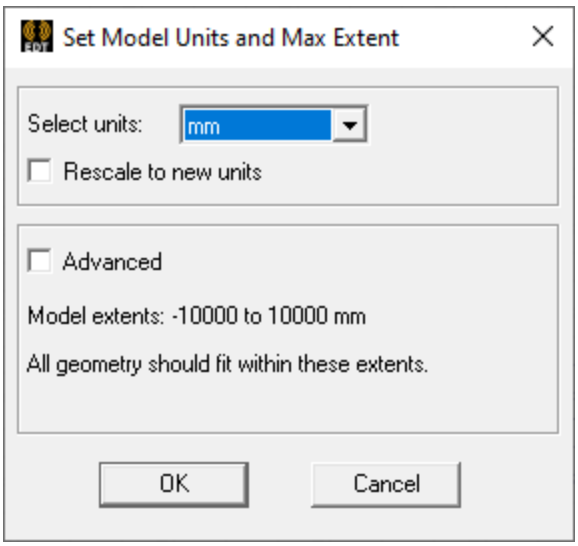
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can then choose to display the model's dimensions in the new units, or rescale the model's dimensions to the new units.

To set the model's units of measurement:

1. Click **Modeler** > **Units**.

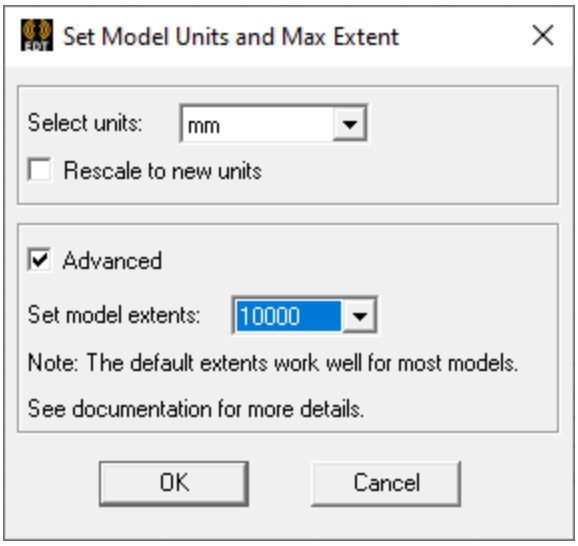
   The **Set Model Units** dialog box appears.



2. Select the new units for the model from the **Select units** drop-down menu.
3. Specify how the change in units affects the model:
   - Select the **Rescale to new units** option to rescale the dimensions to the new units. For example, selecting centimeters (cm) as the new unit of measurement results in a dimension of 10 millimeters (mm) becoming 10 cm.
   - Clear the **Rescale to new units** option (the default) to convert the dimensions to the new units without changing their scale. For example, selecting cm as the new unit of measurement results in a dimension of 10 mm becoming 1 cm.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. While most models should fit the default, you can check Advanced to enable Max model extent (E) to specify model extents in the 3D modeler.



.

Parasolid kernel has a strict size limit of +/- 500 units. As many AEDT models go beyond this limit in the desired units, we have implemented internal scaling to allow creation of models with a larger extent. With our default model extent of 10,000, all geometry will be internally scaled down by a factor of 100 to fit within parasolid size limit. We recommend using the default extent of 10,000 as much as possible. Modeling operations will output an error if model goes beyond 50,000 units.

A second extent option of 100 is available in the advanced options.

If model extent is set to 100, we do not scale geometry internally. Modeling operations will output an error if model goes beyond +/- 500 units. Extent of 100 should be only used in rare situations where it is determined that scaling is causing a failure, and there is no workaround.

5. Click **OK** to apply the new units to the model.

Drawing Objects

You can draw one-, two-, or three-dimensional objects using the **Draw** commands. You can alter objects individually or together to create the geometry of your structure. In the **Tools** > **Modeler Options,** **Drawing** tab, you can set a default to either draw objects directly with the mouse or by invoking a **Properties** dialog in which you can enter the values for the object dimensions. The **Dialog** mode drawing feature works with the equation based curve, equation based surface, and all two and three dimensional objects. You can toggle to **Point** mode via the **F3** function key and to **Dialog** mode via the **F4** function key. When you use the **Dialog** mode for drawing objects the Edit property of new primitives setting is ignored.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

One-dimensional (1D) objects in the modeler include straight line, arc line, center-point arc, and spline segments, or a combination of these - called polylines. One-dimensional objects are open objects; their boundaries do not enclose a region, unless you connect their endpoints. They have length, but no surface or volume. Generally they are used as temporary objects from which to create 2D objects.

Two-dimensional (2D) objects in the modeler include objects such as equation based surfaces, rectangles, ellipses, circles, and regular polygons. Two-dimensional objects are closed sheet objects; their boundaries enclose a region. You can create 2D sheet objects by covering the enclosed region. In many applications (FSS, antennas) it is essential to calculate net power flow through a surface.

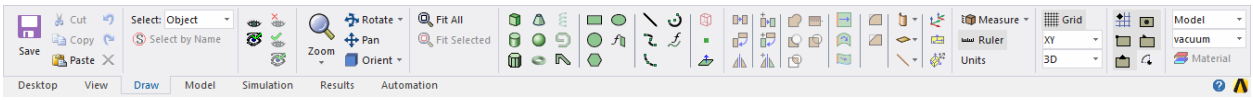
You can also edit the properties of a polyline from the history tree to assign it a Cross Section property as line or rectangular. If you then assign it either a height or a width, the polyline becomes a sheet object.

By default, the history tree organizes sheet objects according to their boundary assignments. To change this, select the **Sheets** icon, and right-click to display the **Group Sheets by Assignment** check box. Within the calculator sheet objects are listed under surface.

Three-dimensional (3D) objects in the modeler include objects such as boxes, cylinders, regular polyhedra, cones, spheres, torii, and helices. These objects have boundaries that enclose a region with volume.

You can create 3D objects by manipulating 2D objects along a plane or by using the appropriate **Draw** commands. You can also edit the properties of a polyline from the history tree to assign it a Cross Section property as circle rectangular. If you then assign it an appropriate diameter or both height or a width, the polyline becomes a 3D object.

You can access the **Draw** commands via icons on the **Draw** tab ribbon:



By default, the history tree organizes 3D objects by material. To change this, select the **Objects** icon, and right-click to display the shortcut menu to see **History Tree Layout commands**. You can also use Group Commands for Modeler Objects.

While you draw objects you can also:

- Select Movement Mode as 3D, In Plane, Out of Plane, Along X, Y or Z axis.
- Select Grid Plane as XY, YZ, or XZ.
- Set the Drawing Plane
- Select the Coordinate System and choose Cartesian, Cylindrical or Spherical coordinates.
- Change Snap Settings

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Set Reference Point for the movement mode
- Adjust the View

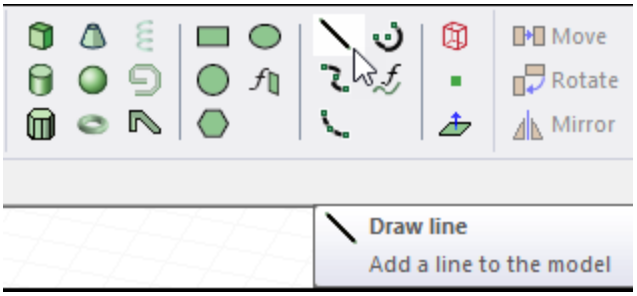
After you draw an object in the **3D Modeler** window, you can modify the object's properties, such as its position, dimensions, or color, in the **Properties** dialog box. Most model object properties can be assigned as Design variables when can then be manipulated during the solve to test their effect on the solution. For non-model objects, you can use Post Processing variables (default and Design variables,.

> **Note:**
>
> If you access your machine via Remote Desktop, if Ansys Electronics Desktop is running and one or more modeler windows are open, those modeler windows automatically close. The message manager window displays a message indicating that Ansys Electronics Desktop closed the modeler windows.

Drawing a Line Segment

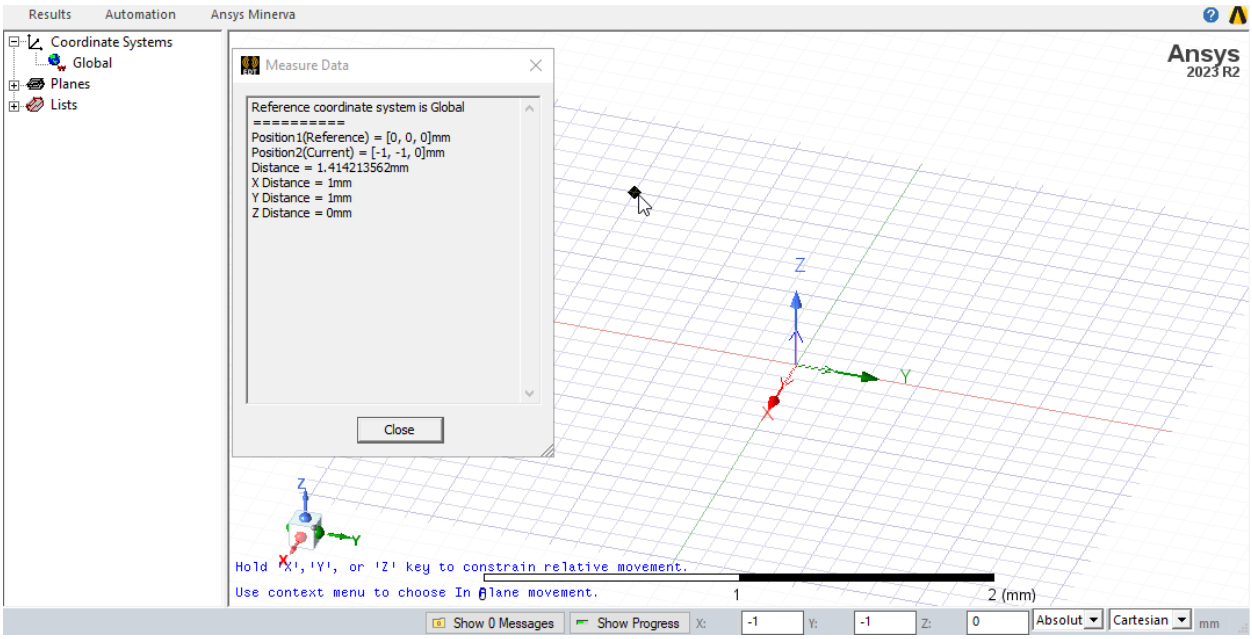
To create an object with one or more straight line segments, use the **Draw** > **Line** command.

1. From the menu bar, click **Draw** > **Line**, or on the **Draw** ribbon tab, click the **Draw line** icon:



2. Select the first point of the line in one of the following ways:
   - Click the point in the Modeler window, checking the Measure Data dialog box for the precise coordinates.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

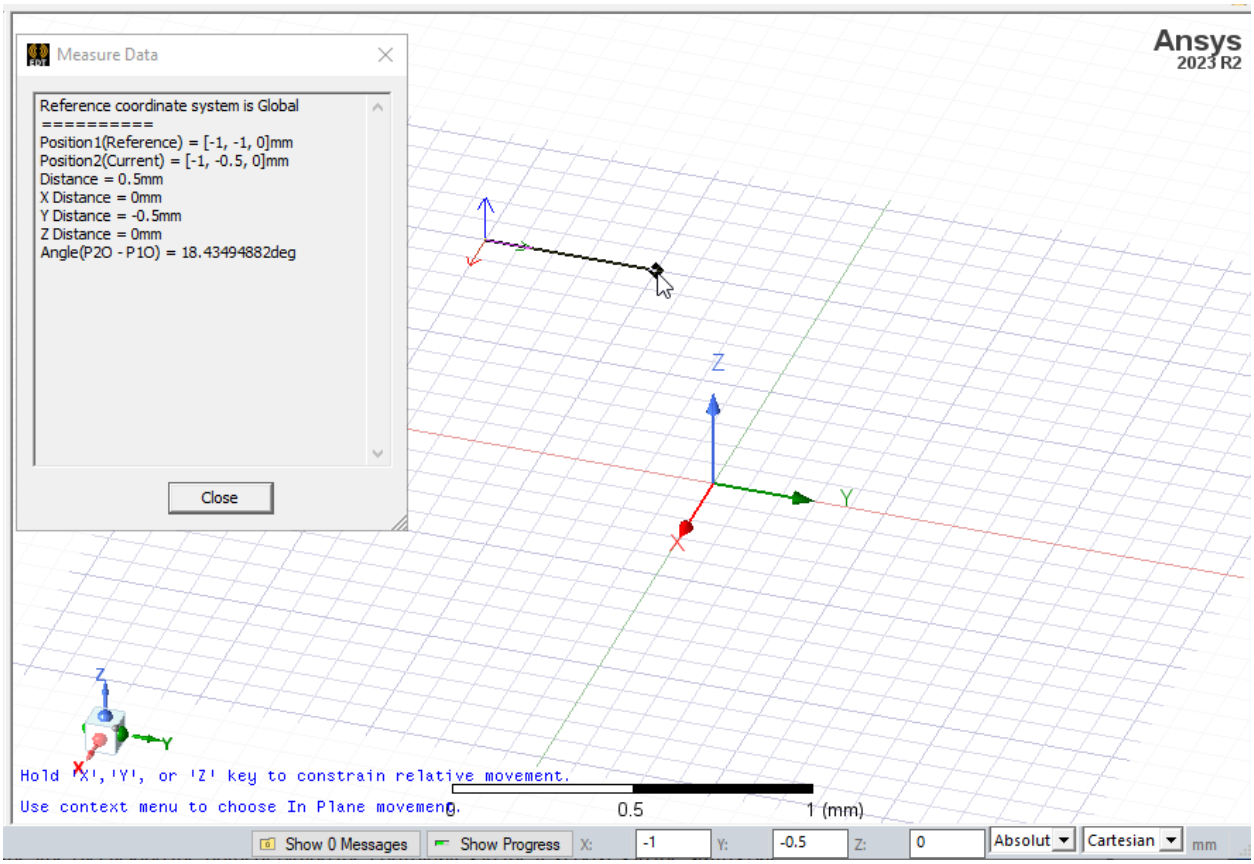


- You can accept the first point or change it by editing its coordinates in the X, Y, and Z boxes in the status bar. Note that the Status bar includes selections for Absolute or Relative Coordinates, as well as for Cartesian, Cylindrical and Spherical Coordinates.

To delete the last point that was entered, click **Undo Previous Segment** on the shortcut menu. After using the undo feature, you can also use **Redo Previous Segment** on the shortcut menu.

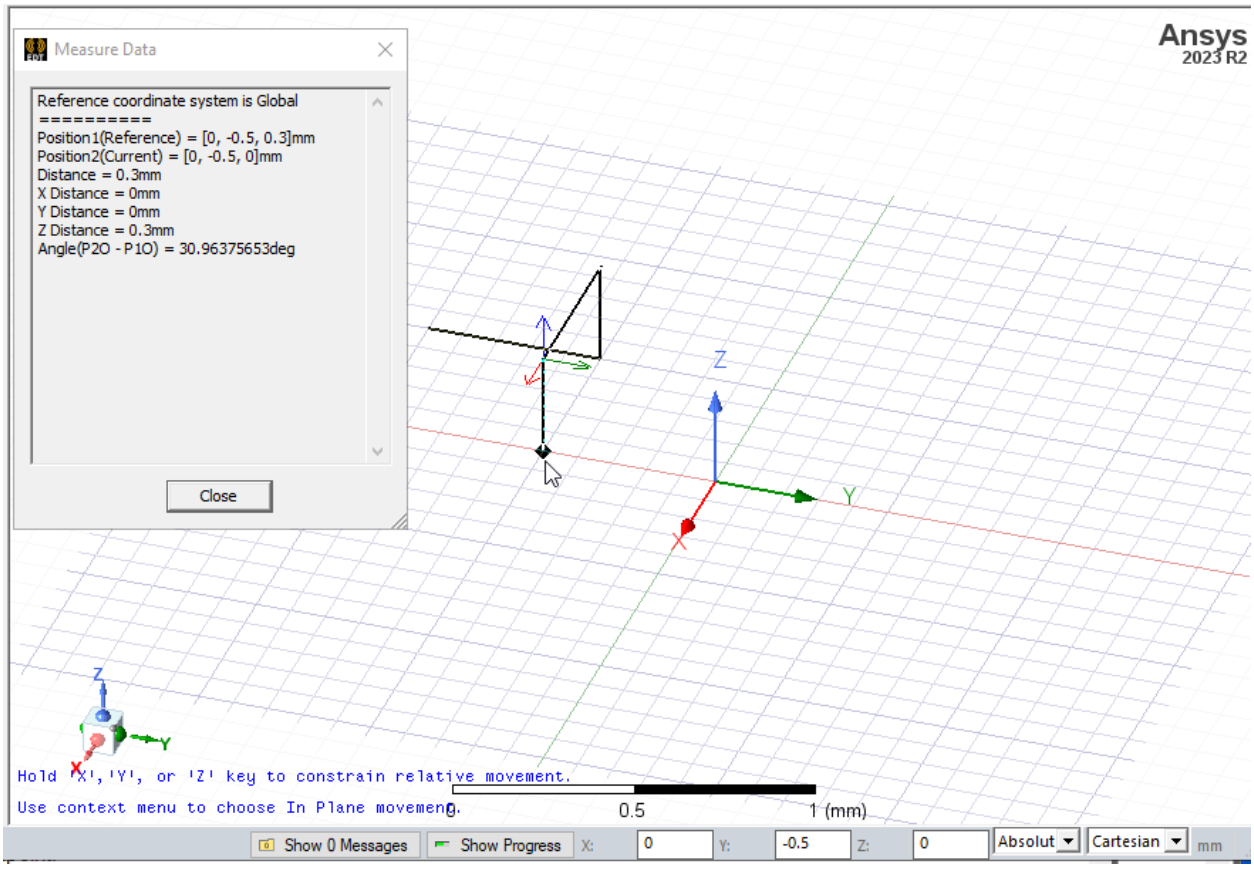
3. Select the next point of the line by clicking the point or typing the coordinates in the text boxes in the status bar.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The endpoint serves as the start point for a subsequent line segment. You can specify the plane through which the line passes by selecting the X, Y, or Z keys to constrain movement. For example, the following figure shows a line with five points that includes X, Y, and Z movements.

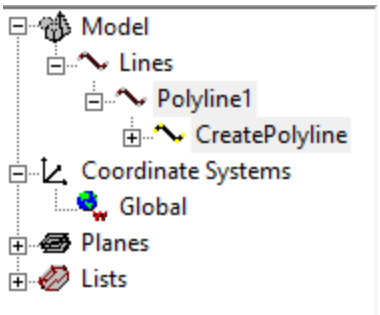
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To delete all points and start over, press **Esc** or click **Escape Draw Mode** on the shortcut menu.

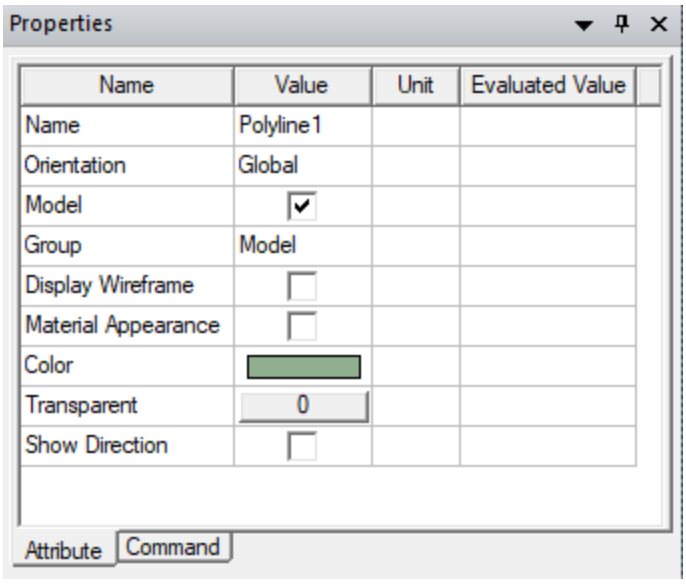
4. Complete the line in one of the following ways:
   - Double-click the endpoint.
   - Click **Done** on the context (right-click) menu.
   - Press **Enter**.

   The new line appears selected in the Modeler window and shown in the History tree.

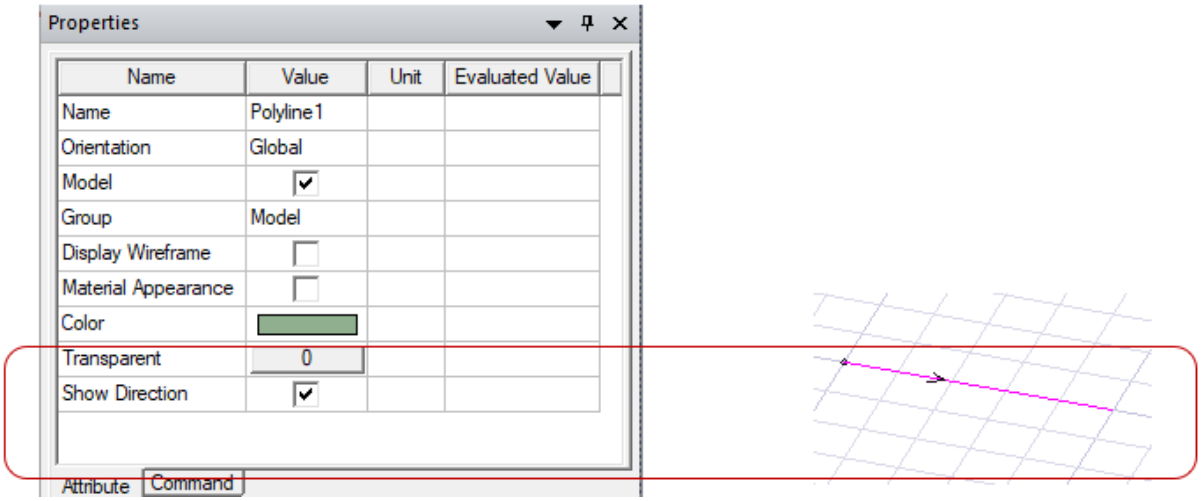


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, in which you can modify the object's attributes by editing the object's properties. You can also do this through the docked Properties window.

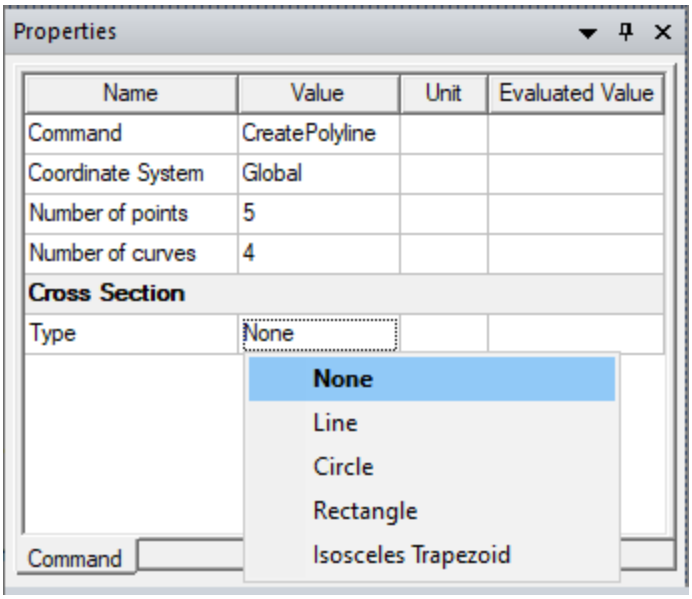


Those listed under the Command tab describe the commands used to create the object. These commands also appear in the History Tree. The properties listed as line attributes include Name, Orientation, whether a Model object, whether to Display Wireframe, Color, Transparency, and whether to Show Direction as arrows. The Show Direction property is most helpful to unambiguously show the line start orientation when plotting fields along a line.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. The Command tab **Properties** includes the Coordinate System that applies to the line, the number of points and curves, as well as the ability to assign a specific Cross Section to the line.



6. If you created the line with the Modeler option for editing properties of new primitives, click **OK** to close the **Properties** dialog box.

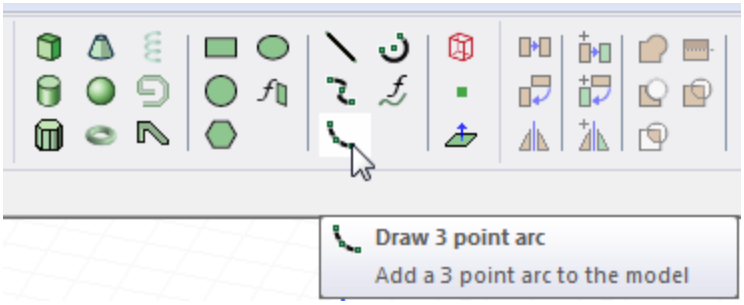
> **Note:** While drawing a polyline, you can switch between straight line, arc line, or spline segments using the **Set Edge Type** commands on the shortcut menu.

Drawing a Three-Point Arc Line

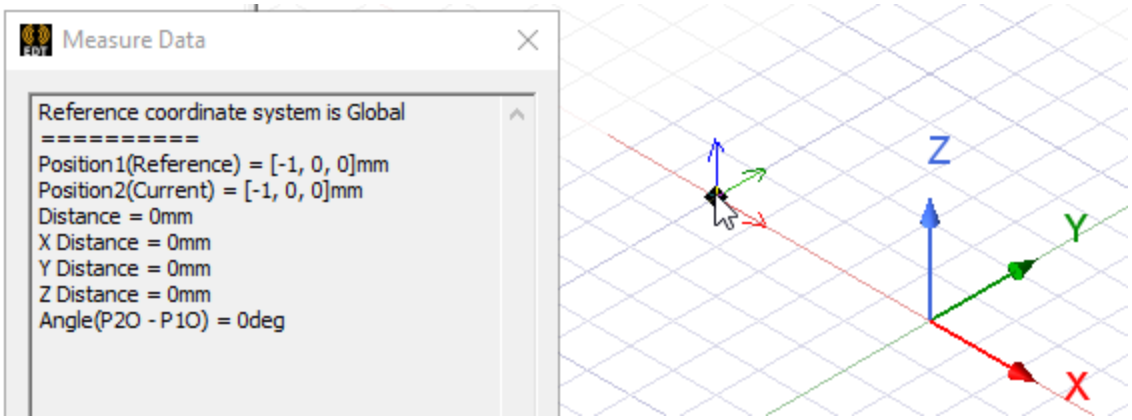
In the modeler, a three-point arc line segment is an arced line defined by three points on its curve. Use the **Draw** > **Arc** > **3 Point** command to create a polyline object with one or more arc line segments. Before you draw a three-point line arc, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw** > **Arc** > **3 Point**, or on the **Draw** ribbon tab, click the **Draw 3 point arc** icon:

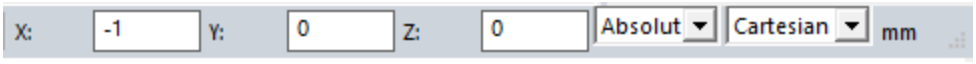
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



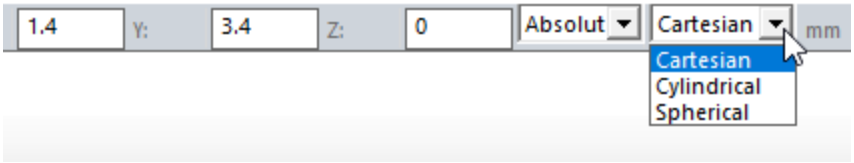
2. Select the start point of the arc in one of the following ways:
   - Click the point.



   - Type the point's coordinates in the **X, Y,** and **Z** text boxes.



     The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop-down menu options to specify the line in Cartesian, Cylindrical, or Spherical coordinates.



3. Select the midpoint of the arc by clicking the point or typing the coordinates in the text boxes in the status bar.

   To delete the last point that was entered, click **Undo Previous Segment** on the shortcut menu. After using the undo feature, you can also use **Redo Previous Segment** on the shortcut menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To delete all points and start over, press **Esc** or click **Escape Draw Mode** on the shortcut menu.

4. Select the endpoint of the arc by clicking the point or typing the coordinates in the text boxes in the status bar.

   The endpoint serves as the start point for a subsequent arc line segment.

5. If the endpoint is the last point of the polyline object, double-click the point to complete the polyline or click **Done** on the shortcut menu.

   If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, in which you can modify the object's attributes by editing the Properties. Those listed under the Command tab describe the commands used to create the object. These commands also appear in the History Tree. The Properties listed as line attributes include Name, Orientation, whether a Model object, whether to Display Wireframe, Color, Transparency, and whether to Show Direction as arrows. The Show Direction property is most helpful to unambiguously show the line start orientation when plotting fields along a line.

6. Click **OK**.

   Based on the three points you specified, the modeler calculates the center point and radius of the arc and draws an arced line through the three points.

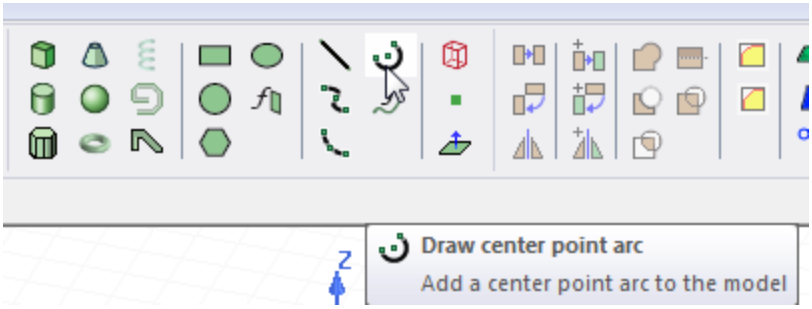
> **Note:**
>
> While drawing a polyline, you can switch between arc line, straight line, or spline segments using the **Set Edge Type** commands on the shortcut menu.

Drawing a Center-Point Arc Line

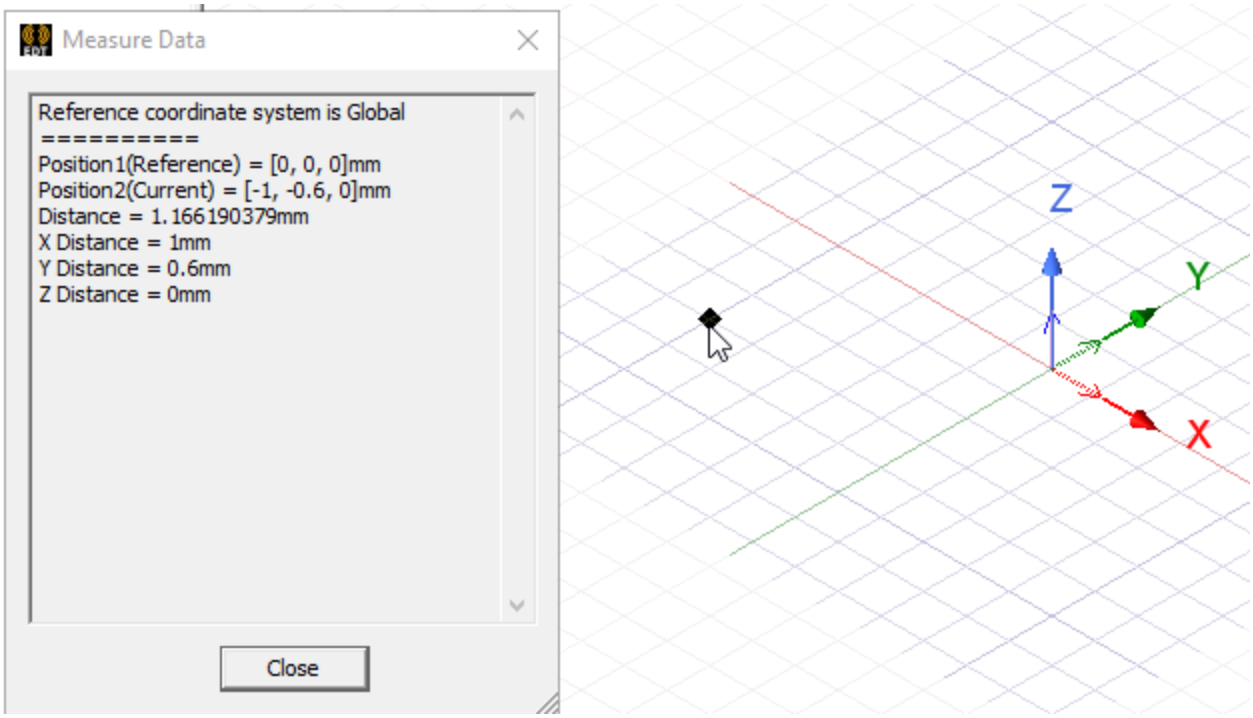
In the modeler, a center-point arc line segment is an arced line defined by a center point, start point and angle. Use the **Draw** > **Arc** > **Center Point** command to create a polyline object with one or more center-point arc line segments. Before you draw a center-pint arc line, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw** > **Arc** > **Center Point**, or on the **Draw** ribbon tab, click the **Draw center point arc** icon:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



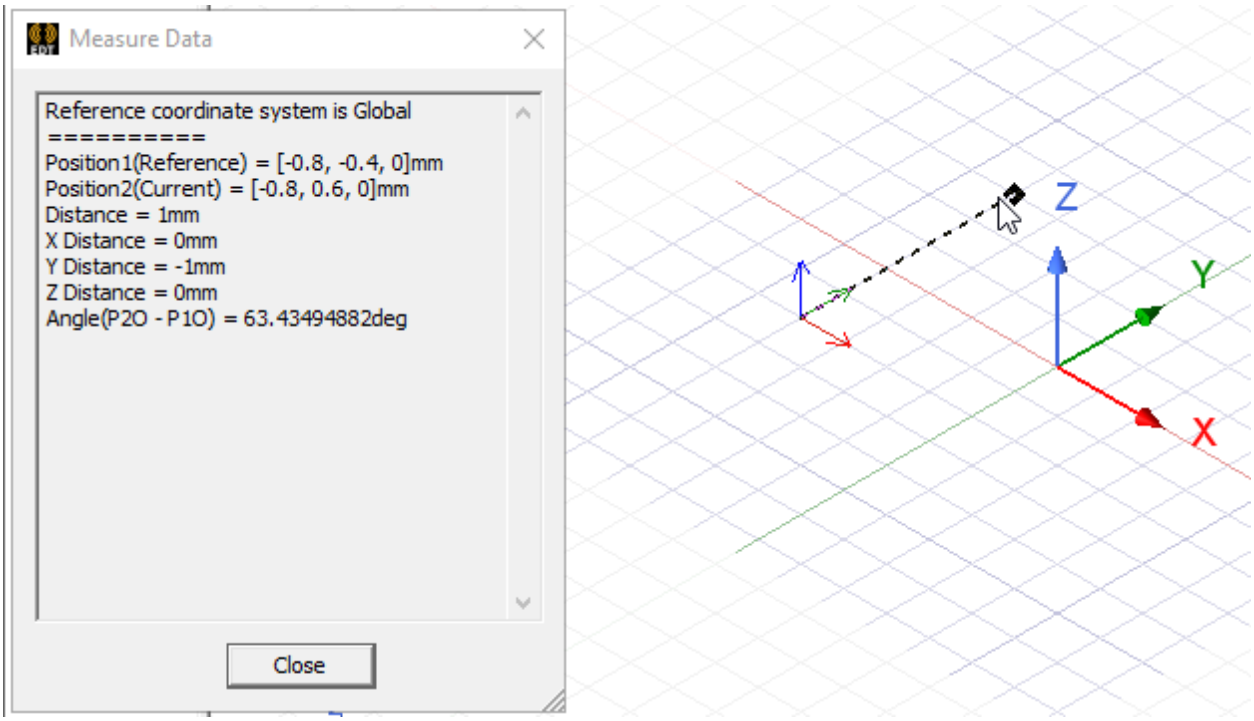
2. Select the center point of the arc in one of the following ways:
   - Click the point.



   - Type the point's coordinates in the text boxes in the status bar.

3. Select the start point, or radius, of the arc by clicking the point or typing the coordinates in the text boxes in the status bar.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

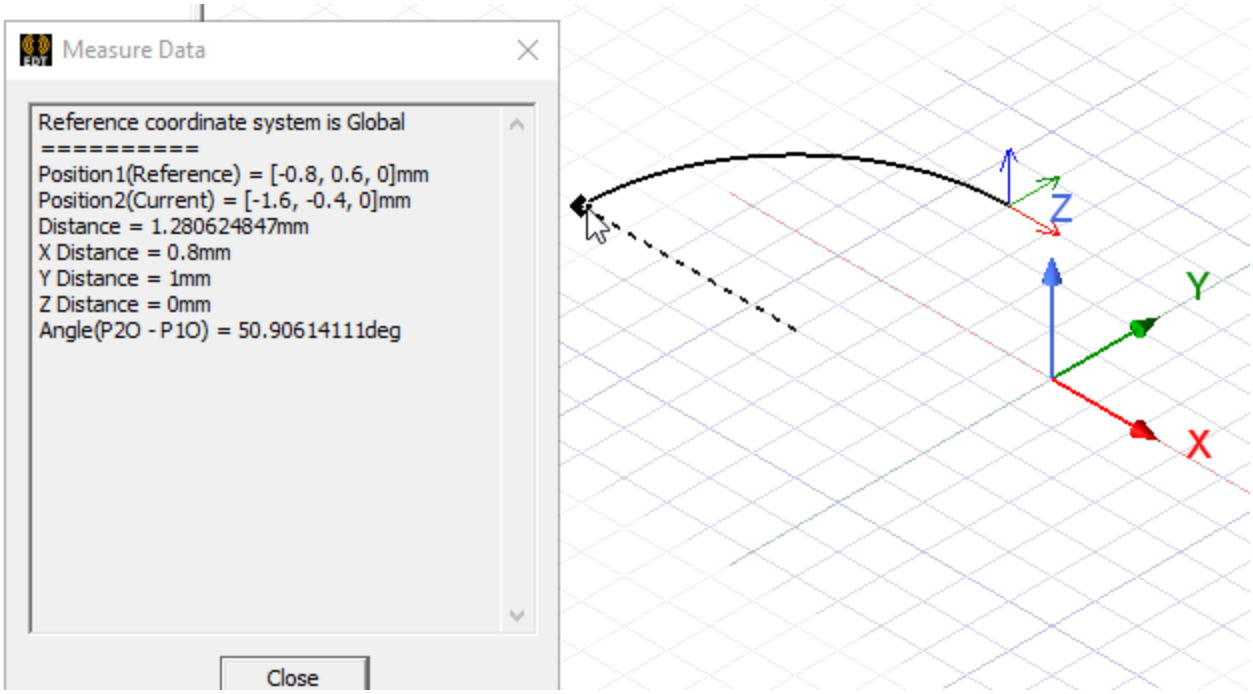


To delete the last point that was entered, click **Undo Previous Segment** on the shortcut menu. After using the undo feature, you can also use **Redo Previous Segment** on the shortcut menu.

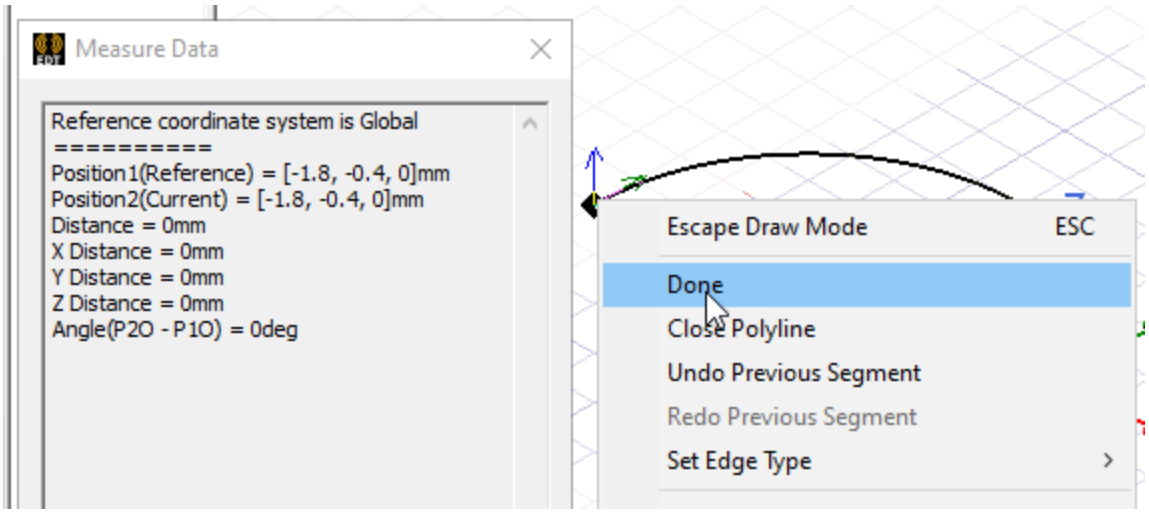
To delete all points and start over, press **Esc** or click **Escape Draw Mode** on the shortcut menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Sweep the angle or the endpoint of the arc by clicking the point or typing the coordinates in the text boxes in the status bar.



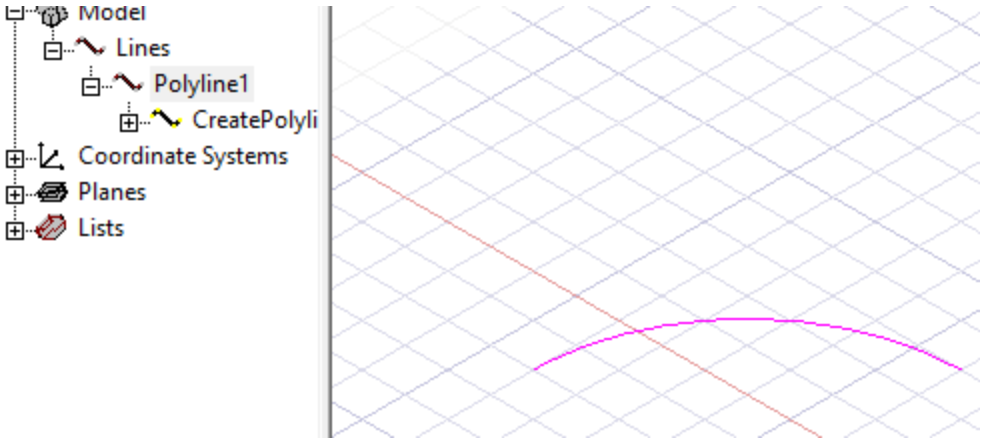
5. If the endpoint is the last point of the polyline object, double-click the point to complete the polyline or click **Done** on the shortcut menu.



If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's attributes. The new polyline

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

appears in the modeler window and in the History tree.



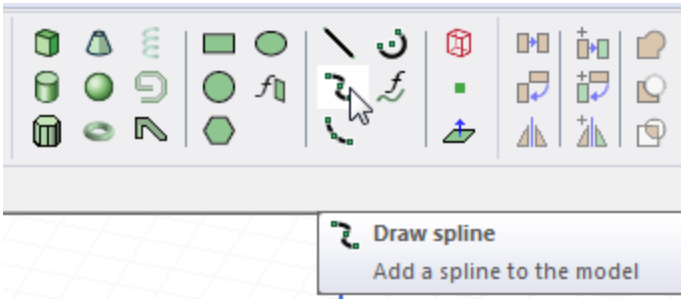
6. Click **OK**.

> **Note:**
>
> While drawing a polyline, you can switch between arc line, straight line, or spline segments using the **Set Edge Type** commands on the shortcut menu.

Drawing a Spline

A spline is a curved line defined by three points. The modeler uses a natural spline type: a piece wise cubic spline with an end condition that has a derivative of zero. Use the **Draw** > **Spline** command to create a polyline object with one or more spline segments. Before you draw a spline, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

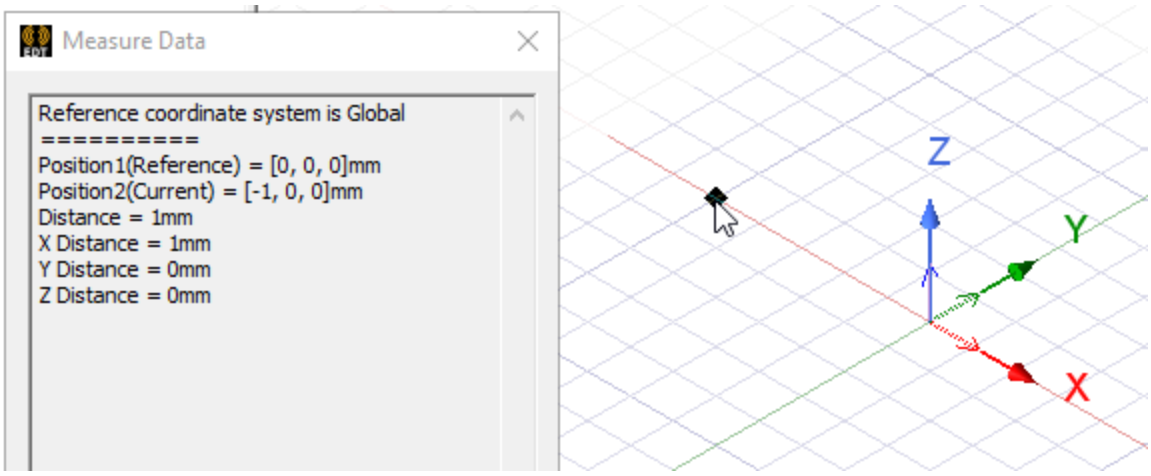
1. From the menu bar, click **Draw** > **Spline**, or on the **Draw** ribbon tab, click the **Draw spline** icon:



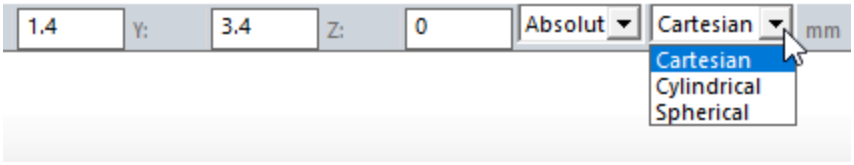
2. Select the spline's start point in one of the following ways:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click the point.



- Type the point's coordinates in the text boxes in the Status bar, and then press **Enter**.
  The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop-down menu options to specify the spline in Cartesian, Cylindrical, or Spherical coordinates.

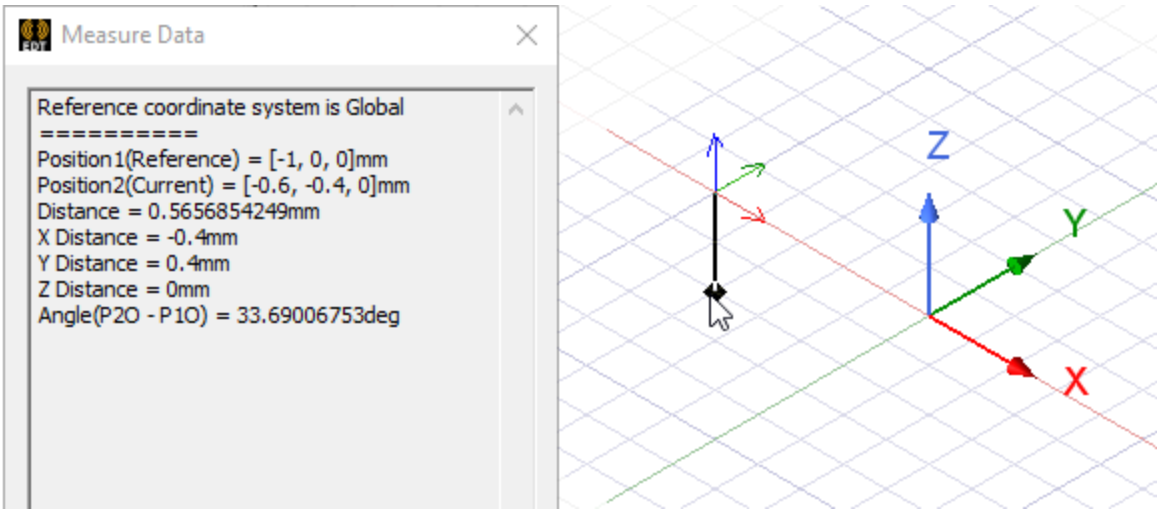


To delete the last point entered, click **Undo Previous Segment** on the shortcut menu. After using the undo feature, you can also use **Redo Previous Segment** on the shortcut menu.

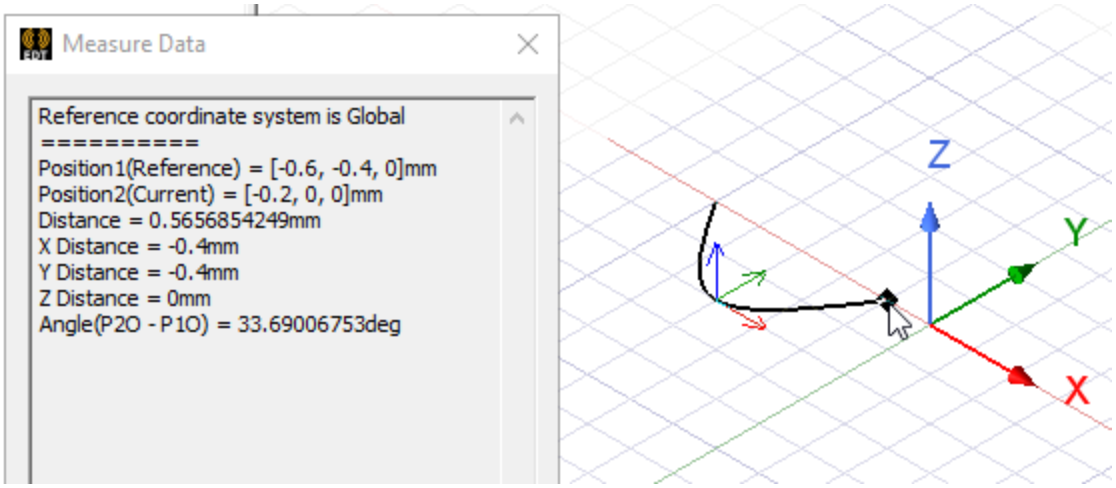
To delete all selected points and start over, press **Esc** or click **Escape Draw Mode** on the shortcut menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select the midpoint of the spline by clicking the point or typing the coordinates in the text boxes in the status bar.



4. Select the endpoint of the spline by clicking the point or typing the coordinates in the text boxes in the status bar.

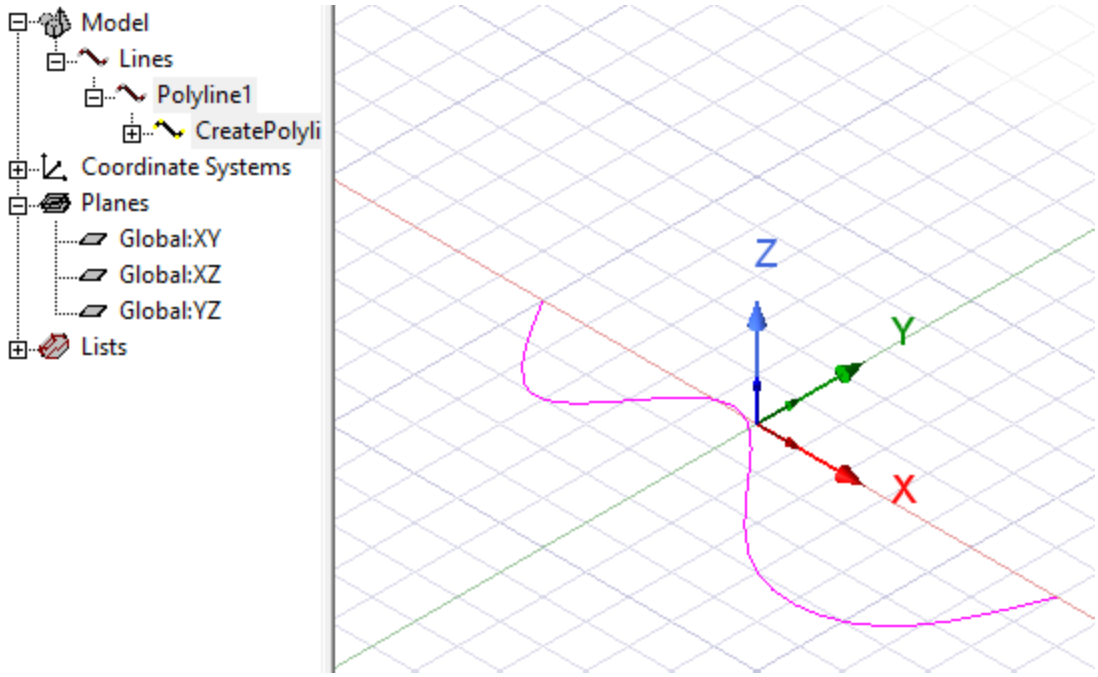


The endpoint serves as the start point for a subsequent spline segment. You can complete the spline at this point, or add additional segments.

5. Complete the spline in one of the following ways:
   - Double-click the endpoint.
   - Click **Done** on the shortcut menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Press **Enter**.
  The spline appears in the Modeler window and the History tree.



If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, in which you can modify the object's attributes by editing the Properties. Those listed under the Command tab describe the commands used to create the object. These commands also appear in the History Tree. The Properties listed as line attributes include Name, Orientation, whether a Model object, whether to Display Wireframe, Color, Transparency, and whether to **Show Direction** as arrows. The **Show Direction** property is most helpful to unambiguously show the line start orientation when plotting fields along a line.

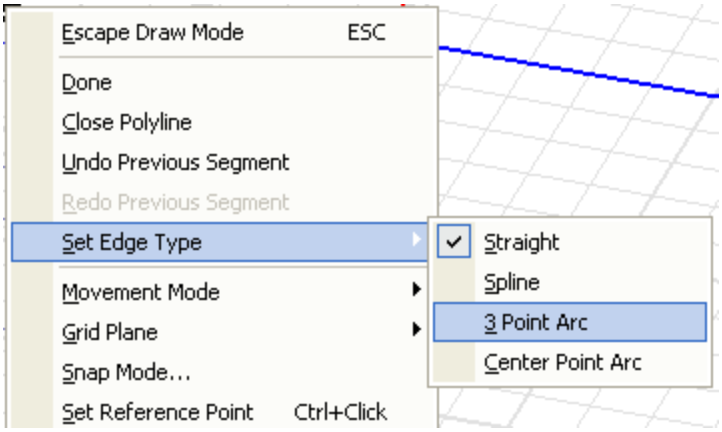
6. If you have edited Properties, click **OK**.

> **Note:**
>
> While drawing a polyline, you can switch between spline, straight line, or arc line segments using the **Set Edge Type** commands on the shortcut menu.

Drawing a Polyline

A polyline is a single object that includes any combination of straight line, arc line, or spline segments. The endpoint of one segment is the start point for the next segment. Use the shortcut menu's **Set Edge Type** commands to switch between straight line, arc line, or spline segments while drawing a polyline.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

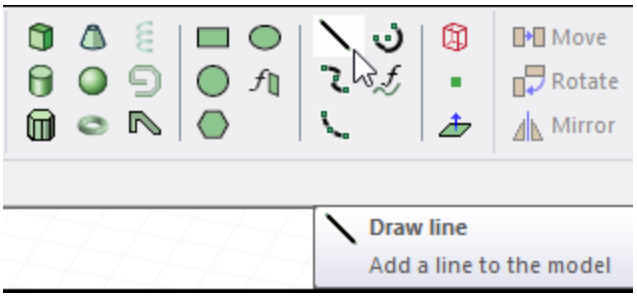


Before drawing a polyline, determine how closed polylines will be handled in the 3D Modeler options, as follows:

Using the menu bar, click **Tools** > **Options** > **General Options**. In the *Options* dialog box that appears, choose **3D Modeler** > **Operation** from the tree on the left side. Then, in the **Polyline Creation** section of the operation options, select or clear the **Automatically cover closed polylines** check box.

If checked, closed polylines become sheet objects and are listed under *Sheets* in the History Tree. If unchecked, closed polylines are listed under *Lines* in the History Tree.

1. From the menu bar, click **Draw** > **Line**, or on the **Draw** ribbon tab, click the **Draw line** icon:



2. Right-click in the **3D Modeler** window to access the shortcut menu, and then point to **Set Edge Type**.
3. Click **Straight, Spline, 3 Point Arc,** or **Center Point Arc**, depending on which type of polyline segment you want to draw.
4. Depending on your selection in the previous step, do one of the following:
   - If you clicked **Straight**, follow the procedure for drawing a straight line.
   - If you clicked **Spline**, follow the procedure for drawing a spline.
   - If you clicked **3 Point Arc**, follow the procedure for drawing a three-point arc line.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you clicked **Center Point Arc**, follow the procedure for drawing a center-point arc line.

5. Repeat steps 2 and 3 for each segment of the polyline object. The endpoint of the previous segment serves as the start point for the next segment.

   The shortcut menu lets you do the following for each segment:

   Undo Previous Segment or **Redo Previous Segment**.

6. Complete the polyline in one of the following ways:
   - Double-click the endpoint of the final segment.
   - Click **Done** on the shortcut menu.

   > **Note:**
   >
   > To connect the polyline's start and endpoints, click **Close Polyline** on the shortcut menu.

   If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, in which you can modify the object's attributes by editing the Properties. Those listed under the Command tab describe the commands used to create the object. These commands also appear in the History Tree. The Properties listed as line attributes include Name, Orientation, whether a Model object, whether to Display Wireframe, Color, Transparency, and whether to Show Direction as arrows. The Show Direction property is most helpful to unambiguously show the line start orientation plotting fields along a line. Notice that by going to the History Tree and selecting Create Polyline for that object, you can assign a cross section and dimensions to a polyline.

7. Click **OK**.

   If you select a polyline in the History Tree, you can use the Measure mode to see the total length.

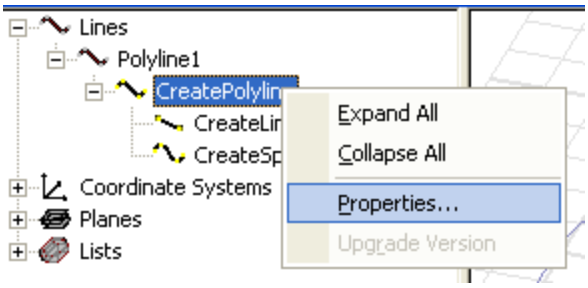
Assigning a Cross Section and Dimensions to a Polyline

By viewing the properties of a polyline from the History Tree, you can assign either a Line, Circle, Rectangle or Isosceles Trapezoid cross section to a polyline. This assignment enables editable dimension properties of width for a line, diameter for a circle, and height and width for a rectangle or trapezoid. To assign a cross section to a polyline:

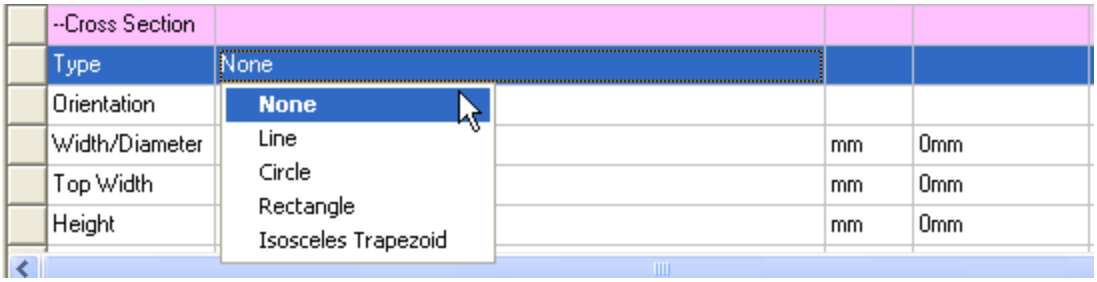
1. In the History Tree of the Modeler window, select the **CreatePolyline** entry for the polyline that you want to give a cross section.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This action displays the polyline properties in the docked **Properties** window (if you have it displayed). Alternatively, right-click the **CreatePolyline** entry and choose **Properties...** from the shortcut menu to display the **Properties** dialog box.



2. In the **Cross Section** section of the properties, in the second column of the **Type** row, click **None** to display the cross section choices.



3. Choose a **Line, Circle, Rectangle,** or **Isosceles Trapezoid** cross section.
    - Selecting **Line** causes the Cross Section area of the polyline properties to display editable fields for **Orientation** and **Width**.
    - Selecting **Circle** causes the Cross Section area of the polyline properties to display an editable field for **Diameter**.
    - Selecting **Rectangle** causes the Cross Section area of the polyline properties to display editable fields for **Orientation**, **Width**, and **Height**.
    - Selecting **Isosceles Trapezoid** causes the Cross Section area of the polyline properties to display editable fields for **Orientation**, **Width/Diameter**, **Top Width**, and **Height**.
4. If you select **Line**, **Rectangle,** or **Isosceles Trapezoid** you can edit the **Orientation** as **Auto, X, Y,** or **Z**. This setting specifies the direction in which the **Height** dimension extends.
5. Specify the dimensions and select the units for the Cross section.

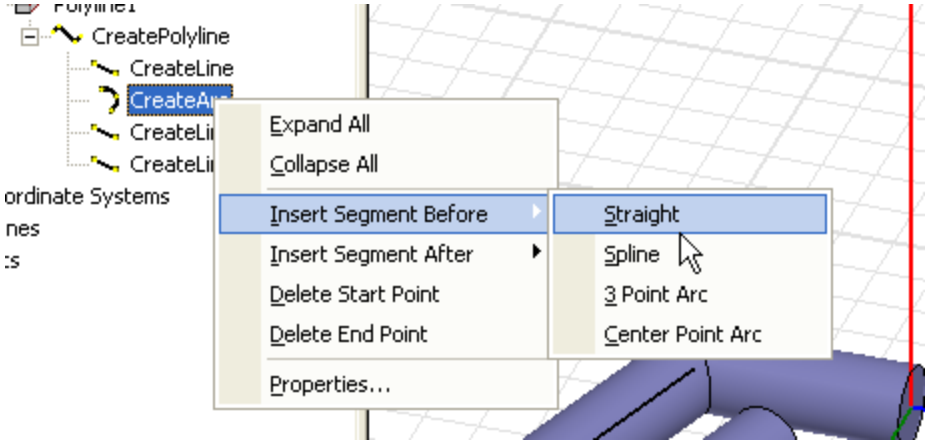
    Type a value in the dimension field(s) and select units from the drop-down menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The dimensions must be reasonable relative to the specified shape and orientation of the polyline. If the polyline cannot be extended into current Orientation for the given dimension(s), you will receive a warning. If you receive a warning, check the Orientation, dimension and units.

When the modeler can extend the dimensions legally, it displays the modified object, and lists it in the History Tree as either a Sheet object (Line or one dimensional Rectangle cross section) or as a Solid object (Circle or two dimensional Rectangle cross section).

6. You can modify the new polyline either by editing the properties, or by using the History Tree to select one of the segments that make up the polyline and right-clicking to display the shortcut menu showing commands to **Insert**, **Delete**, or display the segment **Properties**.



A segment that you select in the History Tree is indicated in the Modeler window by black highlighting. If you insert a new segment, it adopts the dimensions you specified for the polyline object.

Inserting Line Segments

You can insert line segments of various kinds for existing line, arc, or spline objects.

1. Select a line segment in the History Tree (not the modeler window). The individual segments entries are **CreateLine**, **CreateArc**, **CreateAngularArc**, or **CreateSpline**.

   This action highlights the object and enables the **Insert Line Segment** subcommands in the **Draw** menu.

2. Using the menu bar, go to the **Draw** > **Line Segment** submenu and then choose either the **Insert Segment Before** or **Insert Segment After** submenu.

   Alternatively, right-click a polyline segment in the History Tree and, from the shortcut menu, choose either the **Insert Segment Before** or **Insert Segment After** submenu

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Use this submenu to specify the kind of segment to add. The inserted segment can be: **Straight, Spline. 3 Point Arc,** or **Center Point Arc**.
4. Depending on your selection in the previous step, do one of the following:
   - If you clicked **Straight**, follow the procedure for drawing a straight line.
   - If you clicked **3 Point Arc**, follow the procedure for drawing a three-point arc line.
   - If you clicked **Center Point Arc**, follow the procedure for drawing a center-point arc line.
   - If you clicked **Spline**, follow the procedure for drawing a spline.

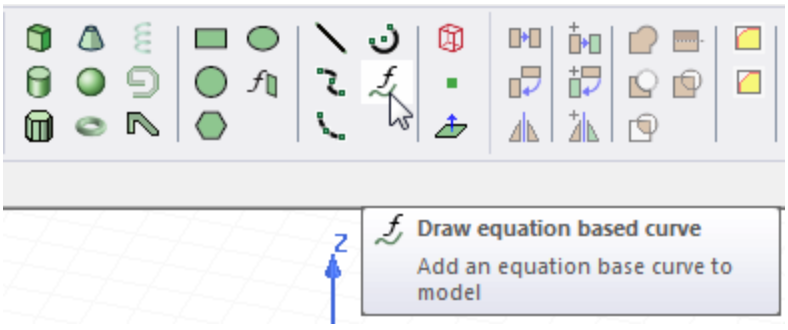
   > **Note:**
   >
   > When inserting straight segments or arcs, the insertion operation will terminate as soon as you complete the definition of a single new segment. For splines, you can insert a spline with any number of defining points, and the insertion operation terminates as soon as you terminate the spline.

5. Repeat steps 1 through 4 for each segment of the polyline before or after which you want to insert a new segment. The endpoint of the previous segment serves as the start point for the next segment.

Drawing an Equation-Based Curve

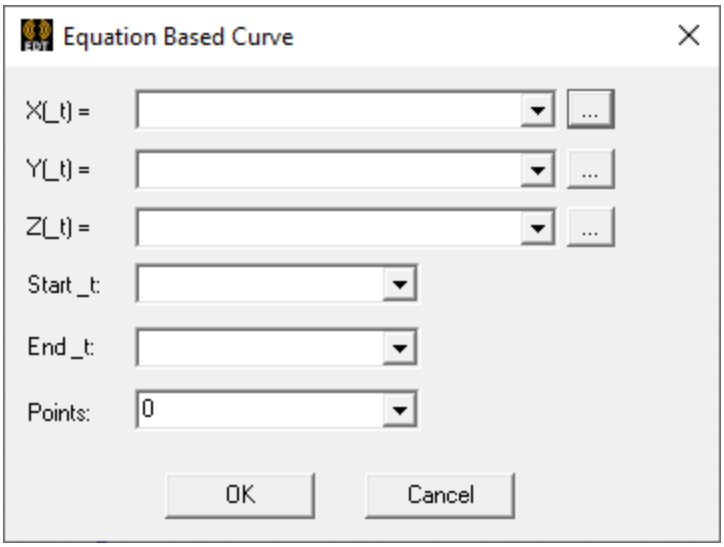
Any line that can be described by an equation in three dimensions can be drawn. Before you draw am equation-based curve, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw** > **Equation Based Curve**, or on the **Draw** ribbon tab, click the **Draw equation based curve** icon:



   The *Equation Based Curve* dialog box opens:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

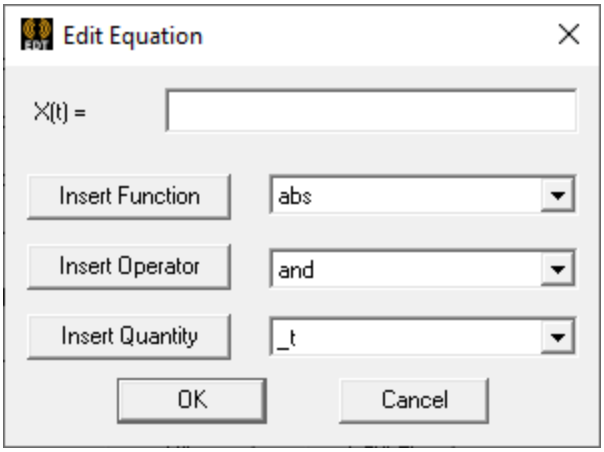


2. Type equations for **X(_t)**, **Y(_t)**, and **Z(_t)**

   You can also define an equation by doing the following:

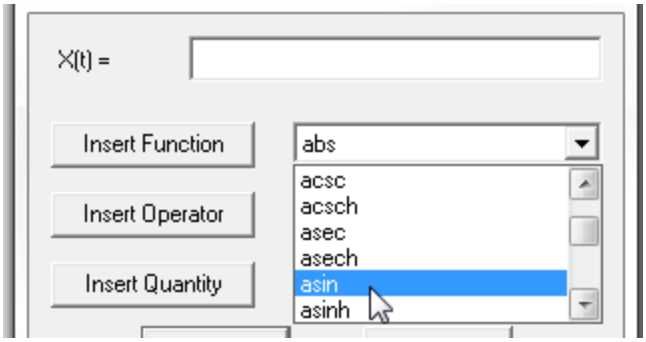
   a. Click an ellipsis button **[...]**.

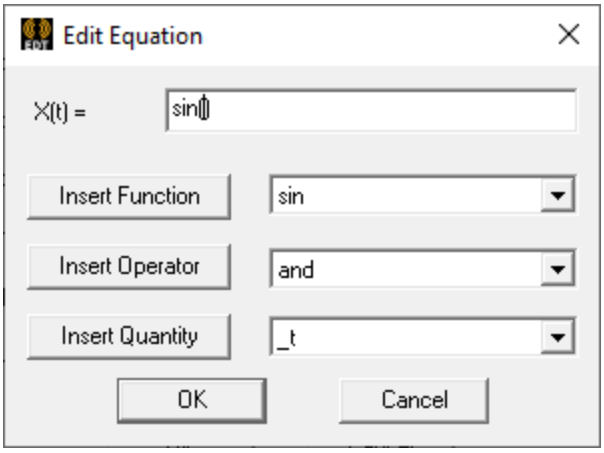
      The **Edit Equation** dialog box appears:



   b. Do one or more of the following to define the equation:
      - Type a numerical value, variable, or expression directly in the text box.
      - Build an equation for X(_t), Y(_t), or Z(_t) by specifying appropriate function, operator, and quantity to insert into the expression from the three drop-down menus:

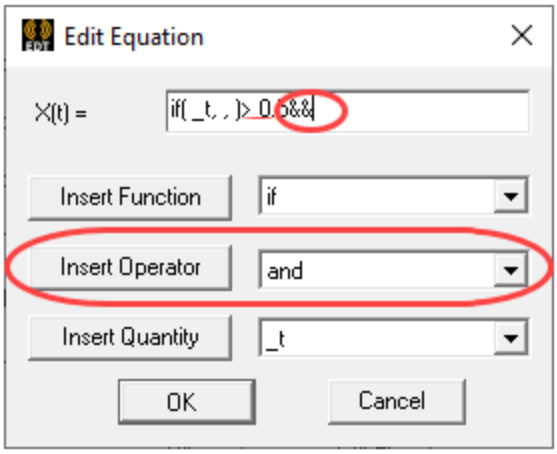
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

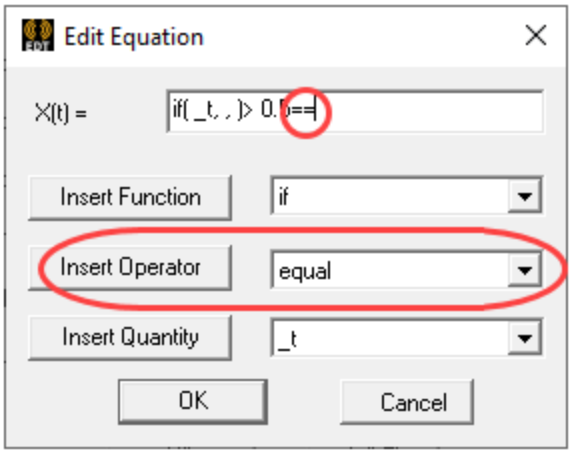




Click the corresponding **Insert Function, Insert Operator,** or **Insert Quantity** buttons to insert the selections into the equation text box at the current cursor location. The if(, ,) function has been added under **Insert Function**.

- The **Insert Operator** button inserts corresponding operator:





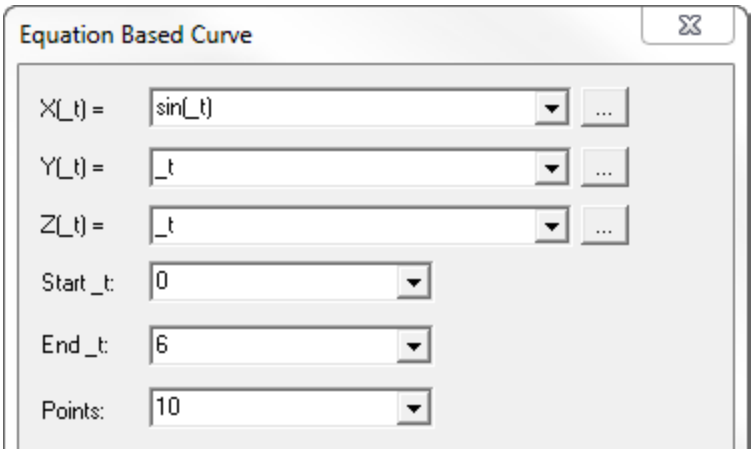
- Use a combination of the preceding two methods to build complex equations.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. When you are finished defining the equation, click **OK** to close the **Edit Equation** dialog box and return to the **Equation Based Curve** dialog box.

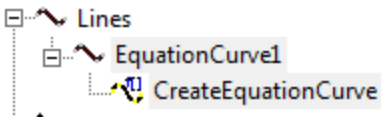
> **Note:**
>
> - The equations are taken to be meters. Use of units in equations is not properly supported.
> - While parsing expressions, equation based curves convert each variable separately to model units and assume that the resulting expression is in model units.
> - Equation based curves depend on the variable value library to correctly evaluate the units of expression.
> - Functions that do not support intrinsic (_t for curve, _u and _v for surface) will give an error if not supported.

3. Select a start value from the **Start_t** drop-down menu.
4. Select an end value from the **End_t** drop-down menu.
5. Type in or select the number of points in the curve from the **Points** drop-down menu.



6. Click **OK** on dialog to close it and create the curve.

   The curve appears in the modeler window, and the History tree shows the curve object and the command.



If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, in which you can modify the object's attributes by editing the Properties.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Properties listed as line attributes include Name, Orientation, whether a Model object, whether to Display Wireframe, Color, Transparency, and whether to Show Direction as arrows. The Show Direction property is most helpful to unambiguously show the line start orientation when plotting fields along a line.

Functions, Operators and Quantities for the Edit Equation Dialog Box

This table lists the functions, operators and Quantities for the Edit Equation dialog box.

Item	Definition
Functions	
abs	Absolute value of the simulation quantity which results in a number that is always positive.
acos	Arc cosine (the inverse function of a cosine).
acosh	Inverse hyperbolic arc cosine.
acot	Inverse cotangent
acoth	Inverse hyperbolic cotangent
acsc	Inverse cosecant
acsch	Inverse hyperbolic cosecant
asec	Inverse secant
asech	Inverse hyperbolic secant
asin	Arc sine (the inverse function of sine)
asinh	Inverse hyperbolic sine
atan	Arc tangent (the inverse function of a tan)
atanh	Inverse hyperbolic tan
cos	Cosine
cosh	Hyperbolic cosine
cot	Cotangent
coth	Hyperbolic cotangent
csc	Cosecant
csch	Hyperbolic cosecant
ln	Natural logarithm.
log	Natural logarithm (same as ln).
sin	Sine.
sinh	Hyperbolic sine.
sqrt	Square root of the selected simulation quantity.
tan	Tangent.

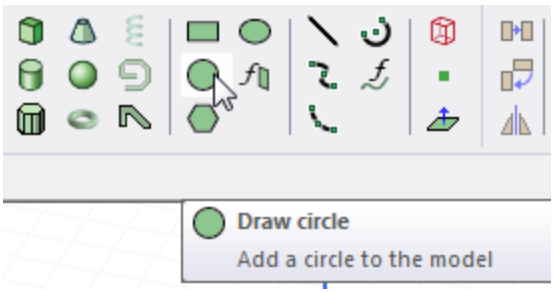
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Item	Definition
tanh	Hyperbolic tangent.
Operators	
and	
cross	
division	
dot	
equal	
exp	
exponent	
greater_than	
greater_than_ or_equal	
if	if(, ,)
not	
not_equal	
or	
Quantities	
_t	For equation based lines
_u,_v	For equation based surfaces.
PI	

Drawing a Circle

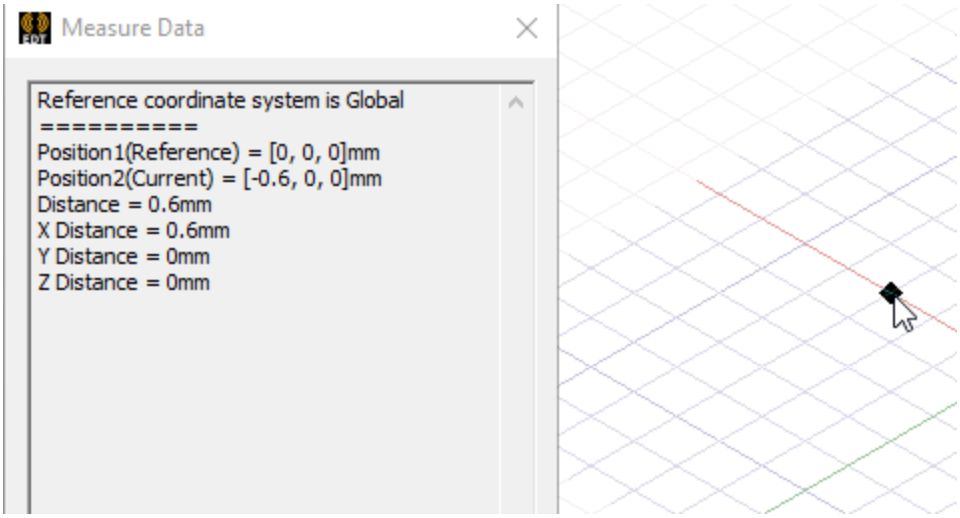
Draw a circle by selecting a center point and a radius. Circles are drawn as true surfaces in the modeler. Before you draw a circle, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw** > **Circle** or on the **Draw** ribbon tab, click the **Draw circle** icon:



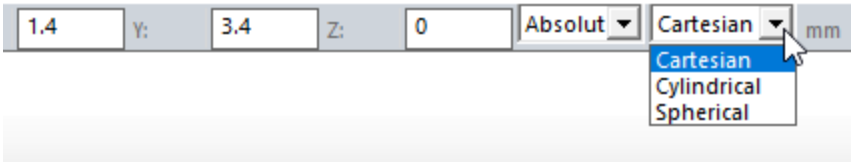
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select the center point of the circle in one of the following ways:
   - Click the point.



   - Type the point's coordinates in the text boxes in the status bar.

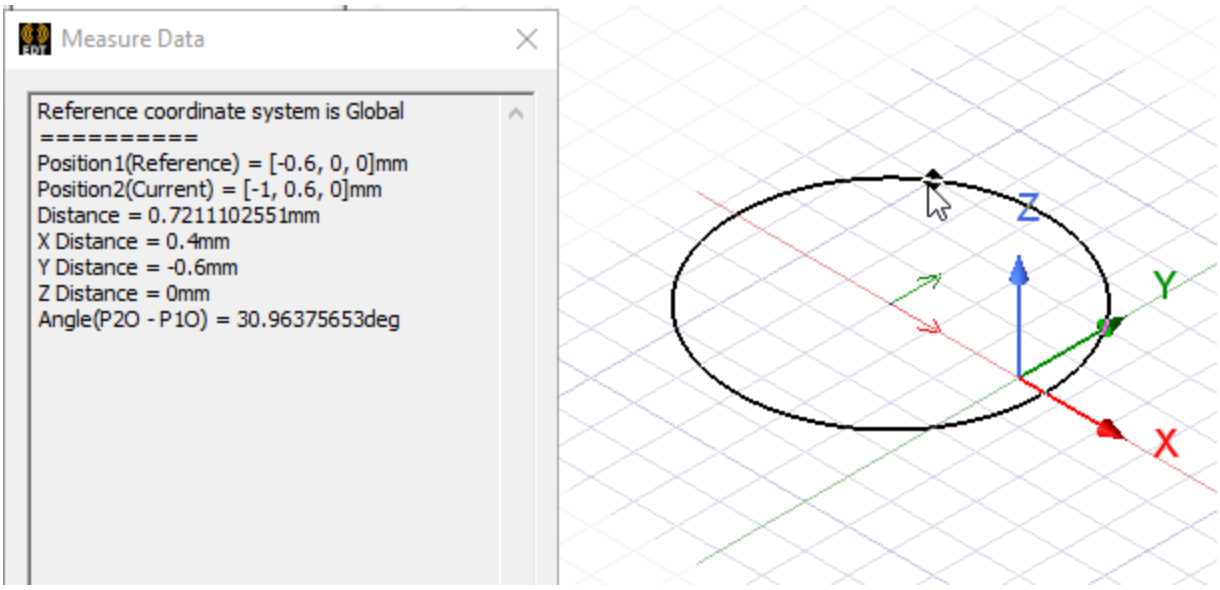
     The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop-down menu options to specify the circle in Cartesian, Cylindrical, or Spherical coordinates.



3. Specify the radius by selecting a point on the circle's circumference in one of the following ways:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

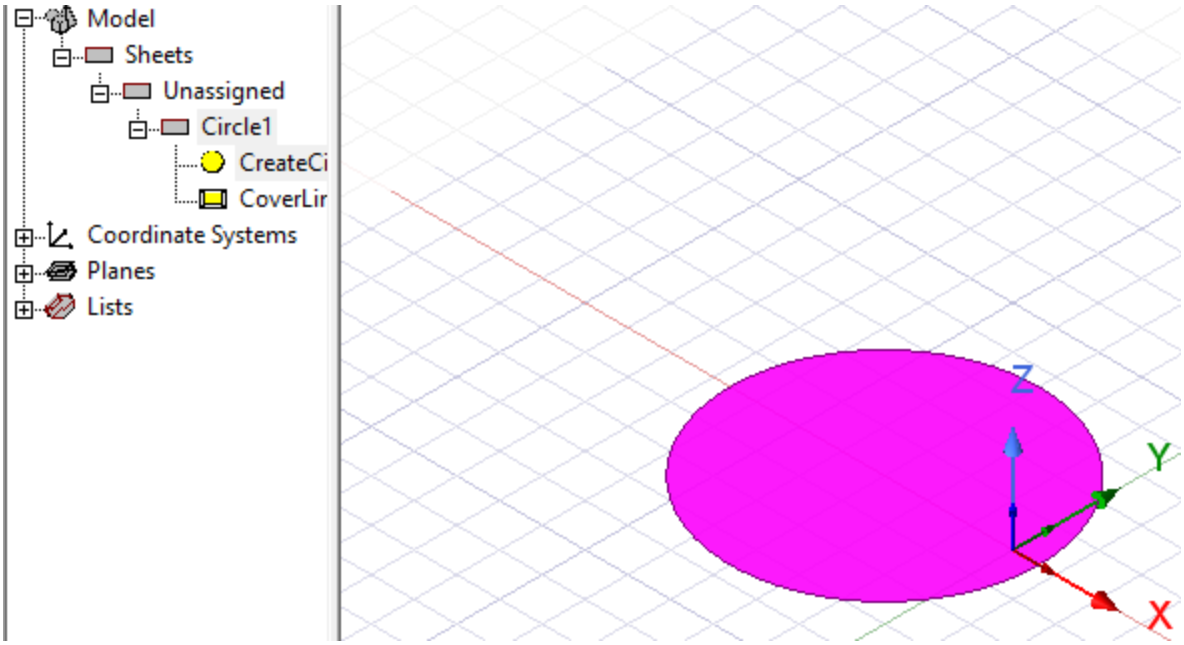
- Click the point.



- Type the coordinates of the point relative to the center point in the **dX**, **dY**, and **dZ** boxes, where **d** is the distance from the previously selected point.

4. Double-click or right-click and select Done from the shortcut menu. If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. If the **Automatically cover closed polyline** option is selected in the **Modeler Options** window, the circle will be covered, resulting in a 2D sheet object. Otherwise it will be a closed 1D polyline object. The new circle appears in the Modeler window and History tree.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

The 3D Modeler permits drawing true curved objects. However, the solution is obtained using a tetrahedral mesh, which conforms to the true surface only within the limits identified by certain mesh settings. The modeler has default settings for this conformance, which is a reasonable trade-off between solution speed and solution quality for most objects but may not be ideal for all such objects. High-aspect ratio curves structures (such as helices with narrow and curved cross-sections) may benefit from user control of the faceting values. For details about these commands, see the following:

Modifying Surface Approximation Settings, and related sections:

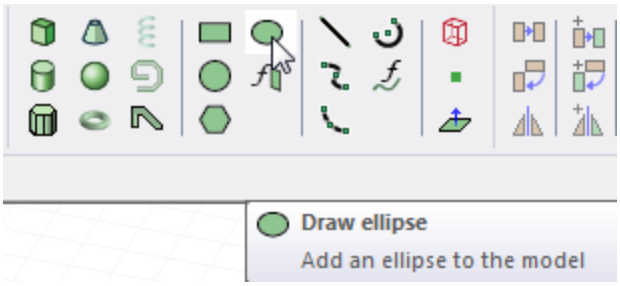
Rectilinear Elements and Curvilinear Elements and Guidelines for Modifying Surface Approximations.

Drawing an Ellipse

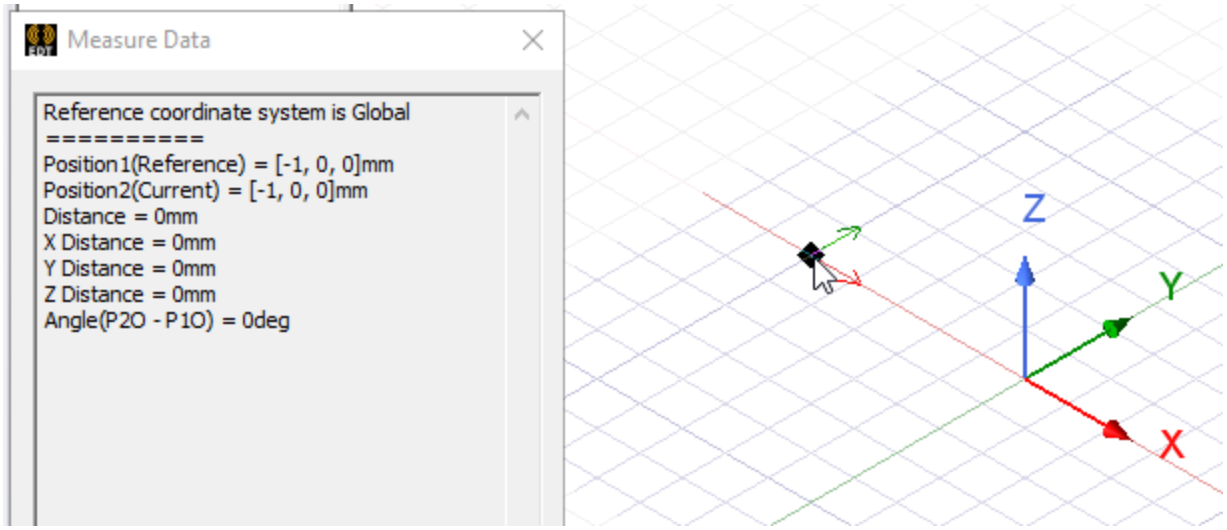
Draw an ellipse by specifying a center point, base radius, and secondary radius. Before you draw an ellipse, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

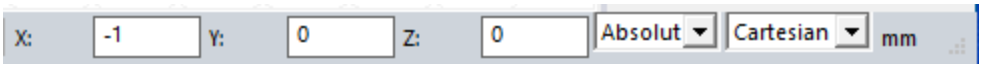
1. From the menu bar, click **Draw** > **Ellipse**, or on the **Draw** ribbon tab, click the **Draw ellipse** icon:



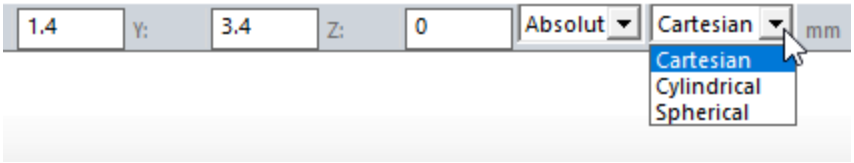
2. Select the center point of the ellipse in one of the following ways:
   - Click the point.



   - Type the point's coordinates in the text boxes in the status bar.



     The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop-down menu options to specify the ellipse in Cartesian, Cylindrical, or Spherical coordinates.

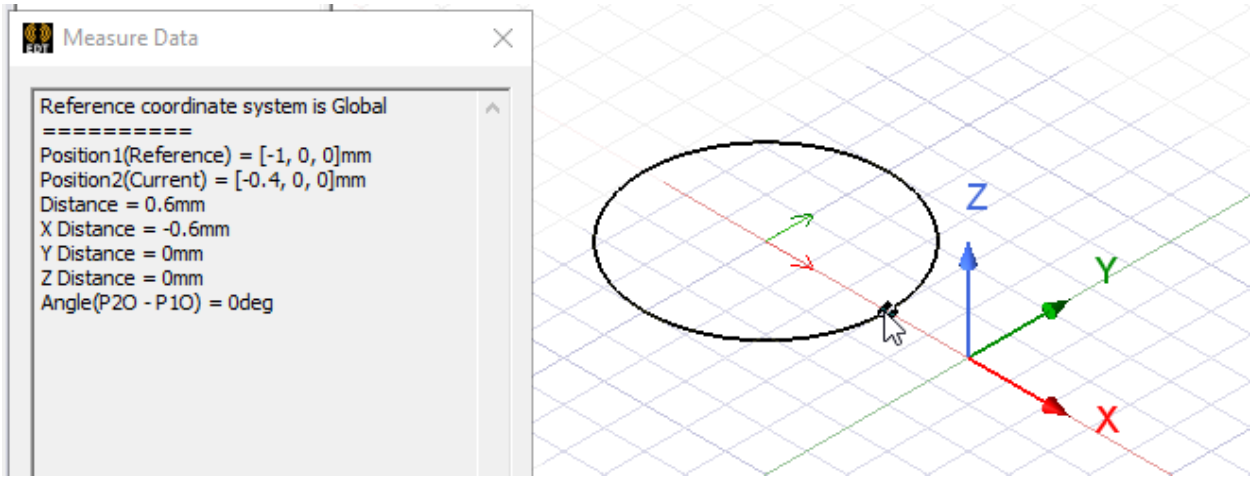


3. Specify the base radius of the ellipse. If the current drawing plane is xy, then x is the base radius direction. If the drawing plane is yz, then y is the base radius direction. If the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

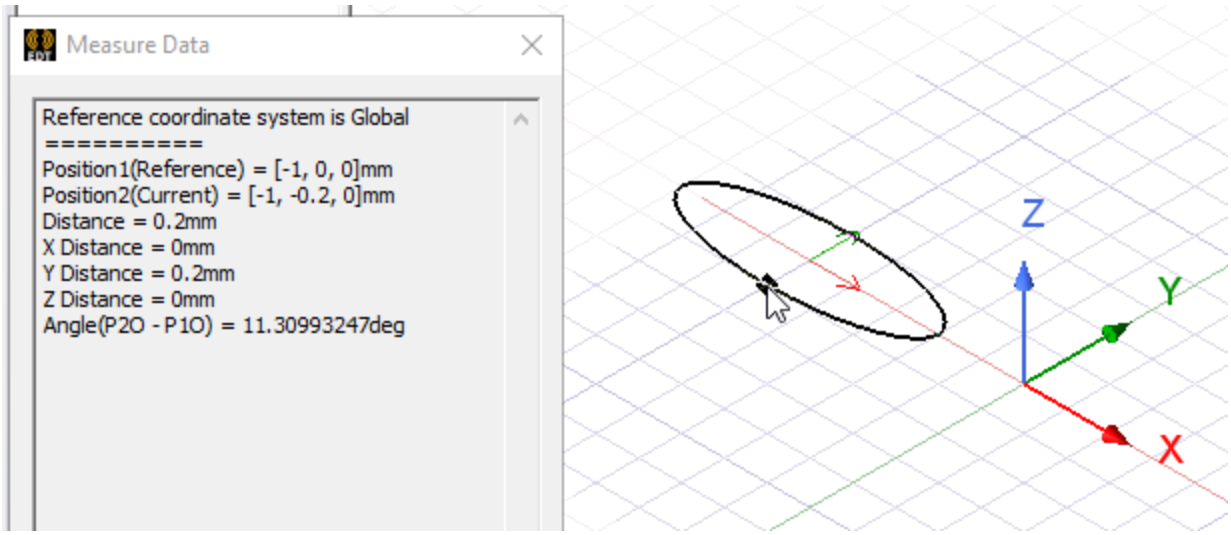
drawing plane is xz, then z is the base radius direction. Select the point in one of the following ways:

- Click the point. Ansys Electronics Desktop constrains mouse movement to the base radius direction.



- Type the coordinates of a point relative to the center point in the **dX, dY,** or **dZ** box, where **d** is the distance from the previously selected point.

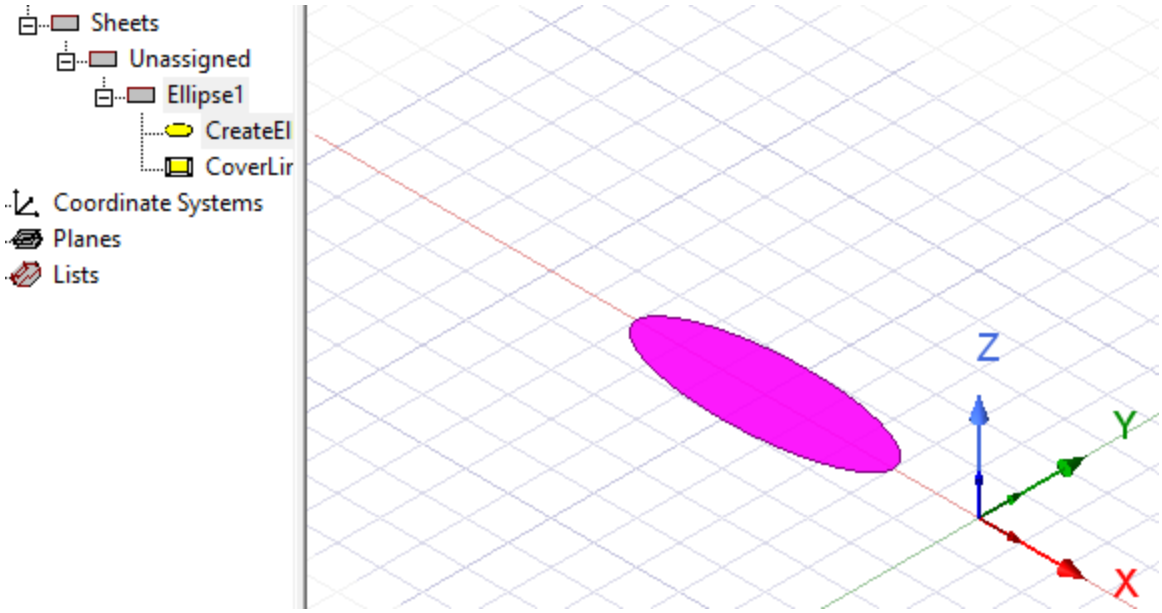
4. Specify the secondary radius of the ellipse. Select the point in one of the following ways:
   - Click the point. Ansys Electronics Desktop constrains mouse movement to a point on the plane orthogonal to the base radius direction.



   - Type the coordinates of a point relative to the center point in the **dXm dY,** or **dZ** box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Double click, or right click and select **Done** from the short cut menu. The ellipse appears in the Modeler window and in the History Tree.



6. If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. The **Ratio** value represents the aspect ratio of the secondary radius to the base radius.

   If the **Automatically cover closed polyline** option is selected in the **Modeler Options** window, the ellipse will be covered, resulting in a 2D sheet object. Otherwise it will be a closed 1D polyline object.

If the base radius is larger than the secondary radius, the ellipse's longer axis will lie along the default base radius direction. If the secondary radius is larger than the base radius, the ellipse's longer axis will lie perpendicular to the default base radius direction. To create an ellipse with an arbitrary orientation, rotate or move the ellipse after drawing it.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The 3D Modeler permits drawing true curved objects. However, the solution is obtained using a tetrahedral mesh, which conforms to the true surface only within the limits identified by certain mesh settings. The modeler has default settings for this conformance, which is a reasonable trade-off between solution speed and solution quality for most objects but may not be ideal for all such objects. High-aspect ratio curves structures (such as helices with narrow and curved cross-sections) may benefit from user control of the faceting values. For details about these commands, see the following:

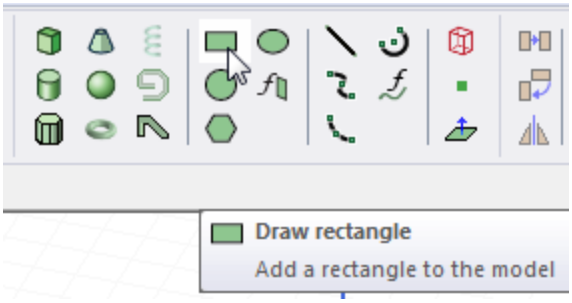
Modifying Surface Approximation Settings, and related sections:

Rectilinear Elements and Curvilinear Elements and Guidelines for Modifying Surface Approximations.

Drawing a Rectangle

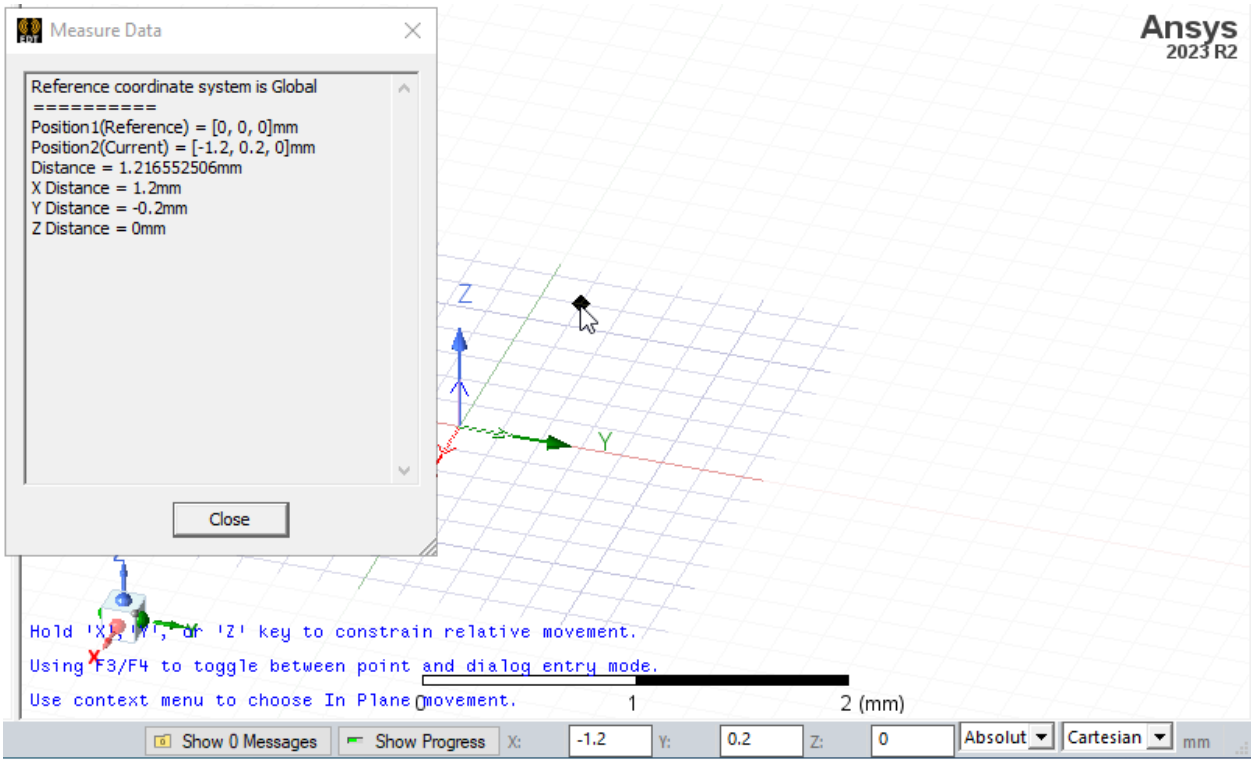
Draw a rectangle (or square) by selecting two diagonally opposite corners. Before you draw a rectangle, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw** > **Rectangle**, or on the **Draw** ribbon tab, click the **Draw rectangle** icon:

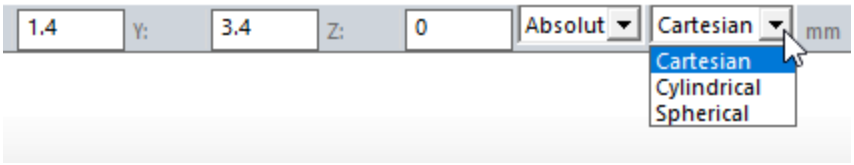


2. Select the first diagonal corner in one of the following ways:
   - Click the point. The Measure Data dialog box shows the coordinates and other information.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



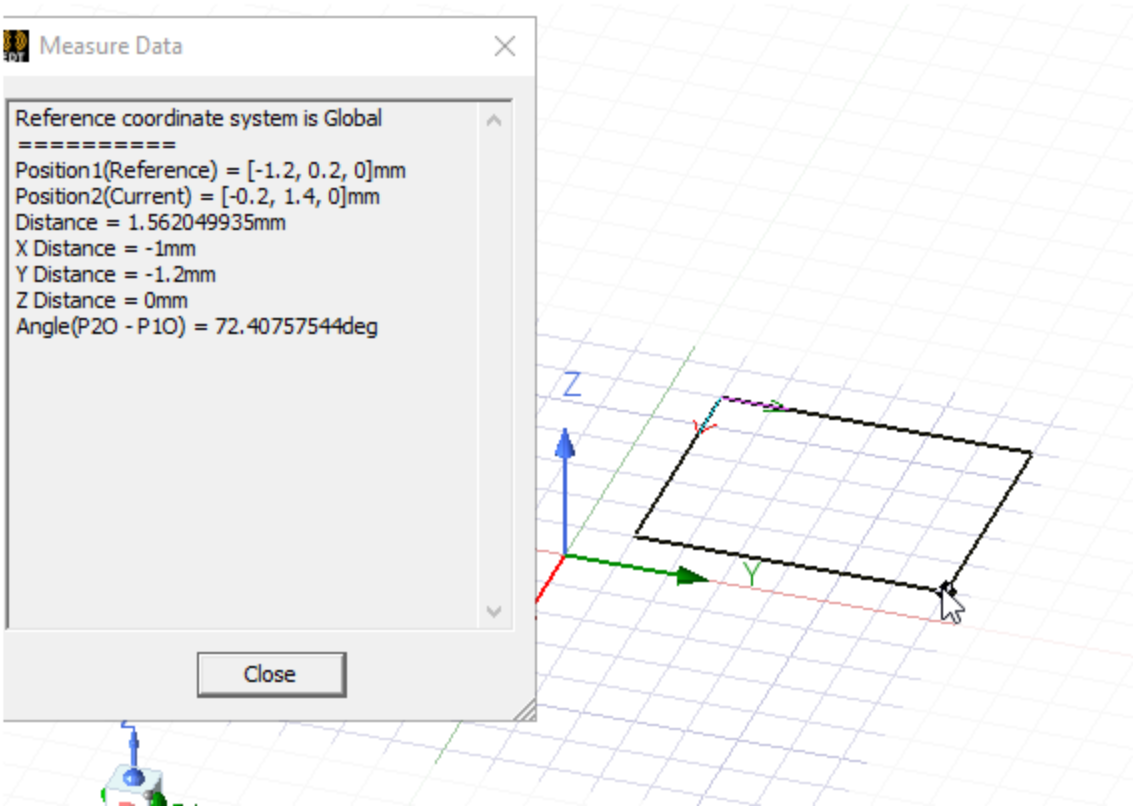
- Type the point's coordinates in the text boxes in the status bar. The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop-down menu options to specify the cylinder in Cartesian, Cylindrical, or Spherical coordinates.



To delete the selected point and start over, press **Esc** or click **Escape Draw Mode** on the shortcut menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select the second corner of the rectangle in one of the following ways:
   - Click the point.

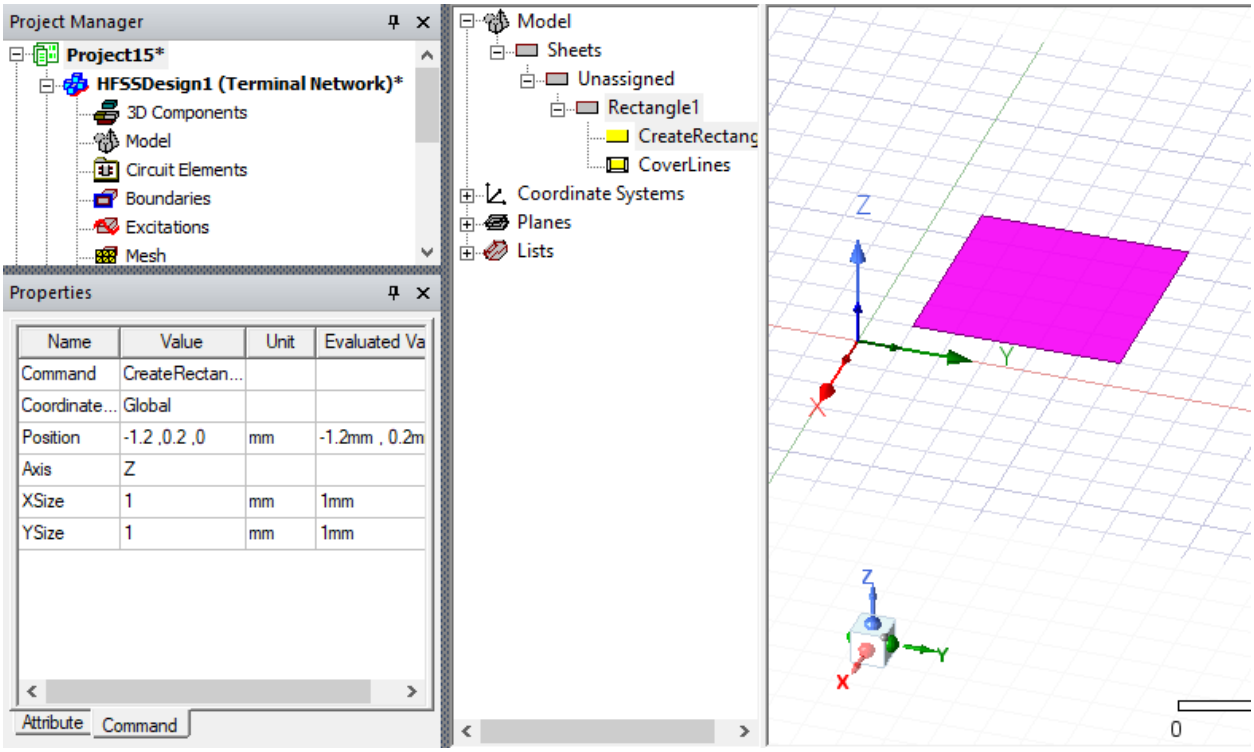


   - Type the coordinates of the point relative to the first diagonal corner in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

   If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. You can also use the docked Properties window.

4. Click **OK**. If the **Automatically cover closed polyline** option is selected in the **Modeler Options** window, the rectangle will be covered, resulting in a 2D sheet object, as shown in the History tree in the figure below. Otherwise it will be a closed 1D polyline object.

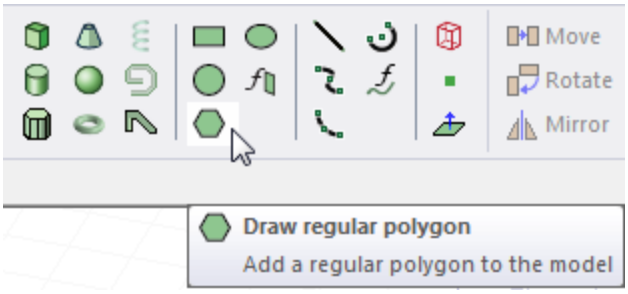
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Drawing a Regular Polygon

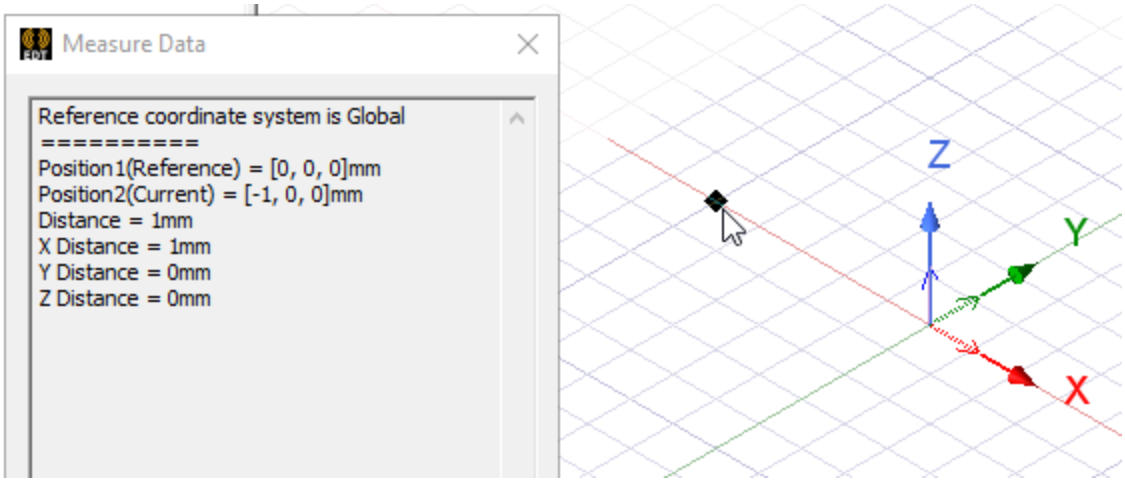
A regular polygon is a 2D object with three or more equal sides. Regular polygons are useful for drawing faceted 2D objects. Before you draw a regular polygon, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. Click **Draw** > **Regular Polygon**, or on the **Draw** ribbon tab, click the **Draw regular polygon** icon:



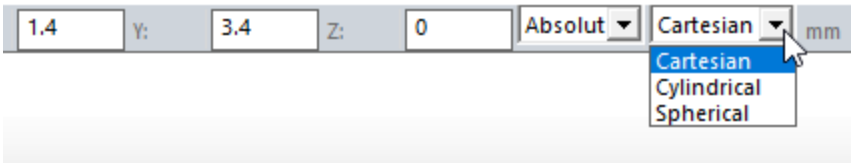
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select the center point of the polygon in one of the following ways:
   - Click the point.



   - Type the point's coordinates in the text boxes in the status bar.

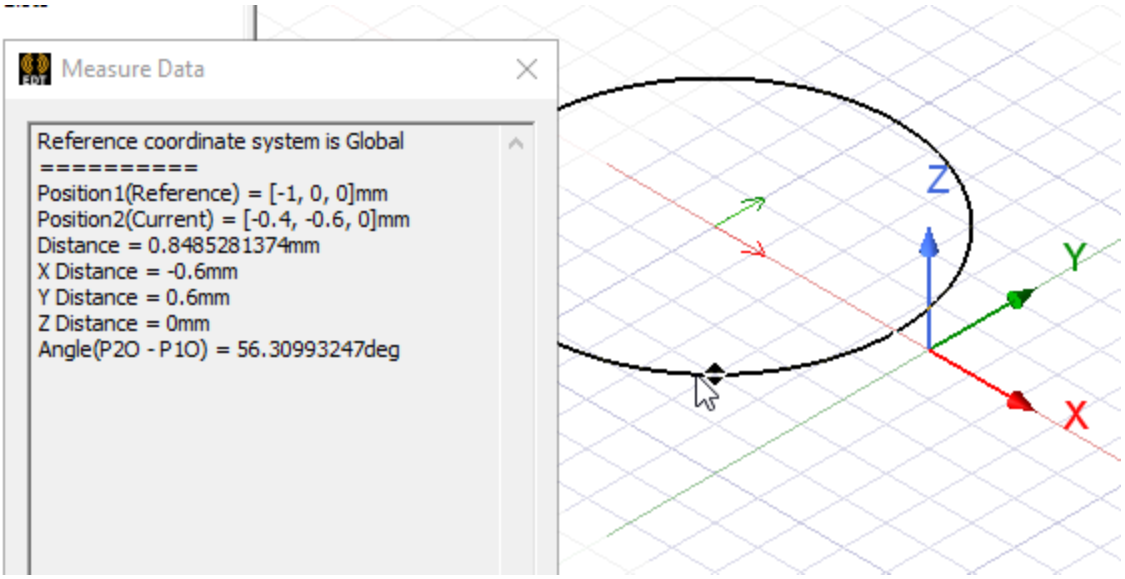
     The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop-down menu options to specify the regular polygon in Cartesian, Cylindrical, or Spherical coordinates.



3. Specify the polygon's radius, the distance from the center point to one of the polygon's vertices, in one of the following ways:

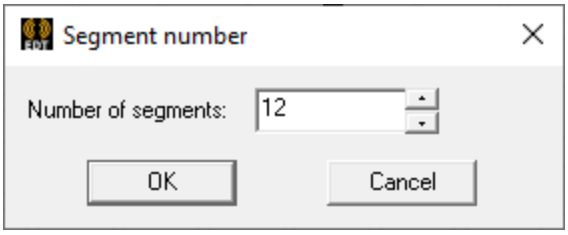
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click the point.



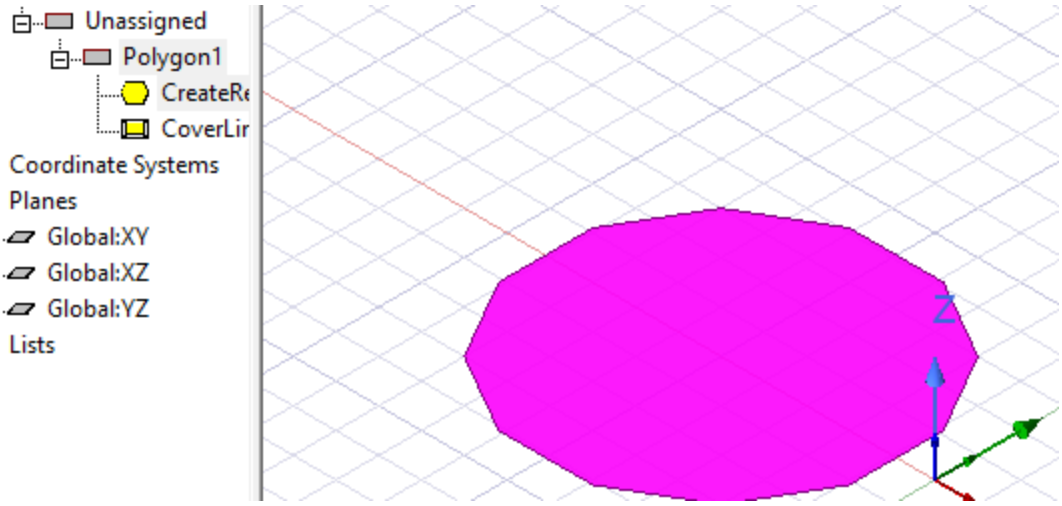
- Type the coordinates of the point relative to the center point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

4. In the **Segment number** dialog box, enter the **Number of segments** in the polygon, and then click **OK**.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Click OK. The polygon appears in the Modeler window and in the History tree.



5. If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. If you have edited the Properties dialog box, click **OK**.

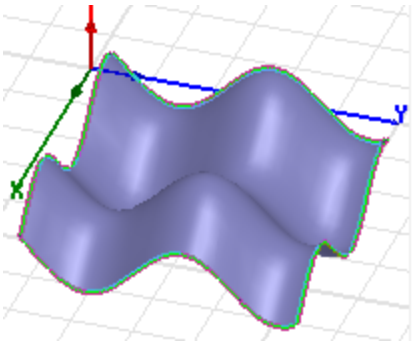
> **Note:**
>
> The radius is measured from the center point to a corner of the polygon, or the intersection of two edges. It is *not* measured from the center point to the midpoint of an edge.

If the **Automatically cover closed polyline** option is selected in the **Modeler Options** window, the polygon will be covered, resulting in a 2D sheet object. Otherwise it will be a closed 1D polyline object.

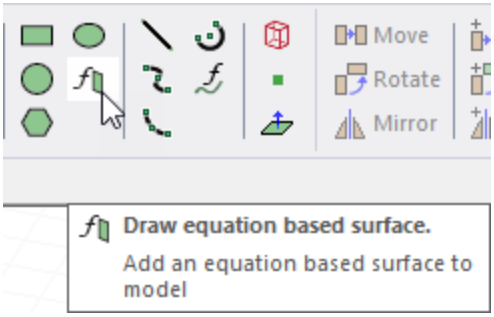
Drawing an Equation-Based Surface

Any surface that can be described by an equation in three dimensions can be drawn. Before you draw an equation-based surface, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

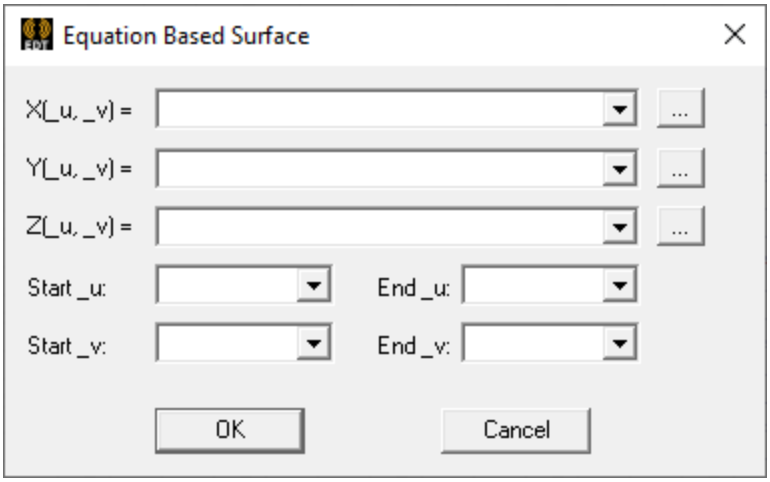
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



1. From the menu bar, click **Draw** > **Equation Based Surface**, or on the **Draw** ribbon tab, click the **Draw equation based surface** icon:



The **Equation Based Surface** dialog box opens.



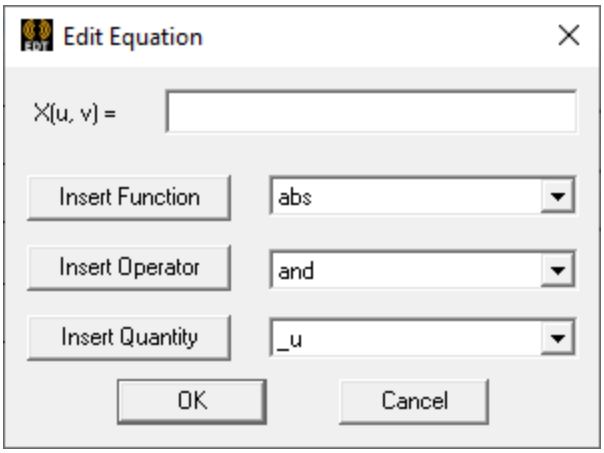
2. Type equations for **X(_u, _v)**, **Y(_u, _v)**, and **Z(_u, _v)**.

   You can also define an equation by doing the following:

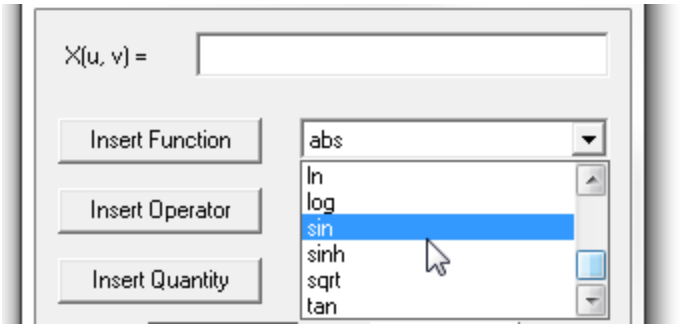
   a. Click the ellipsis button (**...**).

      The **Edit Equation** dialog box appears:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



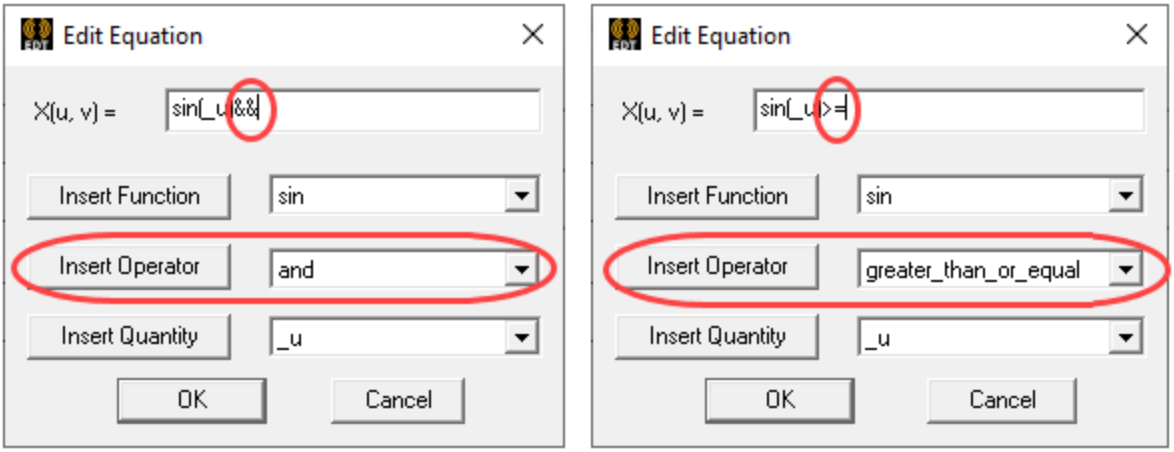
b. Do one or more of the following to define the equation:
   - Type a numerical value, variable, or expression directly in the text box
   - Build an equation for X(u, v), Y(u, v), or Z(u, v), by selecting the function, operator, and quantity to insert into the equation from the three drop-down menus:



     Click the corresponding **Insert Function, Insert Operator,** or **Insert Quantity** buttons to insert the selections into the equation text box at the current cursor location. The if(, ,) function has been added under **Insert Function**.

   - The **Insert Operator** button inserts corresponding operator:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

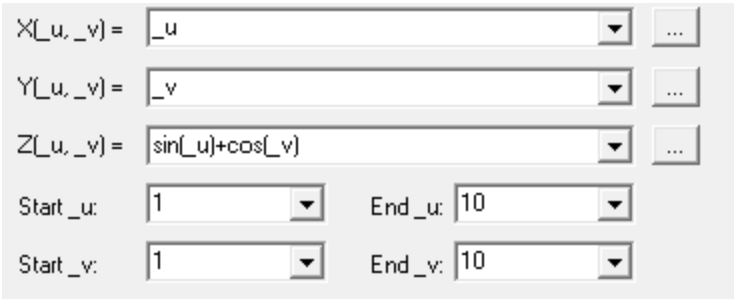


- You can combine the preceding two methods to build complex equations.

c. When you are finished defining the equation, click **OK** to close the **Edit Equation** dialog box and return to the **Equation Based Surface** dialog box.
   - The equations are taken to be meters. Use of units in equations is not properly supported..
   - While parsing expressions, equation based curves convert each variable separately to model units and assume that the resulting expression is in model units.
   - Equation based curves depend on the variable value library to correctly evaluate the units of expression.
   - Functions that do not support intrinsic (_t for curve, _u and _v for surface) will give an error if not supported.

3. Specify start values for the **Start_u** and **Start_v** fields.

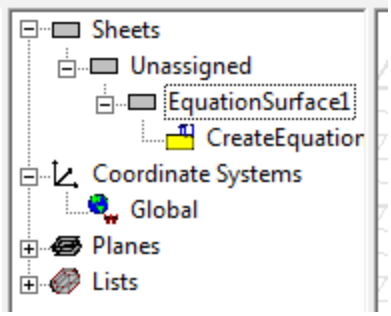
   Select end values for **End_u** and **End_v** fields.



4. Click **OK** on the **Properties** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The surface is drawn in the Modeler window. The History tree contains the Equation Surface object and the command properties.

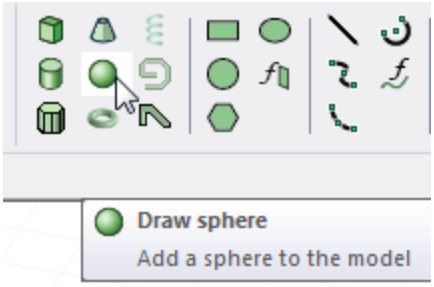


If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's attributes.

Drawing a Sphere

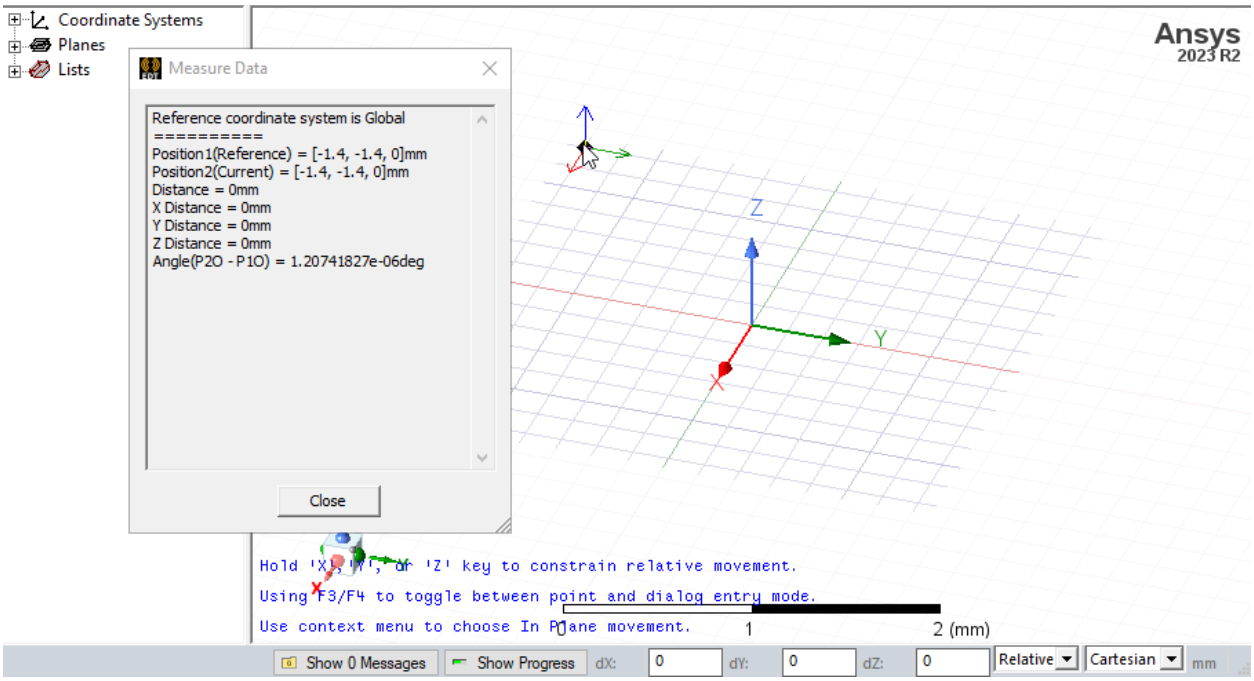
Draw a sphere, a 3D circle, by selecting a center point and a radius. Spheres are drawn as true surfaces in the modeler. Before you draw a sphere, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw** > **Sphere**, or on the **Draw** ribbon tab, click the **Draw sphere** icon:



2. Select the center point of the sphere in one of the following ways:
   - Click the point. The Measure Data dialog box shows the coordinates and other information.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

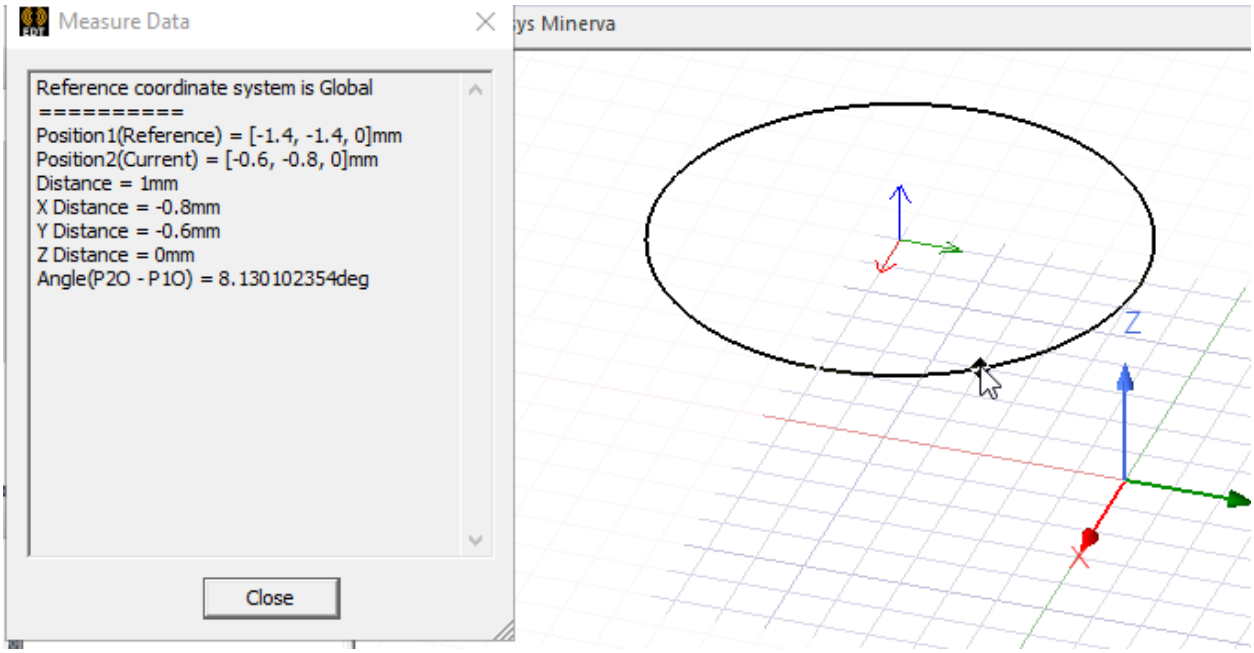


- Type the point's coordinates in the text boxes in the status bar. The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop-down menu options to specify the sphere in Cartesian, Cylindrical, or Spherical coordinates.

3. Specify the radius by selecting a point on the sphere's circumference in one of the following ways:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

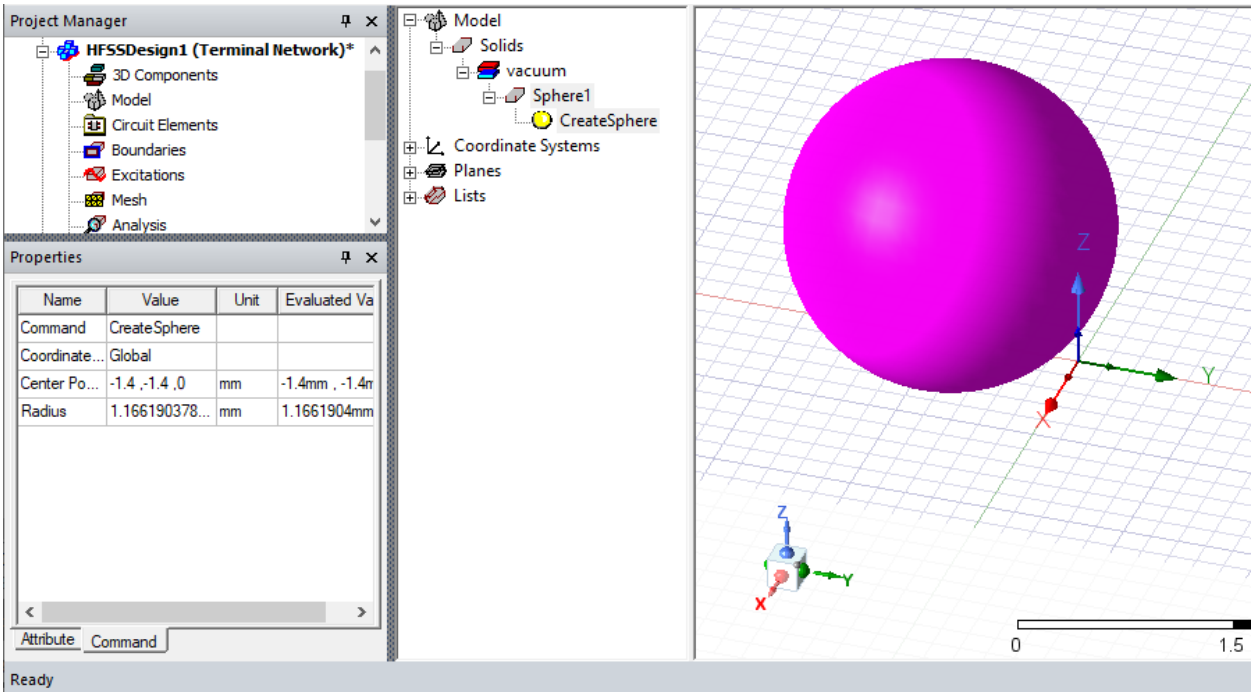
- Click the point



- Type the coordinates of the point relative to the center point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. You can also use the docked Properties window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Click **OK**.

Note:

The 3D Modeler permits drawing true curved objects. However, the solution is obtained using a tetrahedral mesh, which conforms to the true surface only within the limits identified by certain mesh settings. The modeler has default settings for this conformance, which is a reasonable trade-off between solution speed and solution quality for most objects but may not be ideal for all such objects. High-aspect ratio curves structures (such as helices with narrow and curved cross-sections) may benefit from user control of the faceting values. For details about these commands, see the following:

Modifying Surface Approximation Settings, and related sections:

Rectilinear Elements and Curvilinear Elements and Guidelines for Modifying Surface Approximations.

Related Topics

Modifying Objects

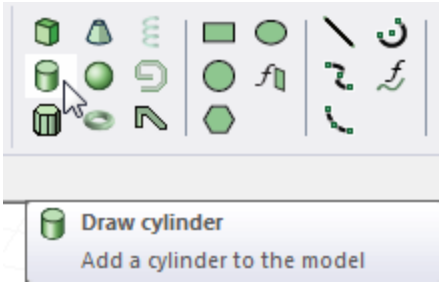
Modifying Surface Approximation Settings

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

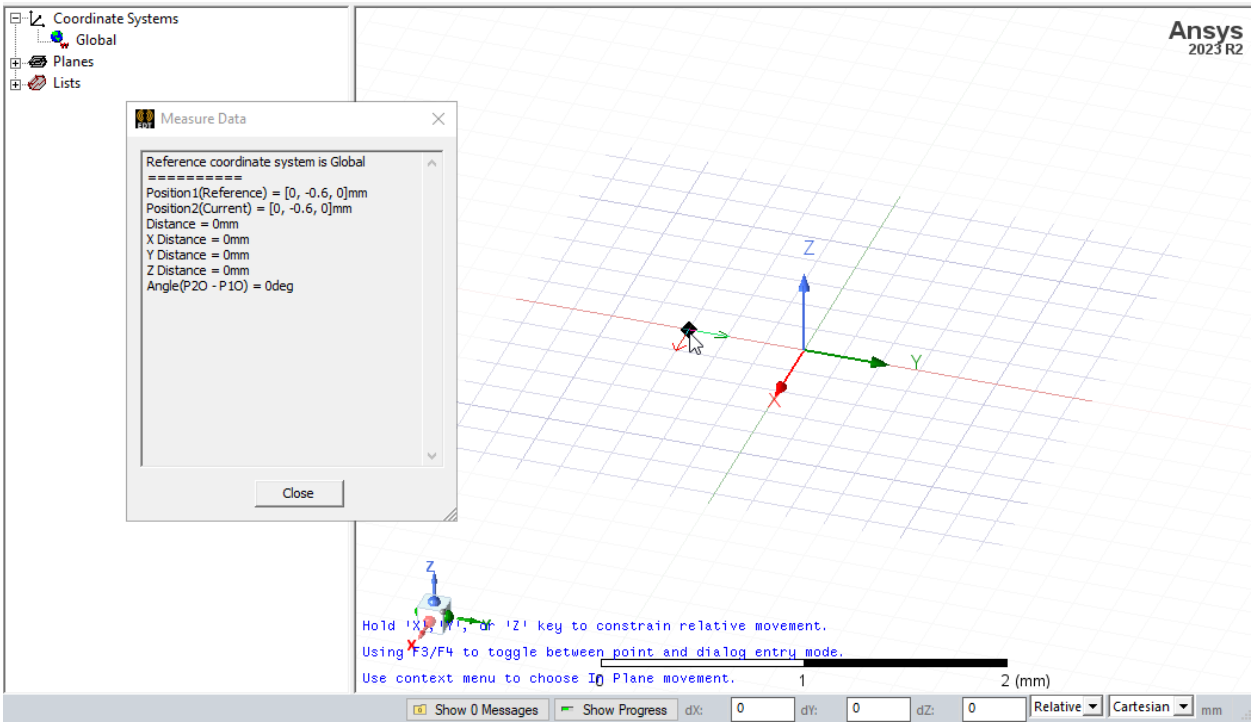
Drawing a Cylinder

Draw a cylinder by selecting a center point, radius, and height. Cylinders are drawn as true surfaces in the modeler. Before you draw a cylinder, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw** > **Cylinder**, or on the **Draw** ribbon tab, click the **Draw cylinder** icon:



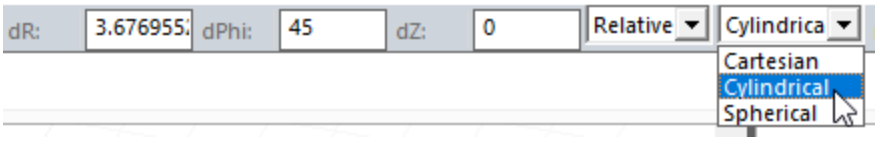
2. Select the center point of the cylinder's base circle in one of the following ways:
   - Click the point. The Measure Data dialog box shows the coordinates and other information.



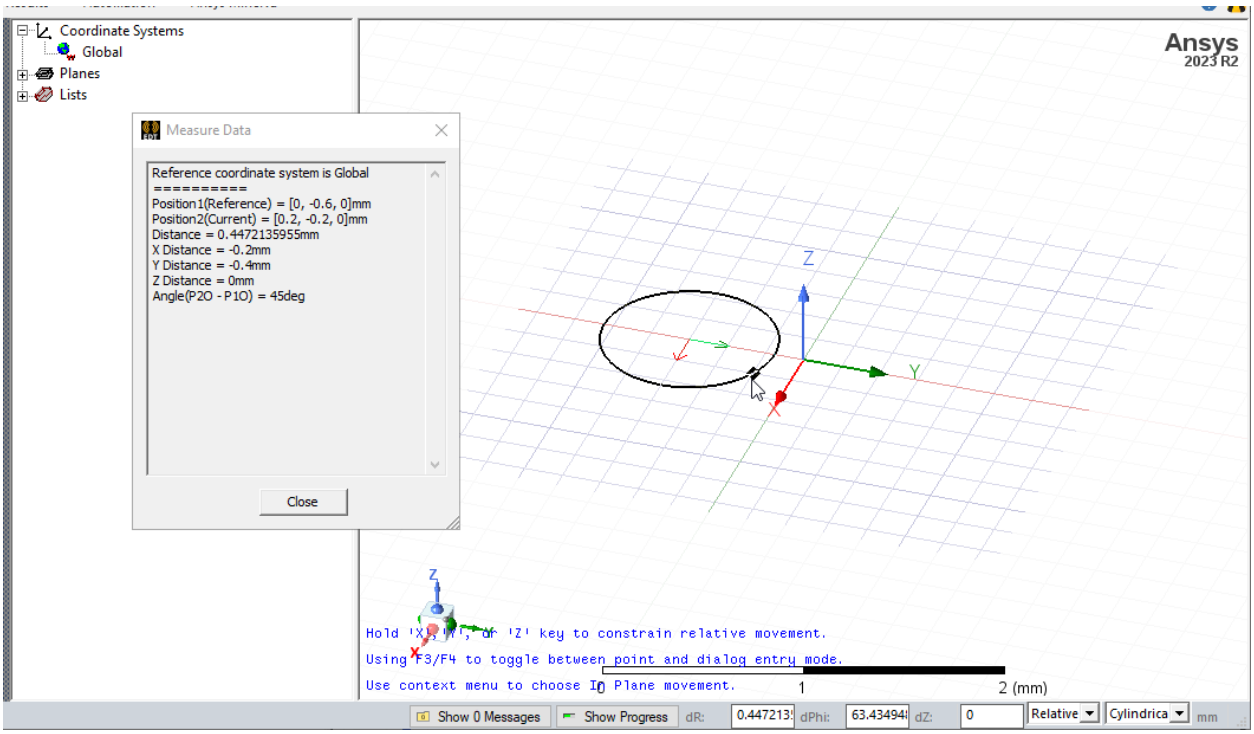
   - Type the point's coordinates in the text boxes in the status bar. The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop-down menu options to specify the cylinder in Cartesian, Cylindrical, or

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Spherical coordinates.

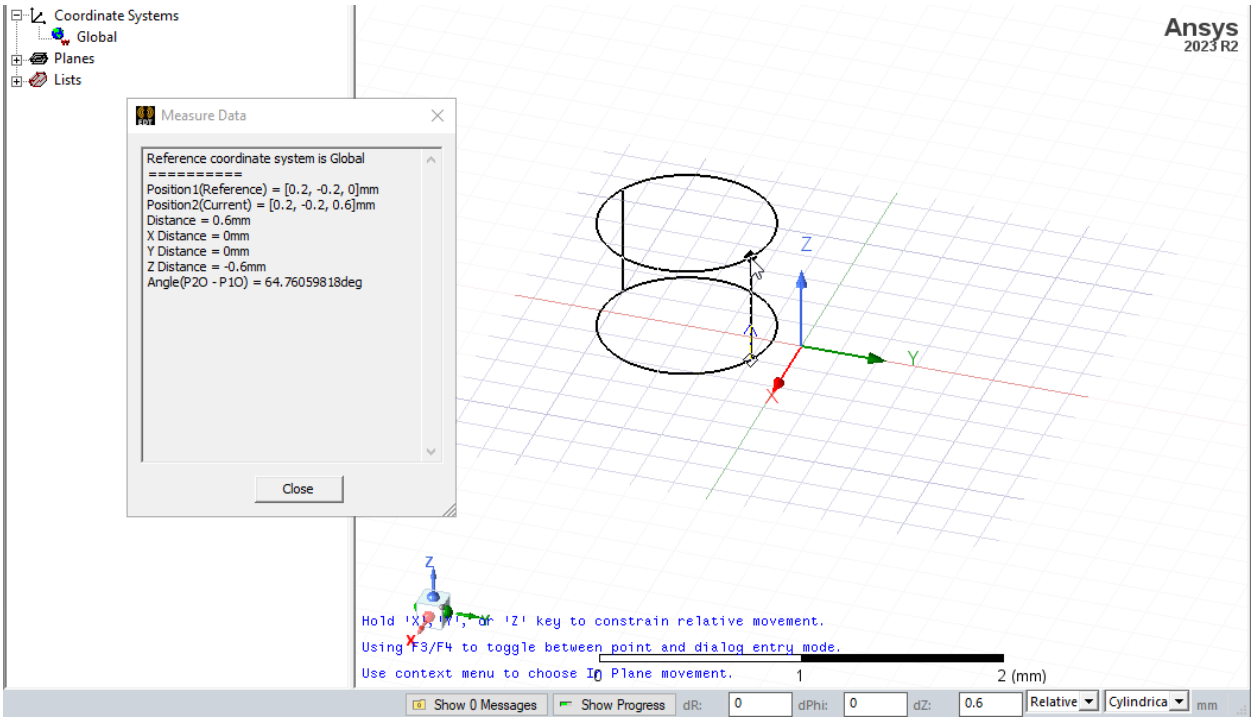


3. Specify the radius by selecting a point on the base circle's circumference in one of the following ways:
   - Click the point.



   - Type the coordinates of the point relative to the center point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.
4. Specify the cylinder's height by selecting a point on the axis perpendicular to the base circle's plane. Select the point by clicking the point or typing the coordinates in the text box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

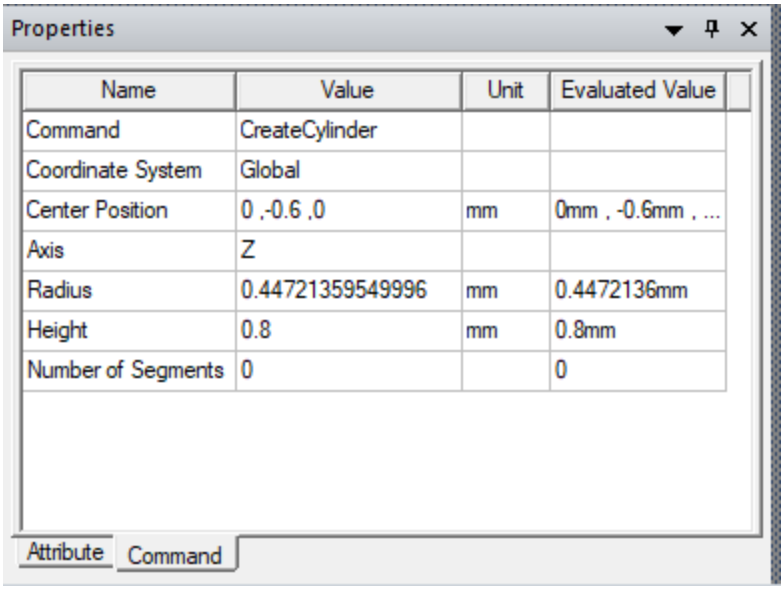


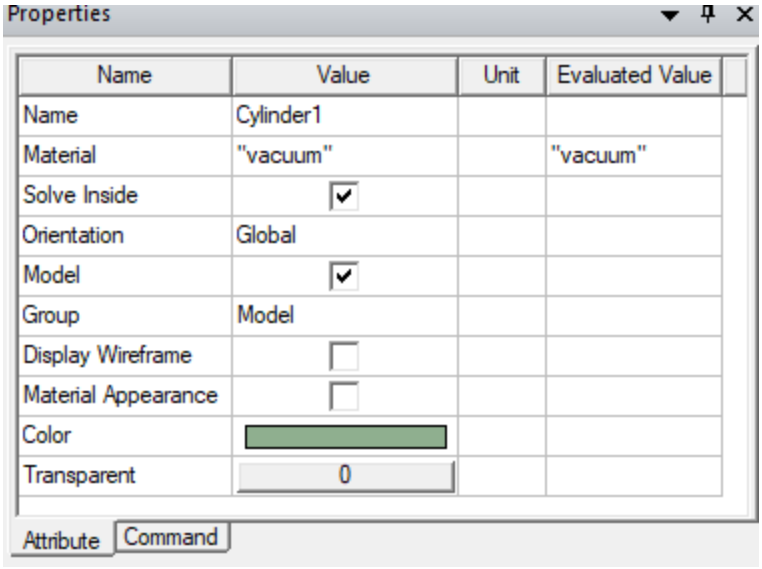
> **Note:**
>
> If you create a cylinder with a height of zero, Ansys Electronics Desktop draws a circular sheet object.

Double click, or right click and select **Done** from the short cut menu. If the Modeler option for editing properties of new primitives is checked, the *Properties* dialog box appears, enabling you to modify the object's Command and Attribute properties. You can also use

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

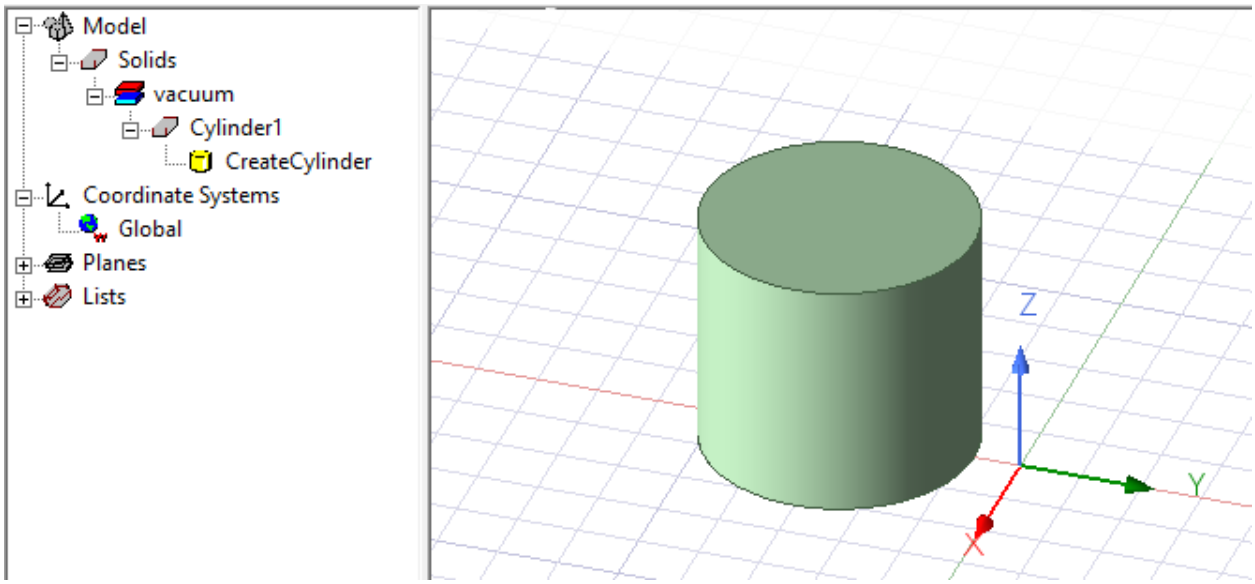
the docked Properties window.





5. If you used the Properties window, click **OK**. The cylinder appears in the Modeler window and is listed in the History tree.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

The 3D Modeler permits drawing true curved objects. However, the solution is obtained using a tetrahedral mesh, which conforms to the true surface only within the limits identified by certain mesh settings. The modeler has default settings for this conformance, which is a reasonable trade-off between solution speed and solution quality for most objects but may not be ideal for all such objects. High-aspect ratio curves structures (such as helices with narrow and curved cross-sections) may benefit from user control of the faceting values. For details about these commands, see the following:

Modifying Surface Approximation Settings, and related sections:

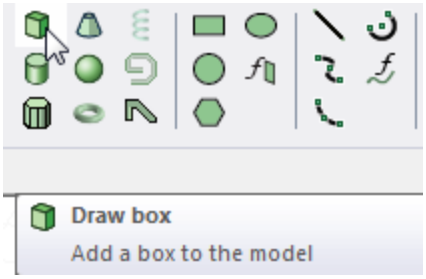
Rectilinear Elements and Curvilinear Elements and Guidelines for Modifying Surface Approximations.

Drawing a Box

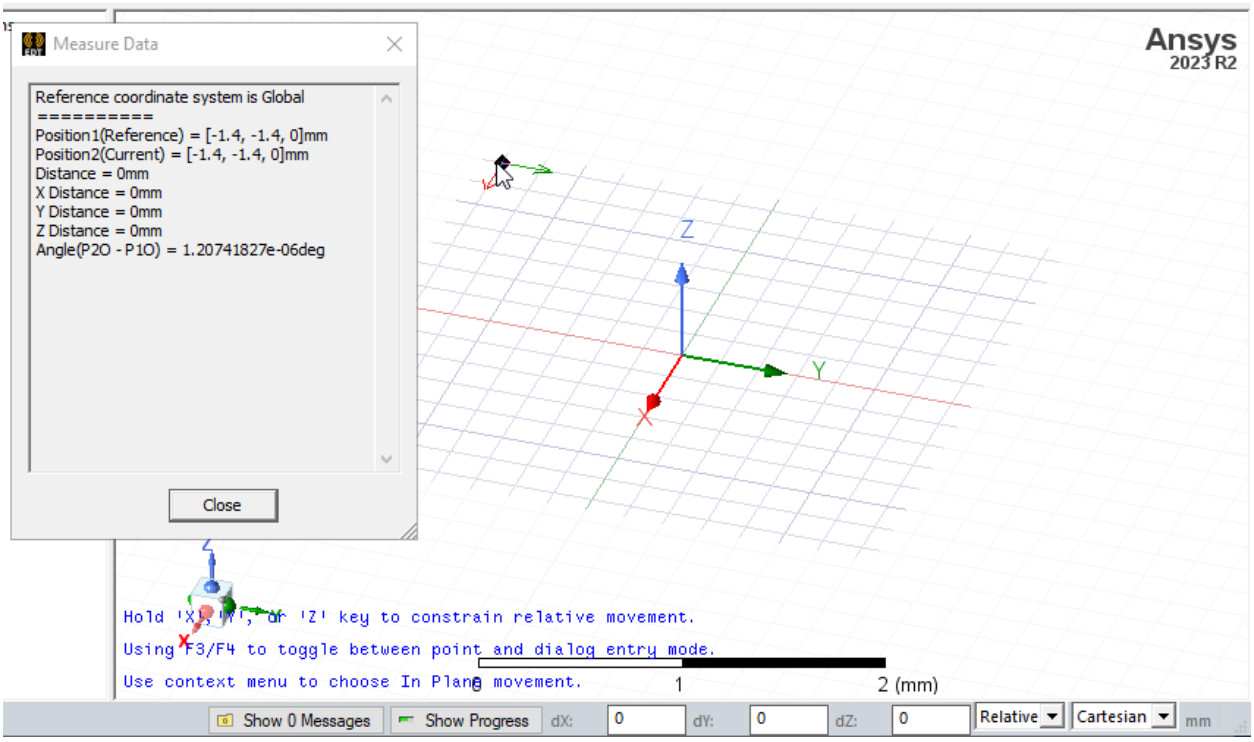
Draw a box by selecting two diagonally opposite corners of the base rectangle, then specifying the height. Before you draw a box, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw** > **Box**, or on the **Draw** tab of the ribbon, click the **Draw box** icon:

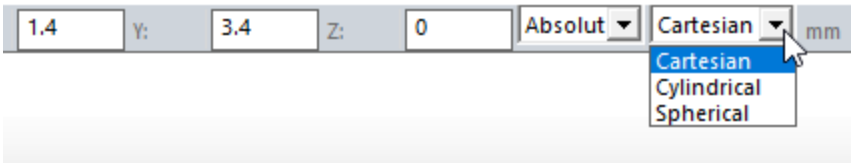
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select the first diagonal corner of the base rectangle in one of the following ways:
   - Click the point. The Measure Data dialog box shows the coordinates and other information.



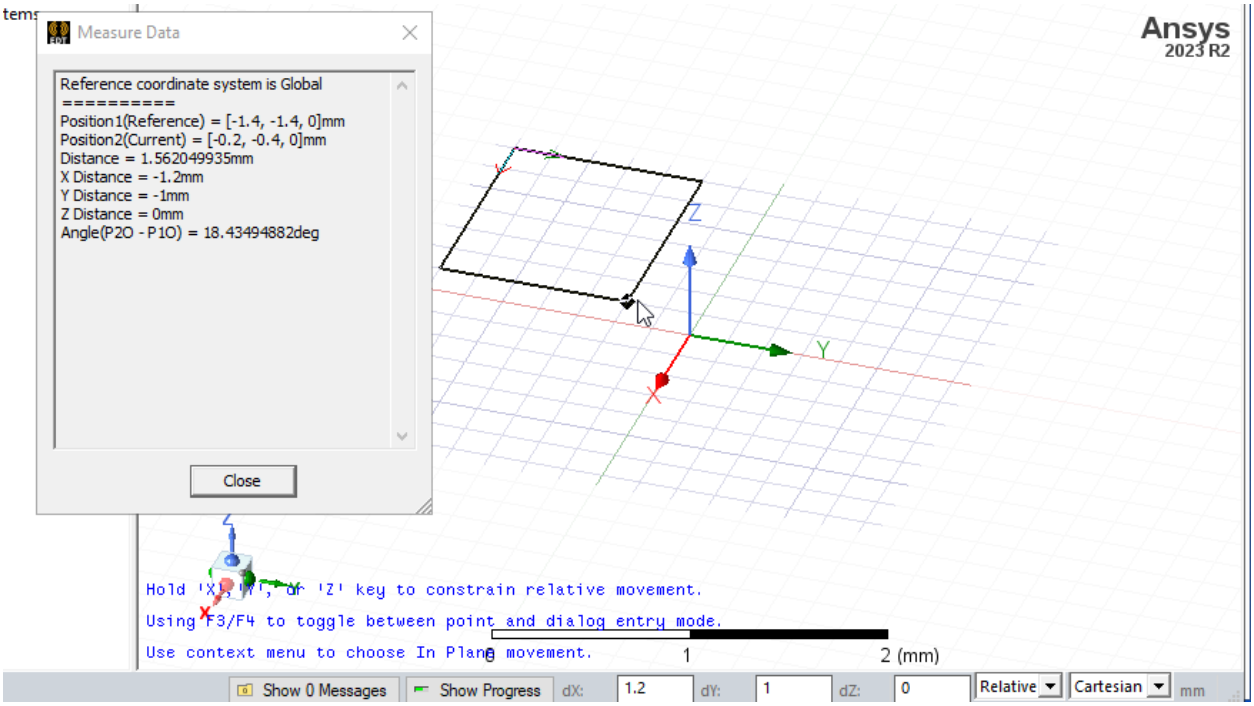
   - Type the point's coordinates in the text boxes in the status bar. The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop-down menu options to specify the box in Cartesian, Cylindrical, or Spherical coordinates.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

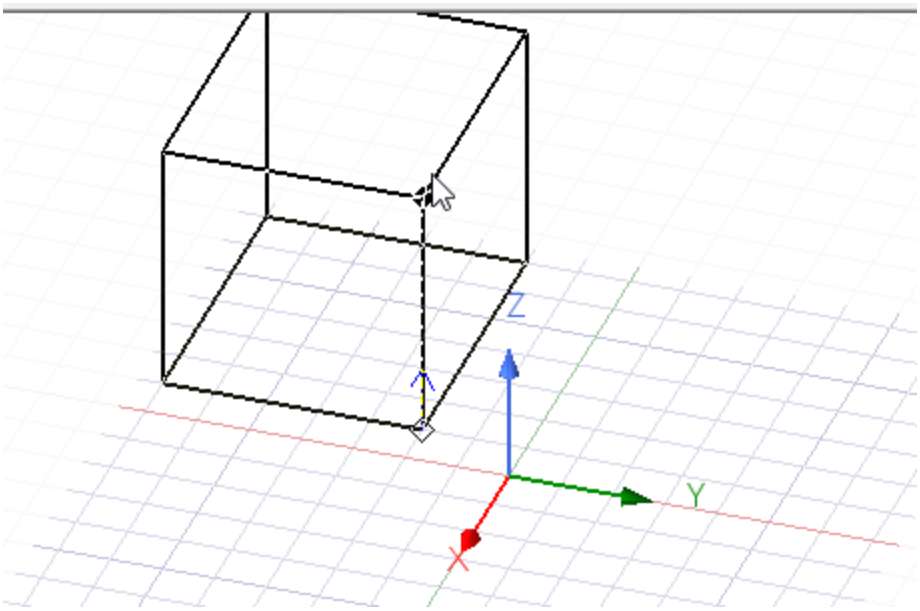
To delete the selected point and start over, press **Esc** or click **Escape Draw Mode** on the shortcut menu.

3. Select the second corner of the base rectangle in one of the following ways:
   - Click the point.



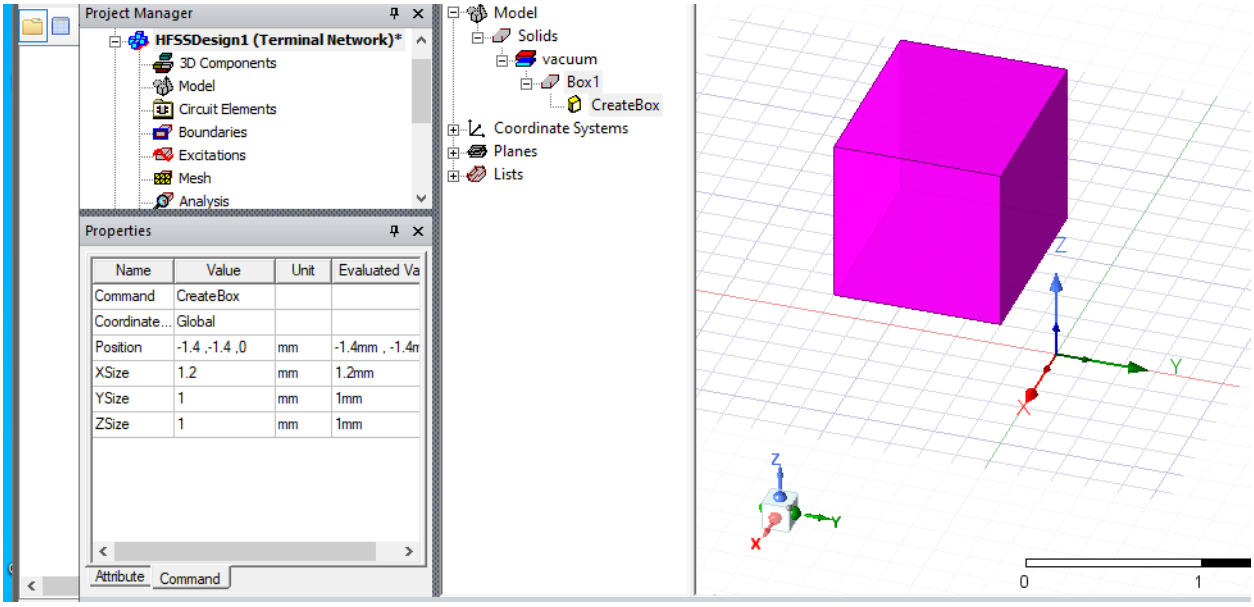
   - Type the coordinates of the point relative to the first diagonal corner in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.
4. Specify the height of the box by selecting a point on the axis perpendicular to the base rectangle. Select the point by clicking the point or typing the coordinates in the **dX, dY,** and **dZ** boxes.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Double -click or right-click and select **Done** from the shortcut menu.

   If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. You can also use the docked Properties window.

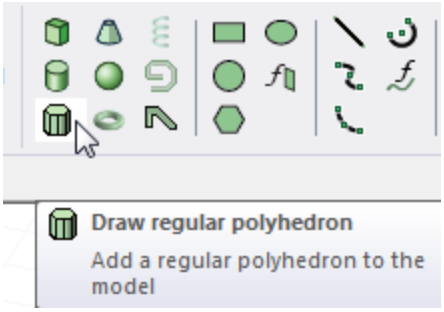


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

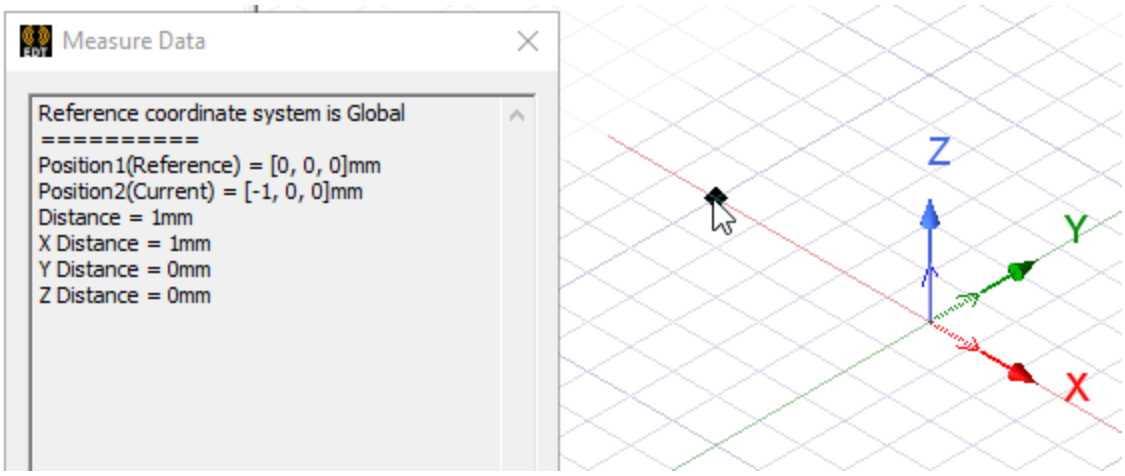
Drawing a Regular Polyhedron

In the modeler, regular polyhedrons are 3D objects with regular polygon faces; each face has three or more equal sides. Regular polyhedrons are useful for drawing faceted 3D objects. Before you draw a regular polyhedron, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

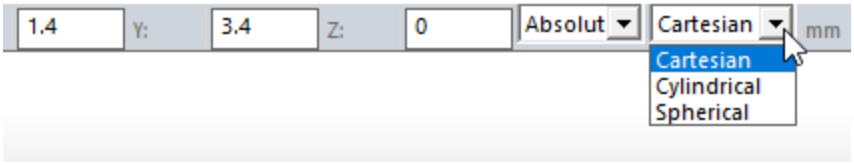
1. From the menu bar, click **Draw** > **Regular Polyhedron**, or on the **Draw** ribbon tab, click the **Draw regular polyhedron** icon:



2. Select the center point of the polyhedron in one of the following ways:
   - Click the point.

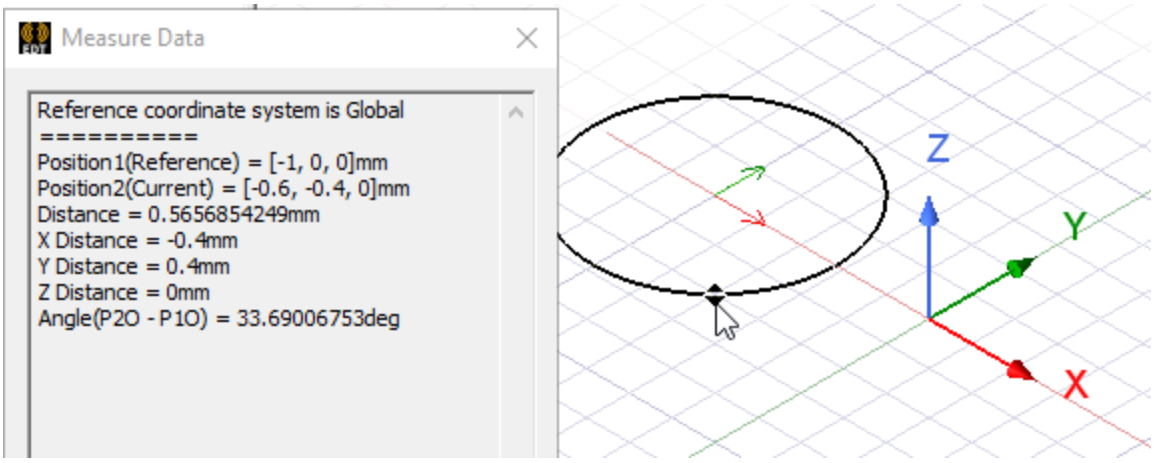


   - Type the point's coordinates in the text boxes in the status bar.The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop-down menu options to specify the regular polygon in Cartesian, Cylindrical, or Spherical coordinates.



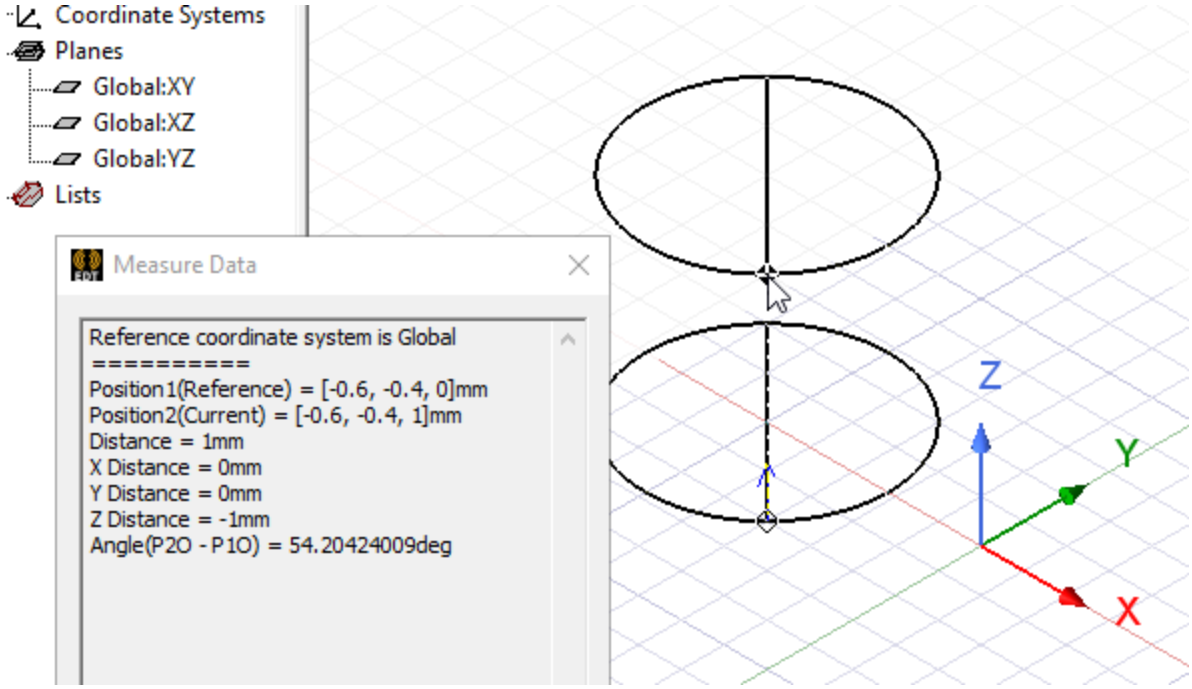
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select the radius of the polyhedron, the distance from the center point to one of the polyhedron's vertices, in one of the following ways:
   - Click the point.



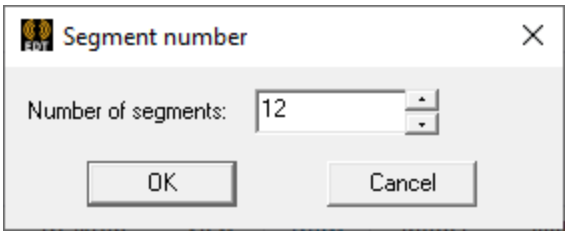
   - Type the coordinates of the point relative to the center point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

4. Select the height of the polyhedron, the distance from the origin plane along the normal from the radius point.

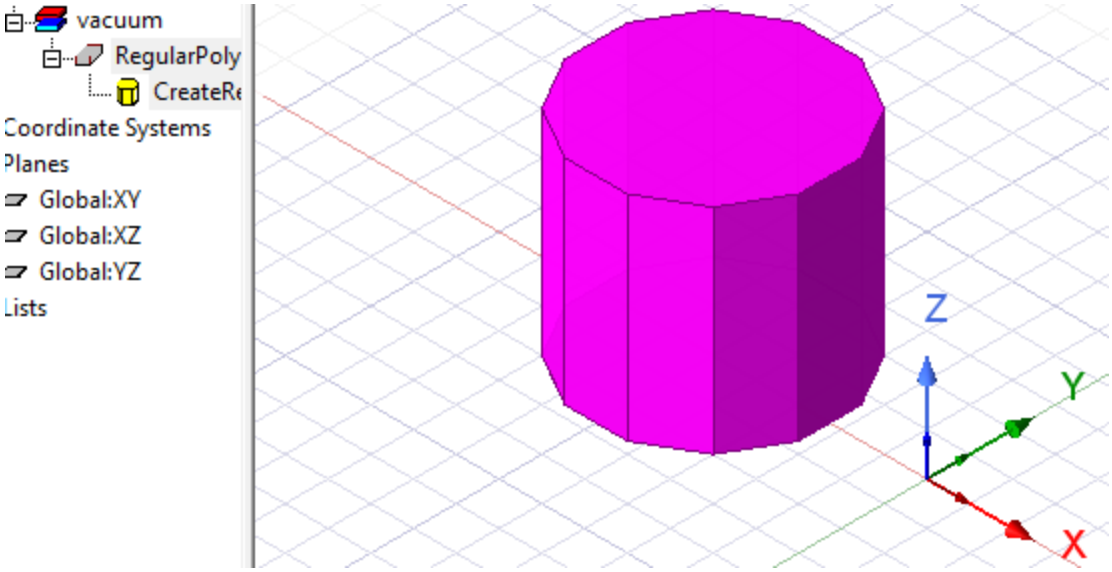


5. In the **Segment number** dialog box, enter the **Number of segments** in the polyhedron, and then click **OK**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Click **OK**. The **Segment number** dialog closes and the regular polyhedron appears in the Modeler window and in the History Tree.



6. If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. Click **OK**.

> **Note:**
>
> The radius is measured from the center point to a corner of the polygon, or the intersection of two edges. It is *not* measured from the center point to the midpoint of an edge.

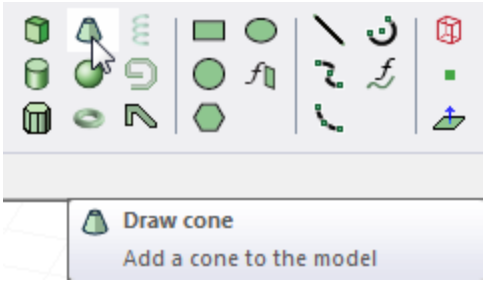
If the **Automatically cover closed polyline** option is selected in the **Modeler Options** dialog box, the polygon will be covered, resulting in a 2D sheet object. Otherwise it will be a closed 1D polyline object.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

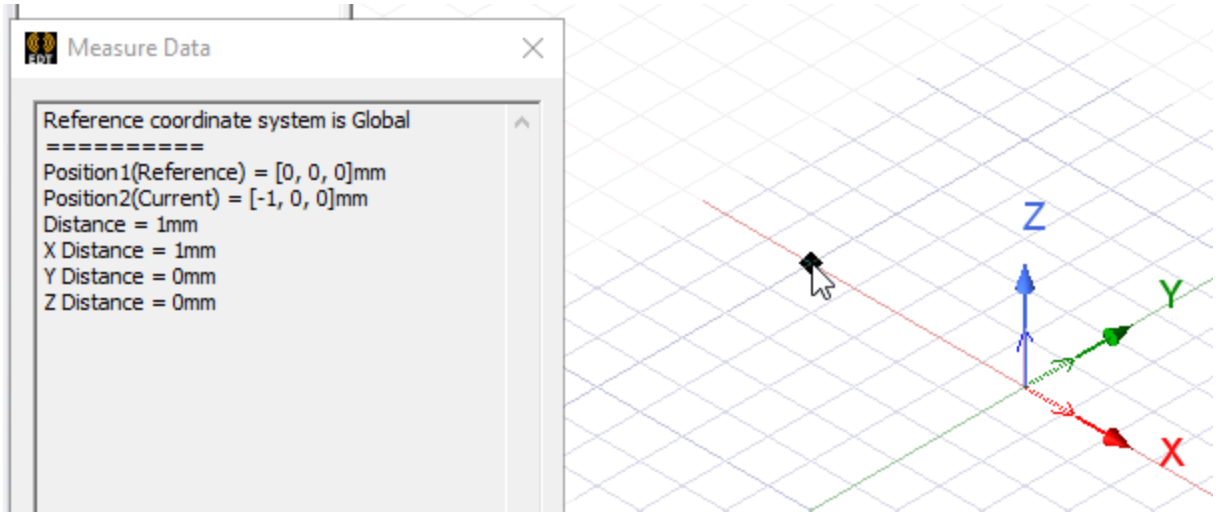
Drawing a Cone

Draw a cone by selecting the center point and radius of the cone's base circle, then specifying the radius of the cone's top circle and the cone's height. Cones are drawn as true surfaces in the modeler. Before you draw a cone, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

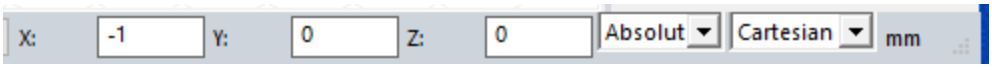
1. From the menu bar, click **Draw** > **Cone**, or on the **Draw** ribbon tab, click the **Draw cone** icon:



2. Select the center point of the cone's base circle in one of the following ways:
   - Click the point.



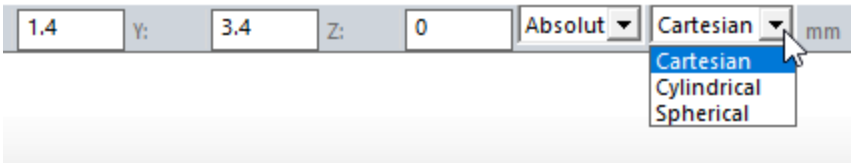
   - Type the point's coordinates in the text boxes in the status bar.



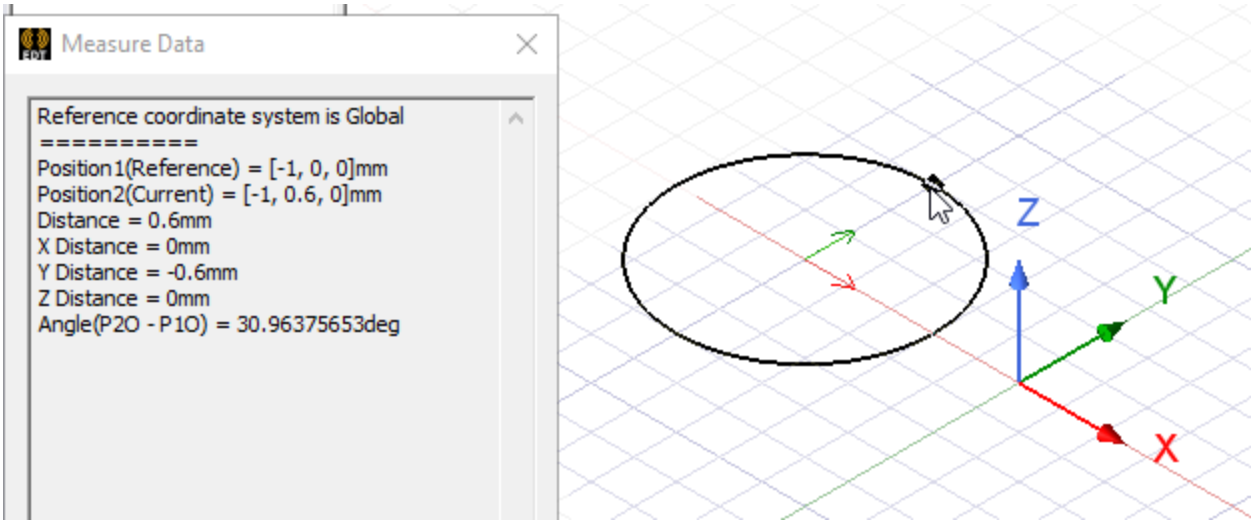
   The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop-down menu options to specify the cone in Cartesian,

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

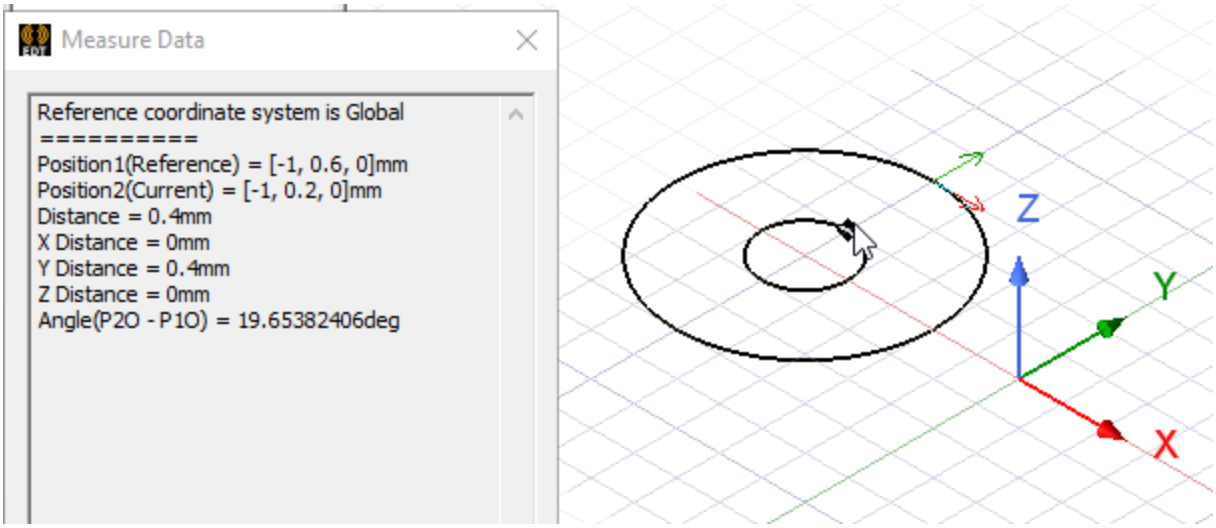
Cylindrical, or Spherical coordinates.



3. Specify the radius of the cone's base circle by selecting a point on the base circle's circumference. Select the point in one of the following ways:
   - Click the point.

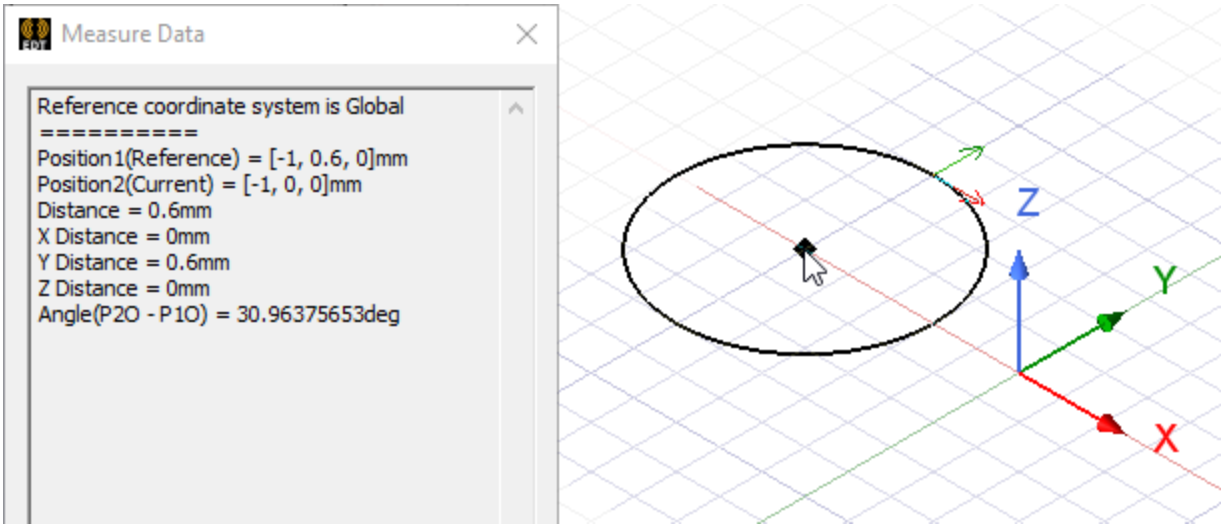


   - Type the coordinates of the point relative to the center point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.
4. Specify the radius of the cone's top circle by selecting a point on its circumference. Select the point by clicking it or typing its coordinates in the **dX, dY,** and **dZ** boxes.

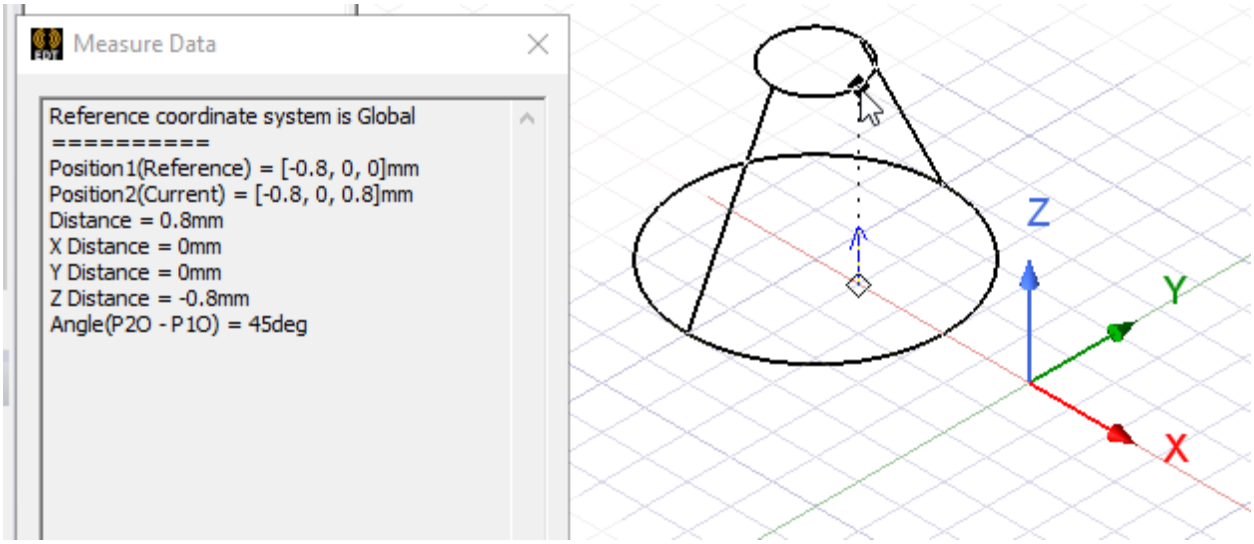


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To create an apex, select the same center point as the cone's base circle.



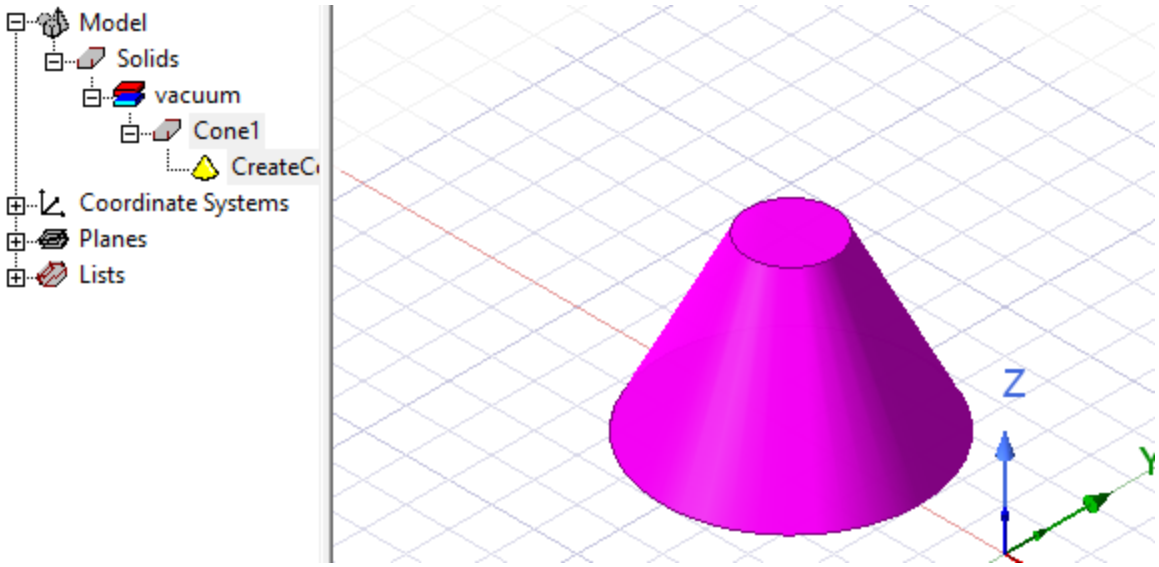
5. Specify the height of the cone by selecting a point on the axis perpendicular to the base circle's plane. Select the point by clicking the point or typing the coordinates in the **dX, dY,** and **dZ** boxes.



6. Double click, or right click and select **Done** from the shortcut menu. If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. The cone appears in the Modeler window

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and in the History Tree.



Note:

The 3D Modeler permits drawing true curved objects. However, the solution is obtained using a tetrahedral mesh, which conforms to the true surface only within the limits identified by certain mesh settings. The modeler has default settings for this conformance, which is a reasonable trade-off between solution speed and solution quality for most objects but may not be ideal for all such objects. High-aspect ratio curves structures (such as helices with narrow and curved cross-sections) may benefit from user control of the faceting values. For details about these commands, see the following:

Modifying Surface Approximation Settings, and related sections:

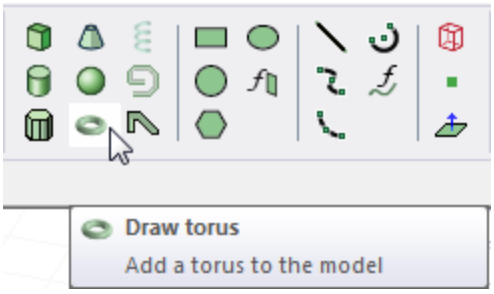
Rectilinear Elements and Curvilinear Elements and Guidelines for Modifying Surface Approximations.

Drawing a Torus

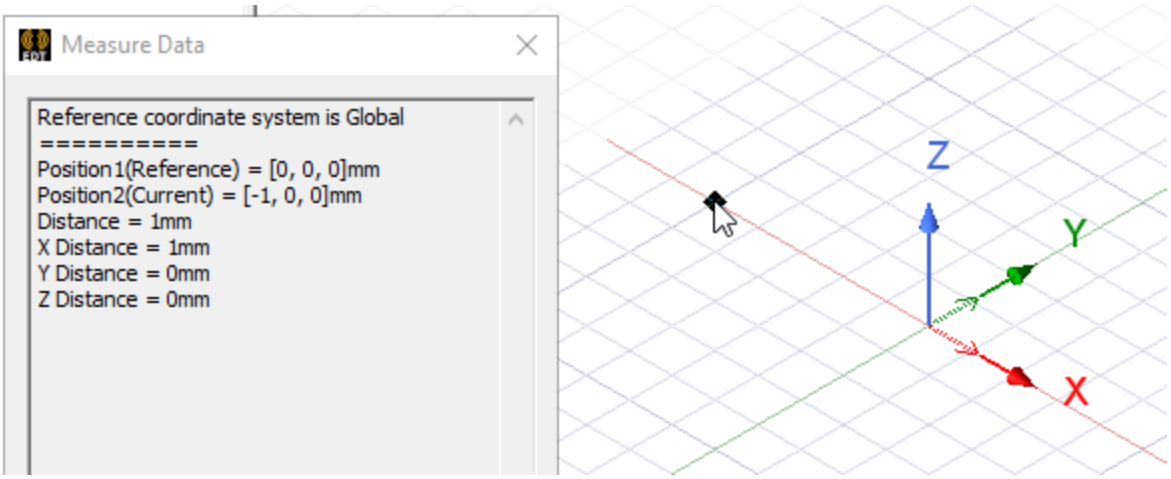
Draw a torus by selecting its center point, major radius, and minor radius. The modeler then sweeps a circle around a circular path. A torus is drawn as true surfaces in the modeler. Before you draw a torus, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw** > **Torus**, or on the **Draw** ribbon tab, click the **Draw torus** icon:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

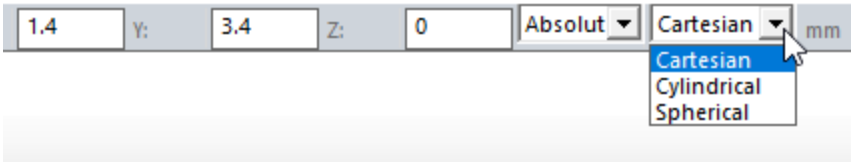


2. Select the center point of the torus in one of the following ways:
   - Click the point.



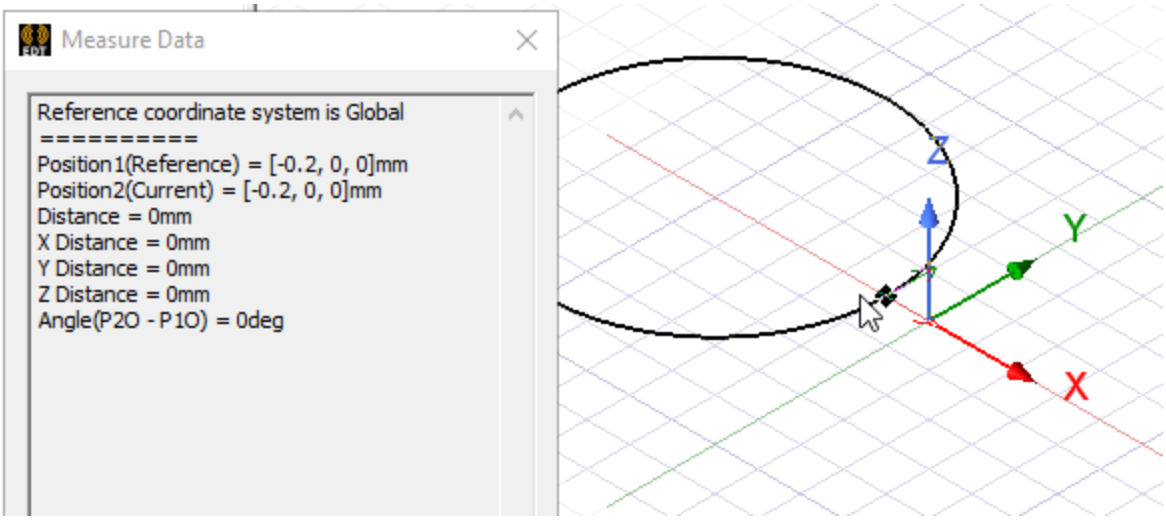
   - Type the point's coordinates in the text boxes in the status bar.

     The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop-down menu options to specify the torus in Cartesian, Cylindrical, or Spherical coordinates.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Specify the major radius by selecting a point in one of the following ways:
   - Click the point.

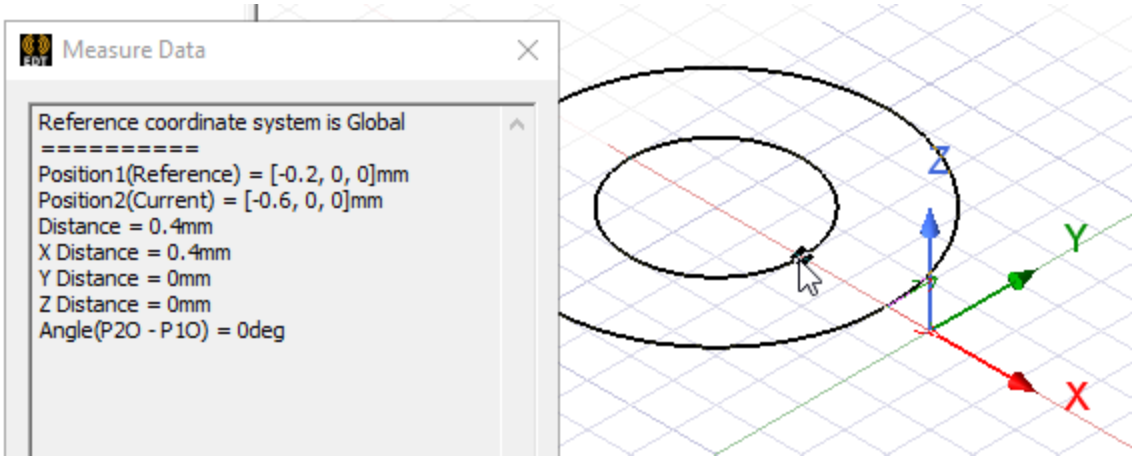


   - Type the coordinates of the point relative to the center point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

   The major radius determines the diameter of the torus.

4. Specify the minor radius by selecting a point relative to the major radius point.

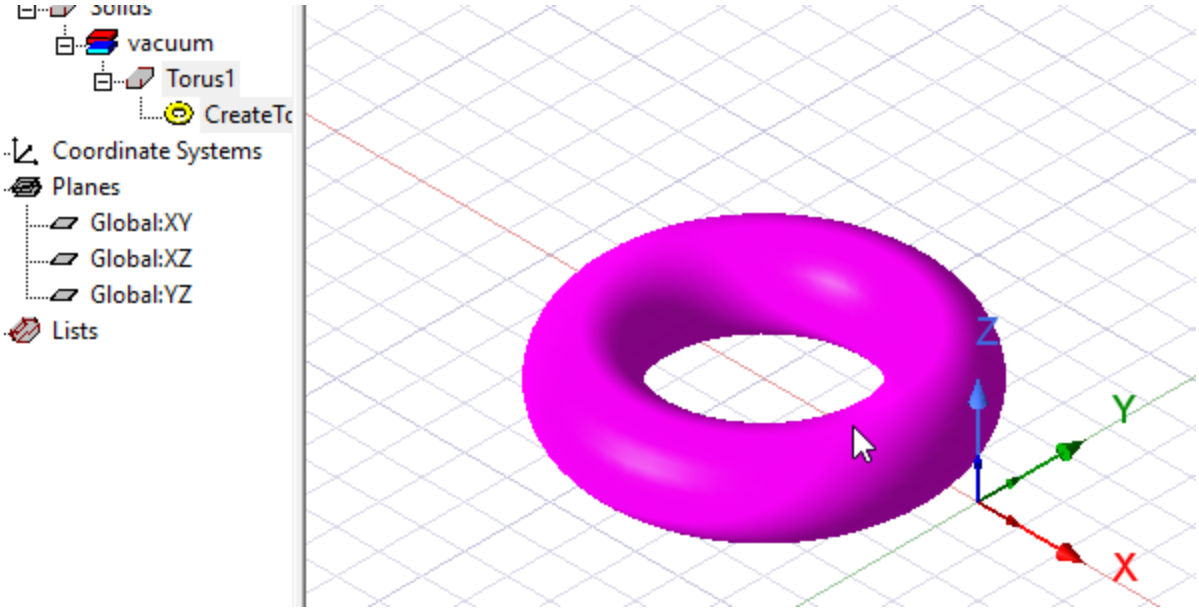
   The minor radius determines the diameter of the "donut hole".



If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Double-click or right-click to open the shortcut menu and click **Done**. The torus appears in the Modeler window and in the History Tree.



6. If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. Click **OK**.

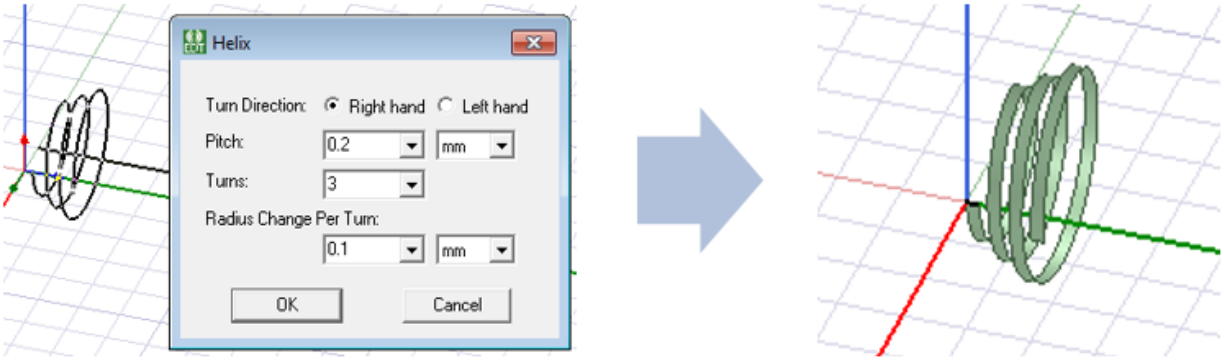
> **Note:**
>
> The 3D Modeler permits drawing true curved objects. However, the solution is obtained using a tetrahedral mesh, which conforms to the true surface only within the limits identified by certain mesh settings. The modeler has default settings for this conformance, which is a reasonable trade-off between solution speed and solution quality for most objects but may not be ideal for all such objects. High-aspect ratio curves structures (such as helices with narrow and curved cross-sections) may benefit from user control of the faceting values. For details about these commands, see the following:
>
> Modifying Surface Approximation Settings, and related sections:
>
> Rectilinear Elements and Curvilinear Elements and Guidelines for Modifying Surface Approximations.

Drawing a Helix

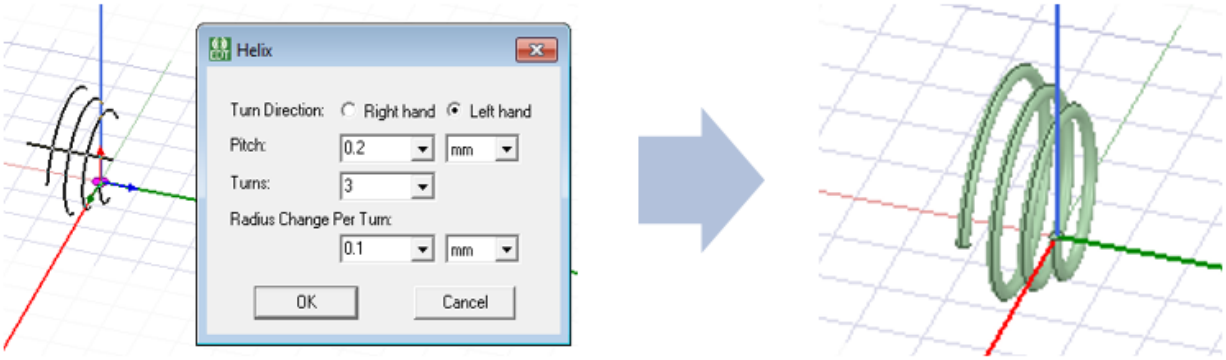
A helix is a 3D spiral object created by sweeping a line or 2D sheet object that you create along a vector that you define. The center of the helix is defined by the placement of the vector relative to the swept object. You must define the vector so that the swept object will not self-intersect, the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

rule being, do not draw the vector so that it centers on or overlaps a selected 2D object. After you create the vector, the **Helix** dialog box opens, where you then specify the turn direction, pitch, number of turns, and radius of the turn to define the helix.

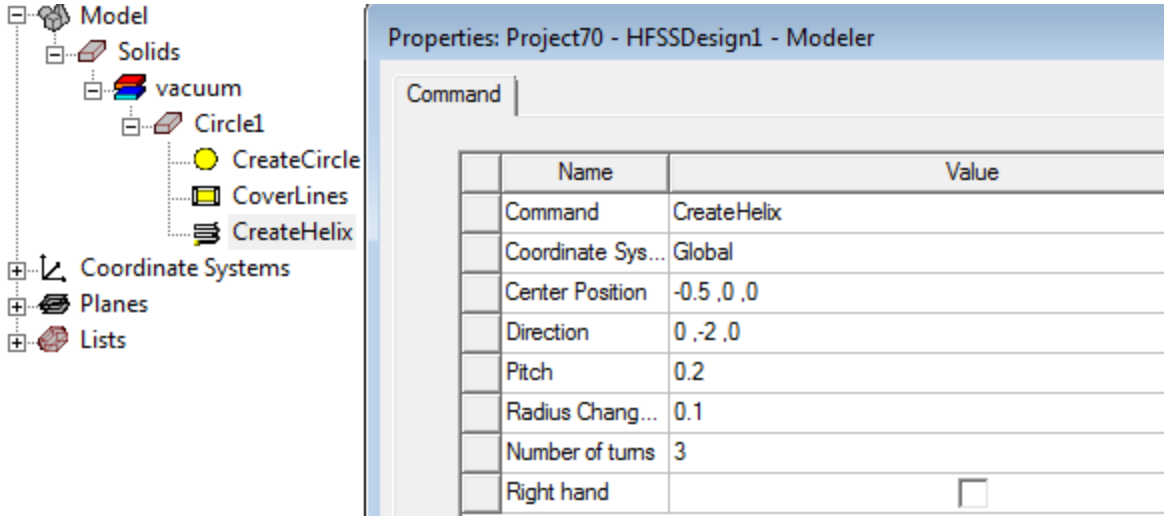


Sweeping a 2D object results in a hollow 3D object. Sweeping a 2D sheet object results in a 3D solid object.



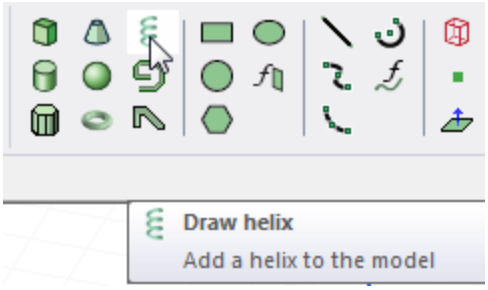
Once you have created a helix, you can changing it by edit the properties of the original object, and by editing the helix command properties for coordinate system, center position (which corresponds to the vector placement), direction, pitch, turns, radius change, and direction.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To create a helix:

1. Create and select the line or 2D object you want to sweep to form a helix. Selecting a valid object enables the **Helix** icon on the **Draw** tab of the ribbon.
2. From the menu bar, click **Draw** > **Helix** or, on the **Draw** ribbon tab, click the **Draw helix** icon:

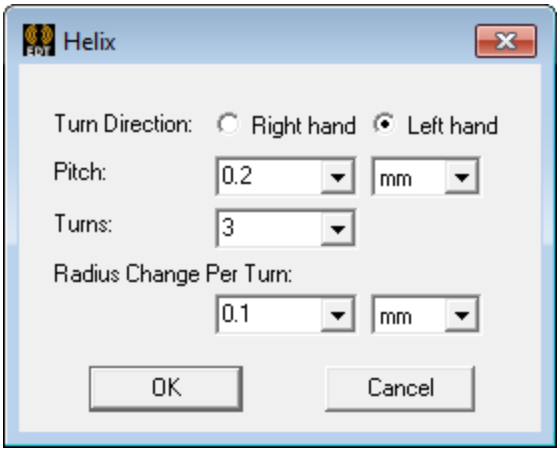


3. Draw the vector you want to sweep the object around. The two points that describe the vector affect axis location and direction only and not the helix length. (The vector definition corresponds to the **Center Position** and **Direction** properties of the **CreateHelix** command in the History tree.) The helix length is determined when you enter the pitch and number of turns in the **Pitch** and **Turns** text boxes. The initial radius of the helix is determined by the axis position relative to the object being swept. It is important that the swept object cannot intersect itself. For a 2D object, you should ensure that the vector you draw does not intersect the interior. If it does, the object cannot be drawn due to self-intersection and you receive a message that "Body could not be created because of invalid parameters."
    a. Select the start point by clicking the point or typing its coordinates in the **X, Y,** and **Z** text boxes. (Remember that you can also edit the parameters of a completed helix).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Select the endpoint by clicking the point or typing its coordinates relative to the start point in the **dX, dY,** and **dZ** boxes.

The **Helix** dialog box appears:



4. For **Turn Direction**, select **Right hand** if the turn direction is clockwise and **Left hand** if the turn direction is counter-clockwise.
5. In the **Pitch** text box, type the distance between each turn in the helix, and click a unit in the drop-down menu.
6. In the **Turns** text box, type the number of complete revolutions the object will make along the vector.
7. In the **Radius Change per Turn** text box, type a number for the increase in the radius and select the units from the drop-down menu.
8. After you set these values, the selected object is swept along the vector to form a helix. The original object you swept is deleted.
9. Click **OK** to create the Helix.

   If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. You can also select the commands for the object and the helix command in the History Tree to access and edit their properties.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The 3D Modeler permits drawing true curved objects. However, the solution is obtained using a tetrahedral mesh, which conforms to the true surface only within the limits identified by certain mesh settings. The modeler has default settings for this conformance, which is a reasonable trade-off between solution speed and solution quality for most objects but may not be ideal for all such objects. High-aspect ratio curves structures (such as helices with narrow and curved cross-sections) may benefit from user control of the faceting values. For details about these commands, see the following:

Modifying Surface Approximation Settings, and related sections:

Rectilinear Elements and Curvilinear Elements and Guidelines for Modifying Surface Approximations.

Drawing a Segmented Helix with Polygon Cross-Section Using a User Defined Primitive

Ansys provides you with a DLL to define the parameters of a segmented helix with a polygon cross-section. Before you draw a helix, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. Click **Draw** > **User Defined Primitive** > **SegmentedHelix** > **PolygonHelix**.

   The **Create User Defined Part** dialog box appears. The **Parameters** tab permits you to edit the parameters. An **Info** tab contains information about the user defined primitive, its purpose, the company/author who created it, the date created and the version number.

2. Specify the values for the following parameters:

PolygonSegments	Number of segments in the polygon cross-section. Enter zero (0) for true circle
PolygonRadius	Radius of the polygon cross-section.
StartHelixRadius	The radius of a segmented helix is defined from the helix center of rotation to the center of the helix cross-section at segment transitions. The first and last segments of the helix are half segments. See this figure.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RadiusChange	The radius change per turn of the helix.
Pitch	Distance between helix turns.
Turns	The number of turns in the helix.
SegmentsPerTurn	The number of segments constructing each turn. Enter zero (0) for true curve.
RightHanded	Helix winding direction. Enter non-zero value for right-handed helix.

3. Click **OK**.

Drawing a Segmented Helix with Rectangular Cross-Section Using a User Defined Primitive

Ansys provides you with a DLL to define the parameters of a segmented helix with a rectangular cross-section. Before you draw a segmented helix, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. Click **Draw** > **User Defined Primitive** > **SegmentedHelix** > **RectHelix**.

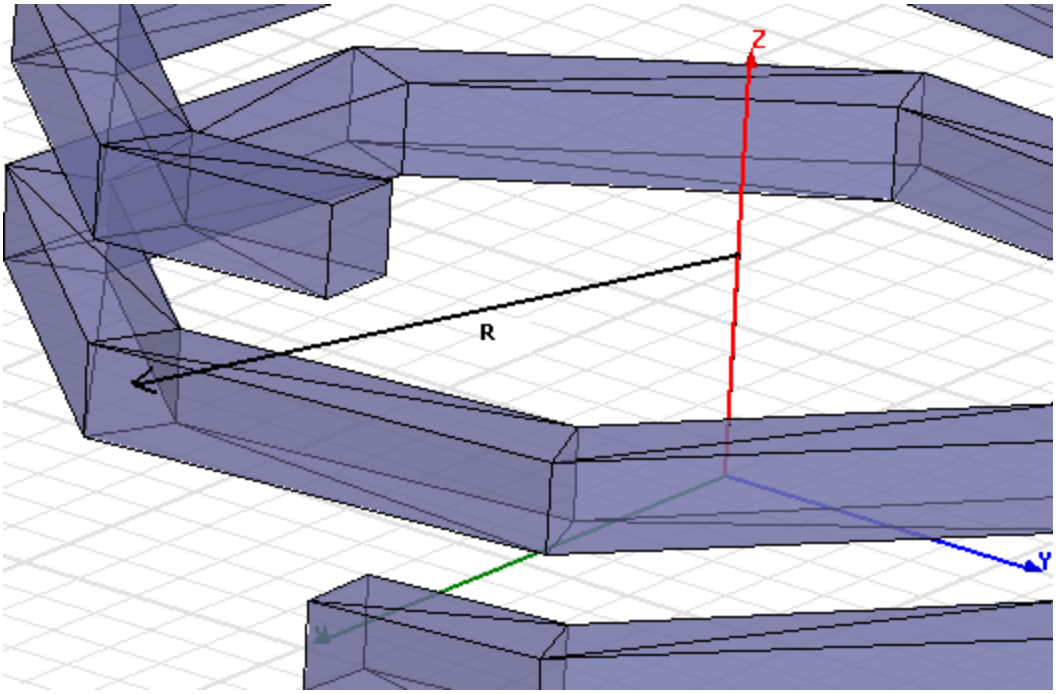
   The **Create User Defined Part** dialog box appears. The **Parameters** tab permits you to edit the parameters. An **Info** tab contains information about the user defined primitive, its purpose, the company/author who created it, the date created and the version number.

2. Specify the values for the following parameters:

RectHeight	Height of rectangular cross-section.
RectWidth	Width of rectangular cross-section.
StartHelixRadius	The radius of a segmented helix is defined from the helix center of rotation to the center of the helix cross-section at segment transitions. The first and last segments of the helix are half segments. See this figure.
RadiusChange	The radius change per turn of the helix.
Pitch	Distance between helix turns.
Turns	The number of turns in the helix.
SegmentsPerTurn	The number of segments constructing each turn. Enter zero (0) for true curve.
RightHanded	Helix winding direction. Enter non-zero value for right-handed helix.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

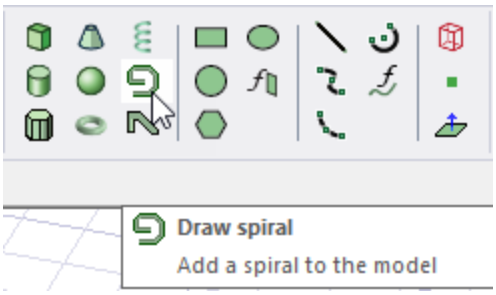
3. Click **OK**.



Drawing a Spiral

A spiral is a 2D or 3D spiral object created by sweeping an object around a vector. Sweeping a 1D object results in a 2D sheet object. Sweeping a 2D sheet object results in a 3D solid object. Before you draw a spiral, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. Select the 1D or 2D object you want to sweep to form a spiral.
2. From the menu bar, click **Draw** > **Spiral** or, on the **Draw** ribbon tab, click the **Draw spiral** icon:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Draw the vector you want to sweep the object around:
   a. Select the start point by clicking the point or typing its coordinates in the **X, Y,** and **Z** text boxes.
   The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop-down menu options to specify the sphere in Cartesian, Cylindrical, or Spherical coordinates.

   b. Select the endpoint by clicking the point or typing its coordinates relative to the start point in the **dX, dY,** and **dZ** boxes.

   The **Spiral** dialog box appears.

4. Select **Right hand** if the turn direction is clockwise and **Left hand** if the turn direction is counter-clockwise.
5. In the **Radius Change** text box, type the difference in radius between each turn of the spiral.

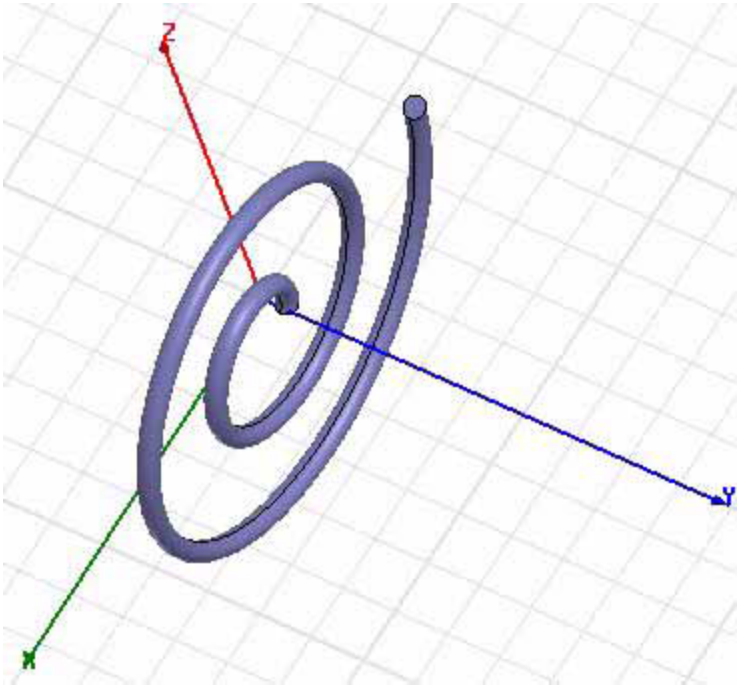
   The radius of the first turn is measured from the center point of the 1D or 2D object you are sweeping to the vector you drew.

6. Click a unit for the radius in the drop-down list.
7. In the **Turns** text box, type the number of complete revolutions the object will make around the vector.

   The selected object is swept around the vector to form a spiral. The original object you swept is deleted. If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Click **OK**.



This 3D spiral was created from a 2D circle drawn at z = 0. The turn direction was right hand, the radius change was set at 2, and the number of turns was set at 2.

Note:

The 3D Modeler permits drawing true curved objects. However, the solution is obtained using a tetrahedral mesh, which conforms to the true surface only within the limits identified by certain mesh settings. The modeler has default settings for this conformance, which is a reasonable trade-off between solution speed and solution quality for most objects but may not be ideal for all such objects. High-aspect ratio curves structures (such as helices with narrow and curved cross-sections) may benefit from user control of the faceting values. For details about these commands, see the following:

Modifying Surface Approximation Settings, and related sections:

Rectilinear Elements and Curvilinear Elements and Guidelines for Modifying Surface Approximations.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Drawing a Spiral Using User-Defined Primitives

Ansys provides you with a DLL and a Python script to define the parameters of a rectangular spiral. Before you draw a spiral, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw** > **User Defined Primitive** > **Examples** > **RectangularSpiral (DLL)** or **Rectangular Spiral (Python)**.

   The **User Defined Primitive Operation** dialog box appears. The **Parameters** tab permits you to see and edit the parameters. An **Info** tab contains information about the user defined primitive, its purpose, the company/author who created it, the date created and the version number.

2. Specify the values for the following parameters:

Xpos	X location of the starting point.
Ypos	Y location of the starting point.
Dist	The separation distance between turns.
Turns	The number of complete revolutions the object will make around the vector
Width	The width of the spiral.
Thickness	The thickness/height of the spiral. If you specify the thickness as zero, the modeler draws a sheetobject.

3. Click **OK**.

   This creates the primitive and displays the **Properties** dialog box for the new object.

Tip:

To see newly created DLLs, click **Draw** > **User Defined Primitive** > **Update Menu**.

To see the primitives that you have created, click **Draw** > **User Defined Primitive** > **UserLib.**

Drawing a Geodesic Sphere Using a User-Defined Primitive

Ansys provides you with a .py script to define the parameters of a geodesic polyhedron and draw a geodesic sphere. A geodesic polyhedron has straight edges and flat faces that approximate a sphere.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Draw** > **User Defined Primitive** > **Examples** > **geodesic_sphere**.

   The **User Defined Primitive Operation** dialog box appears. The **Parameters** tab can be used to edit the parameters. The **Info** tab contains information about the user defined primitive, its purpose, the company/author who created it, the date created and the version number.

2. Specify the values for the following parameters:

radius	The radius of the sphere. Enter a positive number and desired unit of measure.
frequency	Indicates how many times each side of the base triangle is subdivided"

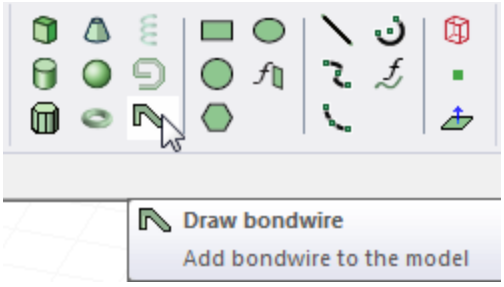
3. Click **OK**.

This creates the primitive and displays the **Properties** dialog box for the new object.

Drawing a Bondwire

A bondwire is a thin metal wire that connects a metal signal trace with a chip. Please see the topic Bondwires in the *HFSS: Technical Notes* before drawing a bondwire.

1. From the menu bar, click **Draw** > **Bondwire** or, on the **Draw** ribbon tab, click the **Draw bondwire** icon:



2. Select the bond pad point in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the text boxes in the status bar.
3. Select the lead point by clicking the point or typing the coordinates in the text boxes in the status bar.

   The **Bondwires** dialog box appears.

4. In the **Type** list, click the JEDEC modeling standard shape you want the bondwire to have: **JEDEC 4-point**, **JEDEC 5-point, or Low**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Type** selection changes the dialog box's bondwire graphic and shows options for that type.

5. Enter the number of facets in the bondwire in the **No. of Facets** text box.

   The minimum value is 3. The value describes the number of faces that make up the circumference of the bondwire.

6. In the diameter field, specify a diameter value and select the units from the drop-down menu.
7. Enter the height between the bond pad and the top of the loop in the **h1** text box. Include the height's unit of length.
8. The value in the **h2** text box is the height between the bond pad and the lead point. It was calculated by Ansys Electronics Desktop based on the lead point you selected. If you modify the value of **h2**, the lead point will be modified.

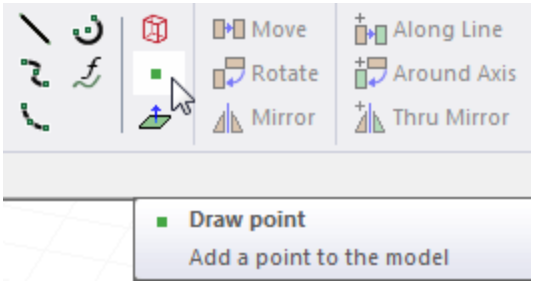
   Optionally, type a new value in the **h2** text box. Include the height's unit of length.

9. If you selected **JEDEC 5-point** or **Low**, do the following:
   a. Type the angle between the horizontal plane and the wire at the bond pad point in the **alpha** text box.
   b. Type the angle between the horizontal plane and the wire at the lead point in the **beta** text box.
10. Click **OK**.

Drawing a Point

Drawing a point object within the problem region enables you to plot fields or perform field computations at that point. Points are always considered non-model objects by the modeler. Before you draw a point, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

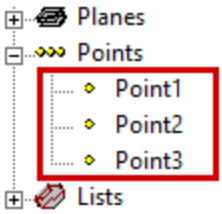
1. From the menu bar, click **Draw** > **Point** or, on the **Draw** ribbon tab, click the **Draw point** icon:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select the point in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the text boxes in the status bar.

   The point is listed under **Points** in the History Tree:

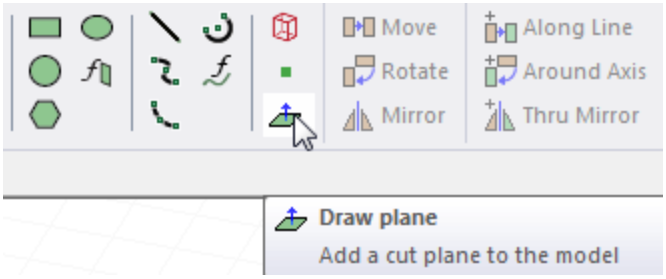


> **Note:**
>
> There is no visible indication of the point in the drawing area unless it is selected in the History Tree. Also, the point cannot be selected graphically once added to the model. It must be selected from the History Tree or from the **Geometry** window within the **Fields Calculator**.

Drawing a Plane

A plane object is a cut plane through the problem region. You can use plot fields to perform field computations on its surface. Planes are always considered non-model objects by the modeler. Before you draw a plane, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw** > **Plane** or, on the **Draw** ribbon tab, click the **Draw plane** icon:



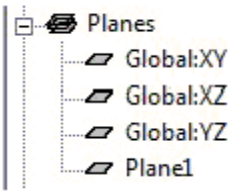
2. Select the origin in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the text boxes in the status bar.

   To delete the selected point and start over, press **Esc**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select a normal point in one of the following ways:
   - Click the point.
   - Type the coordinates of the point relative to the origin in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

   The plane is created. Its center point is located at the origin you specified and oriented perpendicular to the normal point you specified. The new plane is listed under **Planes** in the History Tree.

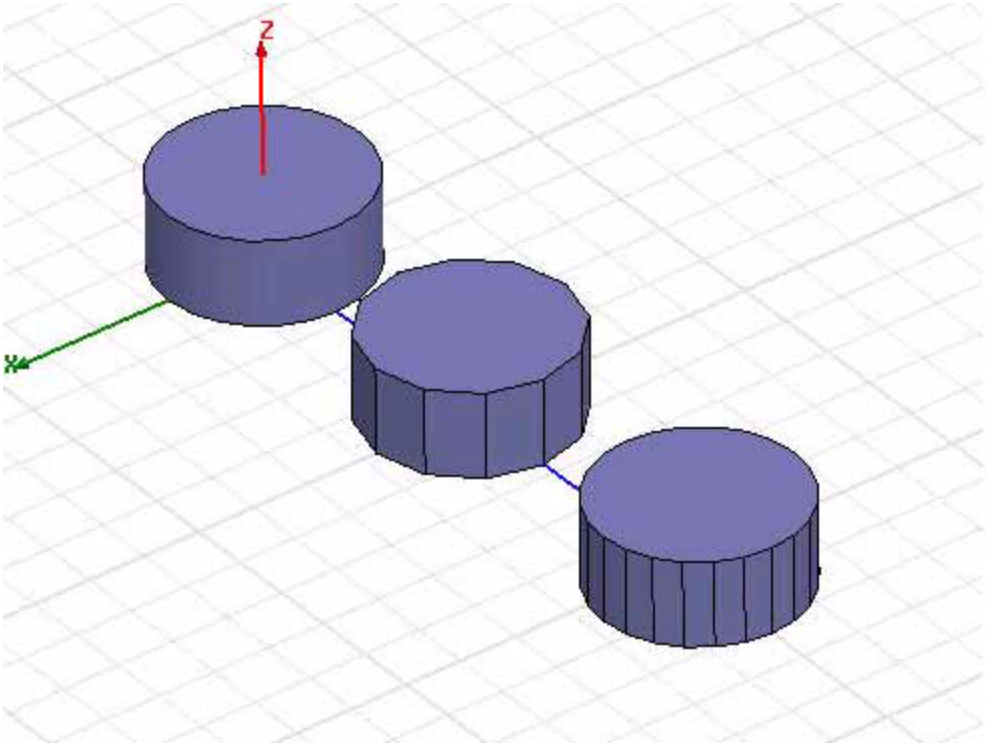


> **Note:**
>
> You only need to draw a plane that does not lie on a predefined xy, yz, and xz plane. Default planes are created on the xy, yz, and xz planes of the global coordinate system as well as any new coordinate system you create.

Creating Segmented Geometry

For some structures, you may want to create segmented as opposed to smooth (or True) surfaces. The following figure shows a comparison of a cylinder created with true surfaces and with segmented surfaces.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following model objects can be created as segmented structures:

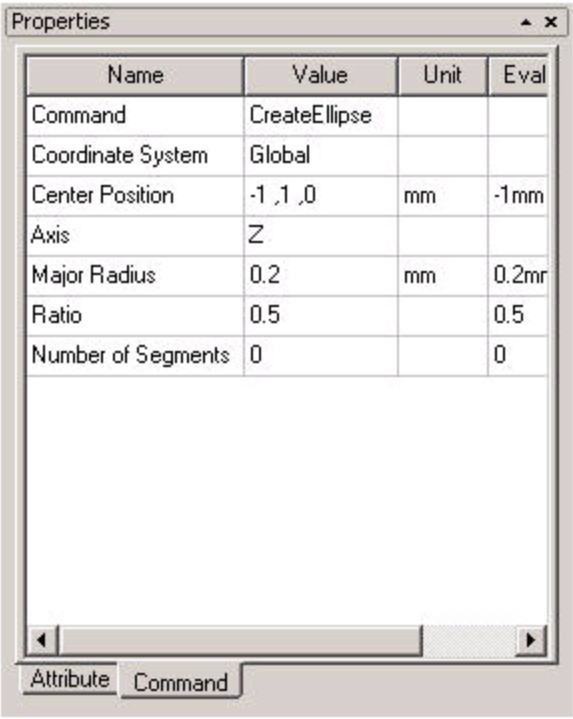
Circle, Ellipse, Cylinder	See: Segmented Objects
Polyline, Arc, Line Segment	See: Converting Polyline Segments

Segmented Objects

To create segmented circles, ellipses, and cylinders use the **Number of Segments** parameter on the Command Tab of the **Properties** dialog box as shown below. To convert an object from true surface to segmented, do the following:

1. Select the **circle**, **ellipse**, or **cylinder** in the modeler window or in the History Tree.
2. In the command tab of the properties window (shown docked below), change the **Number of Segments** to an integer value of three or greater and press **Enter**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Values of 1 and 2 are not valid values for the circle, ellipse, or cylinder command and will cause an error.

Drawing Non-Model Objects

If you want to create an object that does not affect the geometric model, define the object as *non-model*. This ensures that the object is used for analysis only; it will not affect the solution process. After drawing the non-model object, assuming it lies in the problem region, you can use it in the reporter as a place on which to plot field quantities. For example, draw a non-model line across the design, then (in the reporter) plot Mag_E on every point along that line.

You can assign output variables (default) and design variable as property values for non-model objects.

Following are examples of using non-model objects to analyze a solution:

- Draw a polyline along which to plot fields or perform field computations. Note that when you create a value versus distance plot, by default, the line will be divided into 100 equally spaced points. You can modify the number of points into which the line is divided in the **Edit Sweeps** dialog box. For more information, see: Specifying Variable Values for a Sweep Definition.
- Draw a rectangle upon which to plot fields in the reporter or perform field computations.
- Draw a volume box to analyze fields in areas of the problem region that are not occupied by an object or that consist of parts of several objects.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Draw a plane, which is always a non-model object.
- Draw a point, which is always a non-model object, in order to plot fields in the Reporter or perform field computations at that point.

What do you want to do?

Switch to non-model drawing mode. Objects you draw in non-model mode will not be included in the solution process.

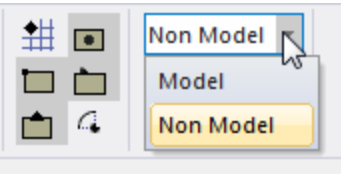
Modify an existing model object to be a non-model object.

Selecting Non-Model Drawing Mode

To switch to non-model drawing mode:

1. Click **Modeler** > **New Object Type**> **Non Model**.

   Alternatively, on the **Draw** ribbon tab, choose **Non Model** from the **Set to model non model type** drop-down menu:

2. Draw the object.

Changing an Object to Non-Model

To modify an existing object to be a non-model object:

1. Select the object you want to modify.
2. In the docked **Properties** window, clear the **Model** option.

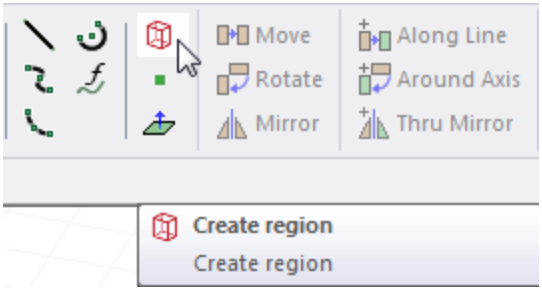
   The object will not be included in the solution process. If the object lies in the problem region, you can plot solution quantities on it.

Drawing a Region

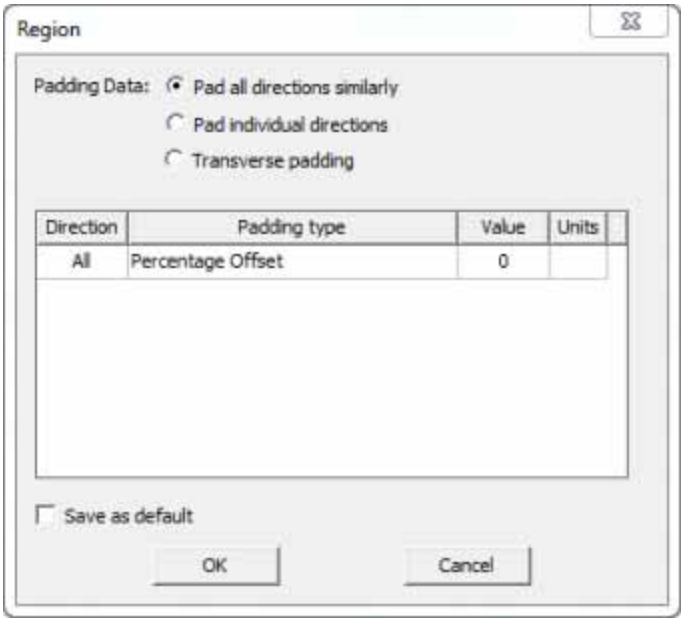
To draw a region encompassing the objects in the current project:

1. From the menu bar, click **Draw** > **Region** or, on the **Draw** ribbon tab, click the **Create region** icon:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

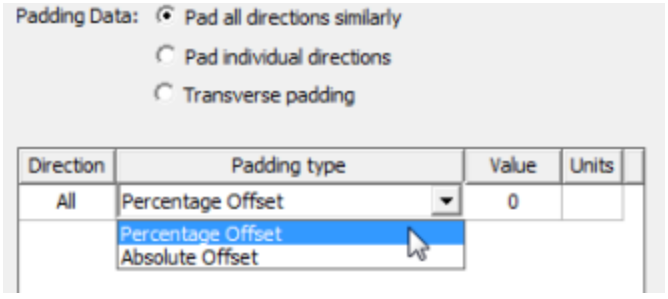


This displays the **Region** dialog box.



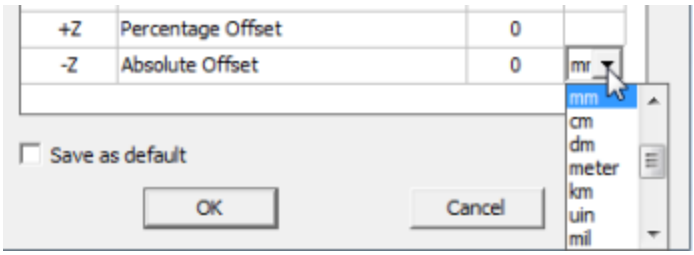
2. For the Padding data, click the Padding Data radio button as **Pad all directions similarly, Pad individual directions**, or **Transverse padding**.

   Selecting **Pad all directions similarly** leaves the Padding type field as requiring a single value that affects all directions. In this case, you can specify the Padding type by selecting **Percentage Offset** or **Absolute Offset** from the drop-down menu.

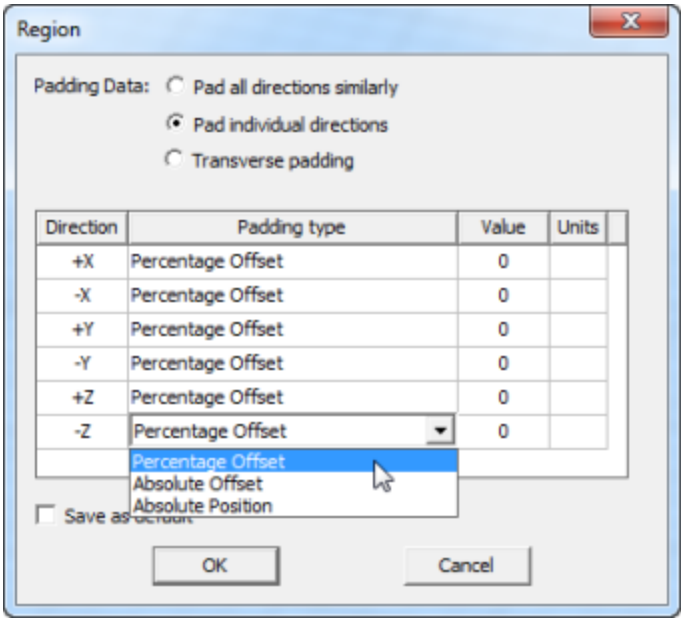


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you select **Absolute Offset**, you also specify the units by selecting from a drop-down menu.



Selecting **Pad individual directions** displays the Padding Percentage as a table of Positive and Negative X,Y, and Z coordinates, permitting you to specify padding for each direction. In this case, you can specify the Padding type by selecting **Percentage Offset** or **Absolute Offset** or **Absolute Position** from the drop-down menu.

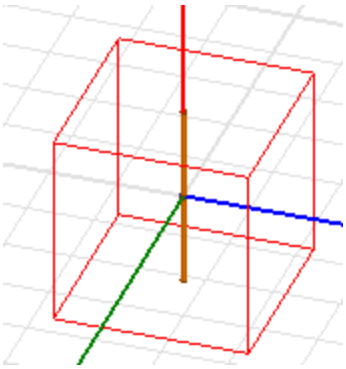


Selecting **Transverse padding** means that padding in one direction is controlled by dimensions in other two directions. Each direction (for example, X) is padded with user-specified percentage of diagonal length of the other two directions (Y and Z).

For a single antenna element, something that is radiating does not radiate with reasonable efficiency until it is lambda/2 in length, L, and it radiates in the direction transverse to the dimension L. So we get the dimension L from the region command and pad it 50% which would then correspond to the lambda/4 of the frequency at which the device is lambda/2 but only in the direction in which energy radiates, the transverse direction.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A basic example is a thin resonate dipole oriented in z. There is zero radiation in the z-direction so no need to pad the abc in that direction since no energy goes in that direction. But in X and Y transverse to the length L we do need to pad lambda/4 (the 50% setting) because it is that direction that the device radiates.



3. Specify the Padding values in the fields and select the units from the drop-down menu.
4. If desired, select the check box to save the values as Default.
5. Click **OK** to close the dialog and create the region.

The region is drawn, selected, and displayed in the History tree. It is created using the current coordinate system. The Properties dialog for the Create Region icon in the History tree shows the coordinate system and Padding values. The Properties dialog for the selected model lists attributes tab including Name, Material (Default, vacuum), Solve inside, Orientation, Model, Color, Display Wireframe, and Transparency. You can edit all of these values.

If you try to create a region that does not contain all of the objects in your model, the modeler automatically expands the region to cover all objects. The region also updates automatically as your geometry changes.

Only one region can be created for a single project using the **Draw** > **Region** command. If you try to create a second region, the **Properties** window appears for the existing region, allowing you to change operation parameters and attributes.

Related Topics

Drawing a Sub-Region

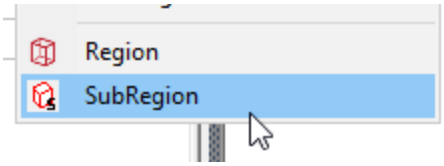
Drawing a SubRegion

The Modeler allows you to define a subregion that encompasses a set of objects inside previously defined region. Like regions, subregions automatically resize if the objects contained in these subregions change. You can create nested subregions, and reassign the content of a subregion.

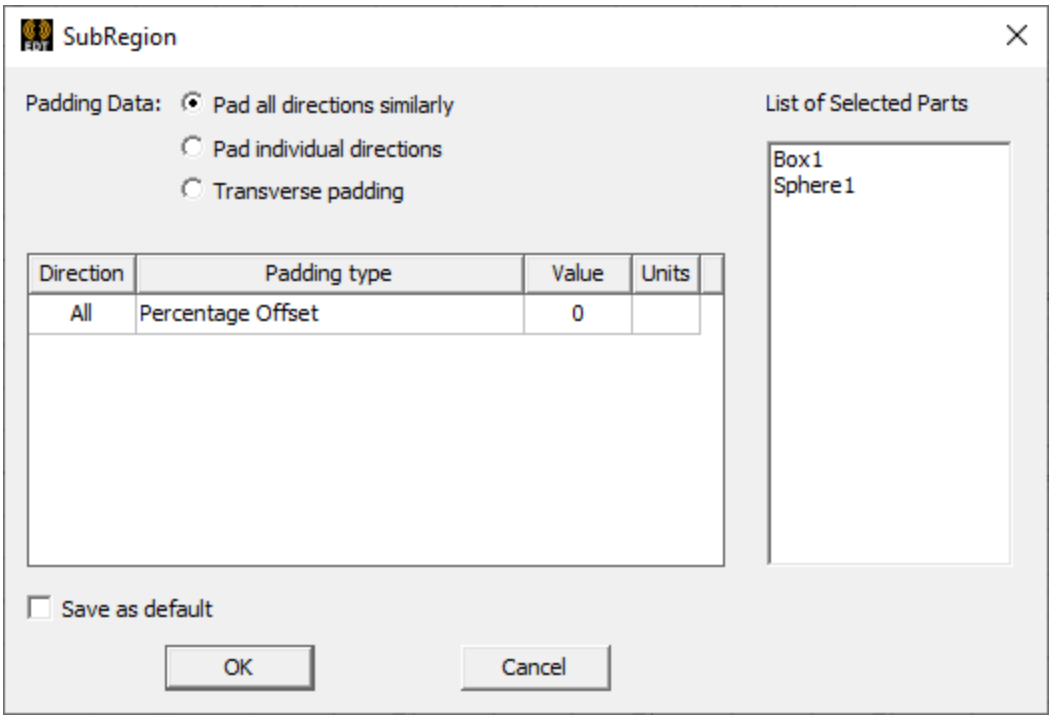
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To draw a sub-region in the current project:

1. Select the parts of components that you want to include in a subregion.
2. From the menu bar, click **Draw** > **SubRegion**:



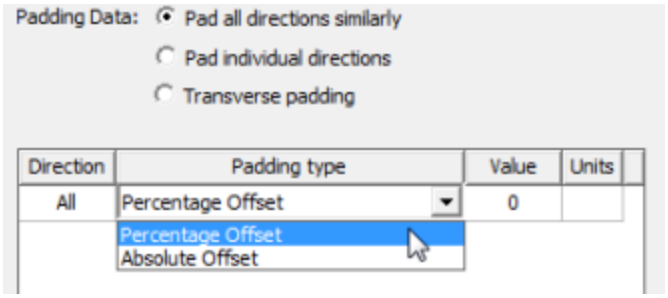
This displays the **SubRegion** dialog box.



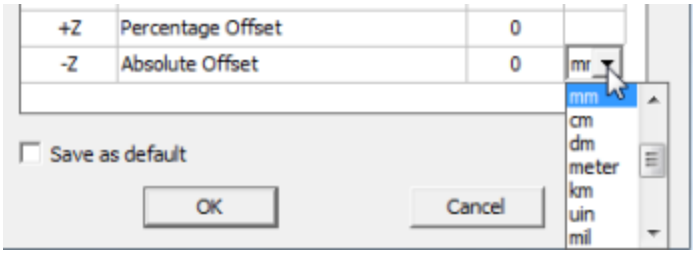
3. For the Padding data, click the Padding Data radio button as **Pad all directions similarly, Pad individual directions**, or **Transverse padding**.

   Selecting **Pad all directions similarly** leaves the Padding type field as requiring a single value that affects all directions. In this case, you can specify the Padding type by selecting **Percentage Offset** or **Absolute Offset** from the drop-down menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

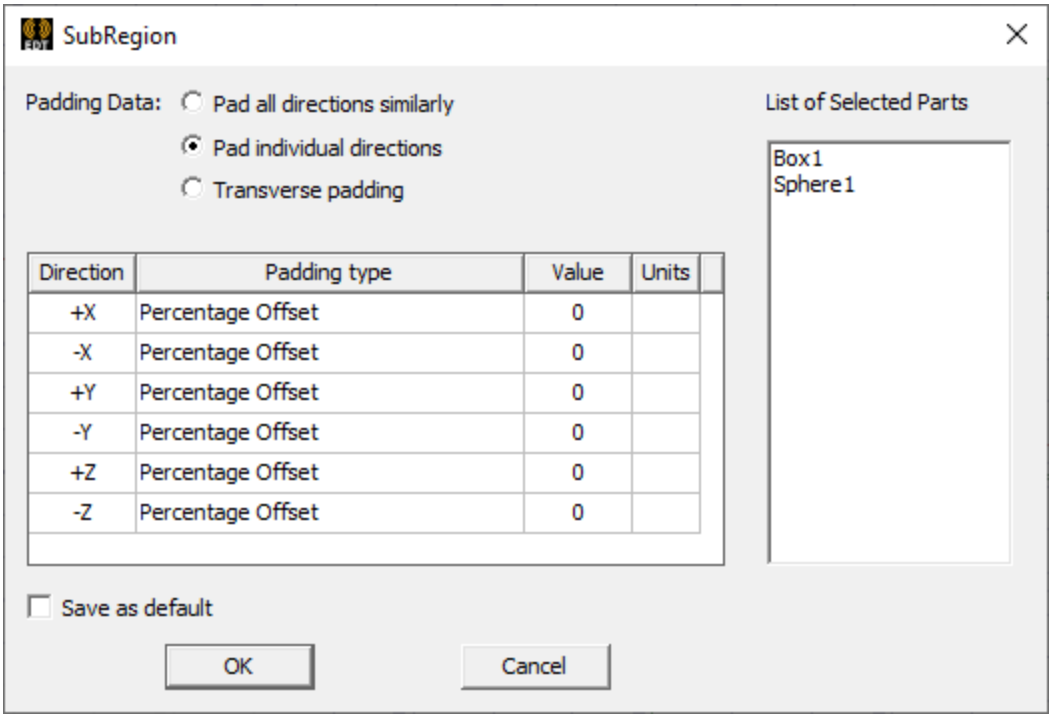


If you select **Absolute Offset**, you also specify the units by selecting from a drop-down menu.



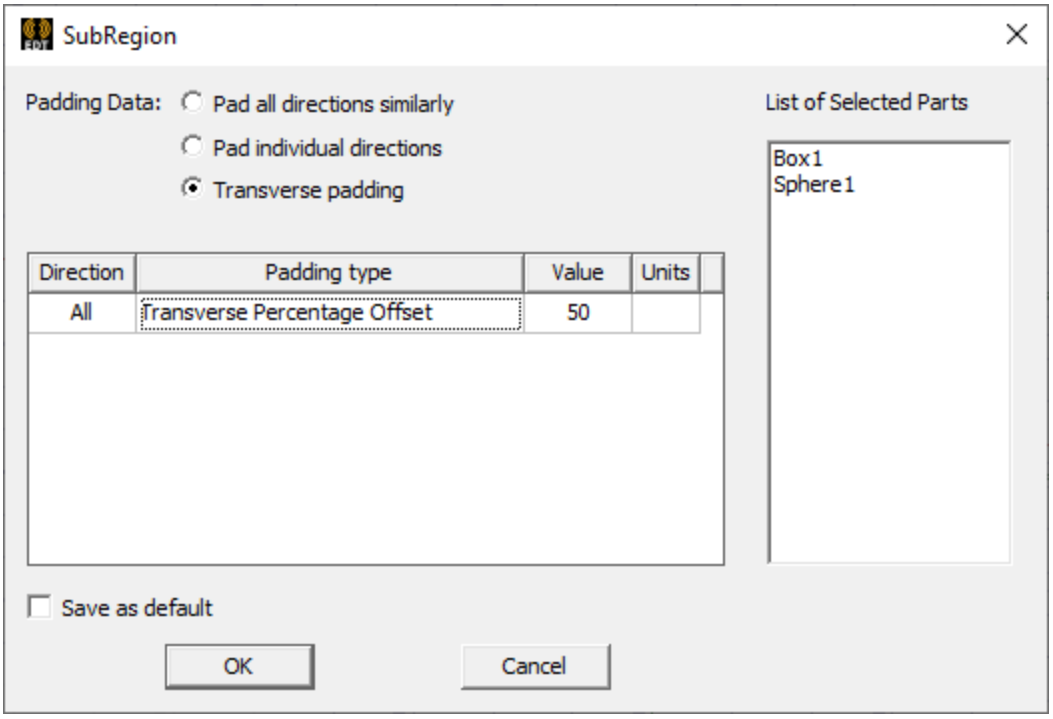
Selecting **Pad individual directions** displays the Padding Percentage as a table of Positive and Negative X,Y, and Z coordinates, permitting you to specify padding for each direction. In this case, you can specify the Padding type by selecting **Percentage Offset** or **Absolute Offset** or **Absolute Position** from the drop-down menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



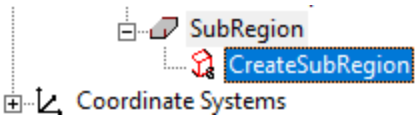
Selecting **Transverse padding** means that padding in one direction is controlled by dimensions in other two directions. Each direction (for example, X) is padded with user-specified percentage of diagonal length of the other two directions (Y and Z).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Specify the Padding values in the fields and select the units from the drop-down menu.
5. If desired, select the check box to save the values as Default.
6. Click **OK** to close the dialog and create the sub-region.

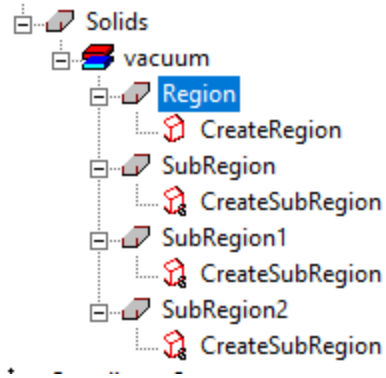
The sub-region is drawn, selected, and displayed in the History tree. It is created using the current coordinate system.



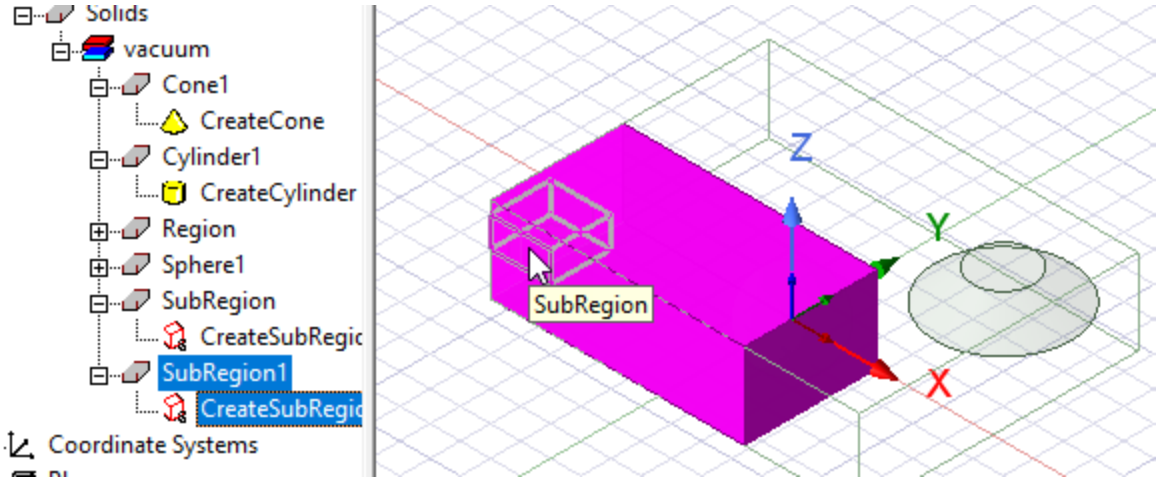
Additional SubRegions and Nested SubRegions.

If you create additional sub-regions, these appear in the History Tree.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



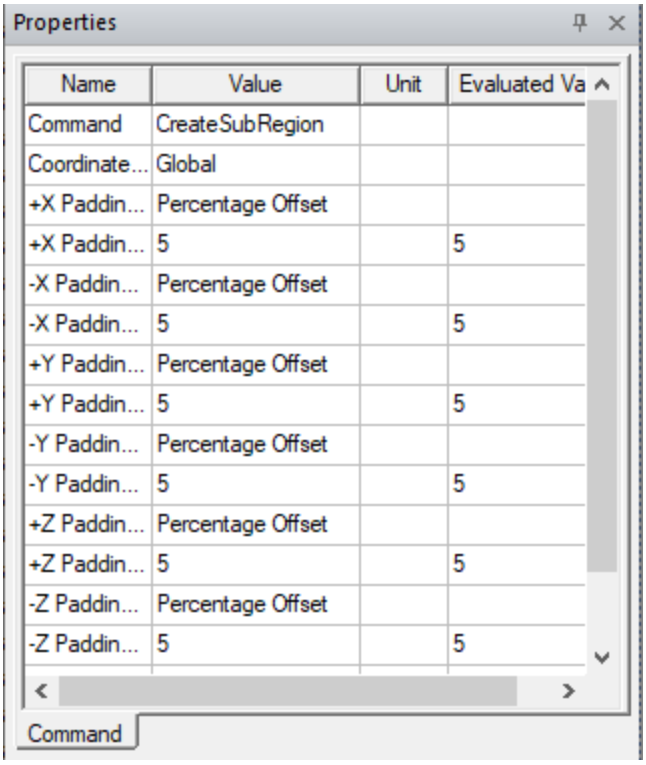
SubRegions can contain other SubRegions, that is, SubRegions can be nested.



Properties for SubRegions

The Properties dialog for the Create SubRegion icon in the History tree shows the coordinate system and Padding values.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Properties dialog for the selected model lists attributes tab including Name, Material (Default, vacuum), Solve inside, Orientation, Model, Color, Display Wireframe, and Transparency. You can edit all of these values.

You can use modeling operations on these sub region parts with following rules

- Geometry of non-subregion parts should not depend on sub-region parts. That is, sub-region parts should not be allowed to be used as a tool, not even after cloning them
- Simple local operations like face move, fillet, and arrange operations should be allowed on sub-region parts.

Global region should take the sub-regions into account in its computation.

Sub-Regions can be included in components.

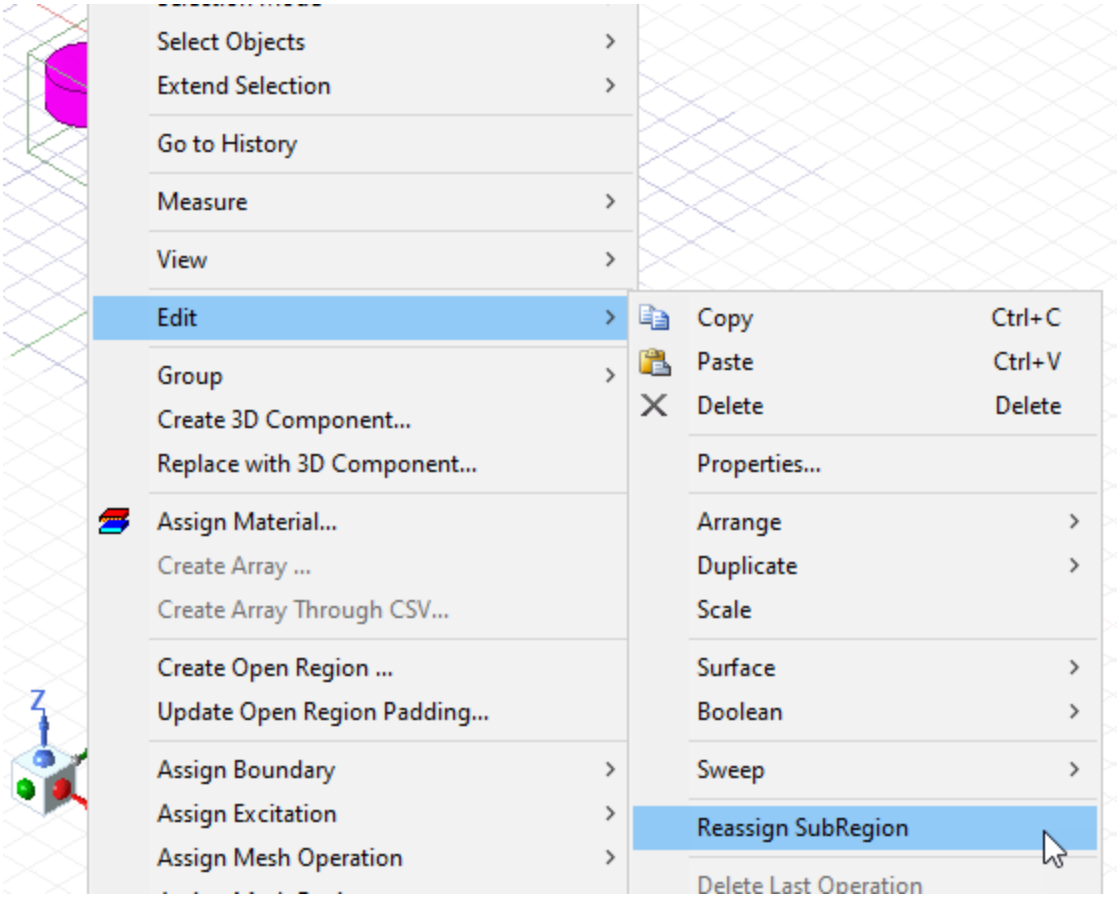
Reassigning a SubRegion

The Modeler allows you to reassign a subregion that encompasses a set of objects inside previously defined region. Like regions, subregions automatically resize if the objects contained in these subregions change.

To reassign a sub-region in the current project:

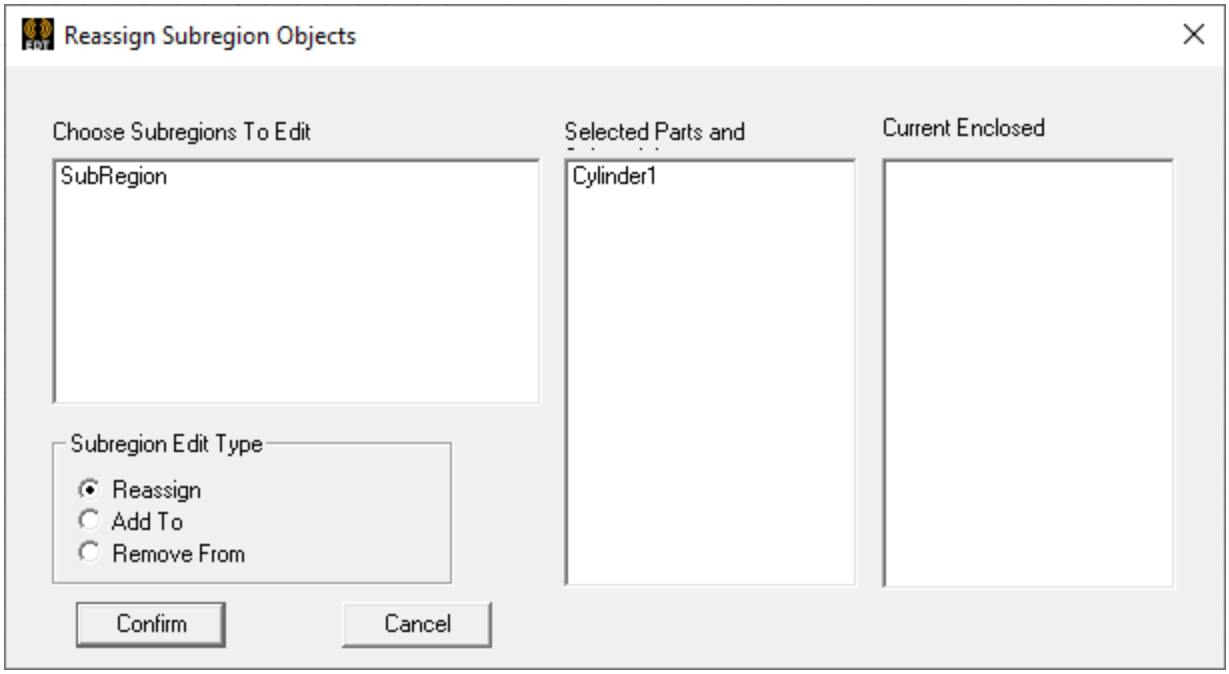
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the components or parts that you want to assign to a subregion.
2. Right-click in the Modeler window to invoke the shortcut menu and click **Edit** > **Reassign SubRegion**:

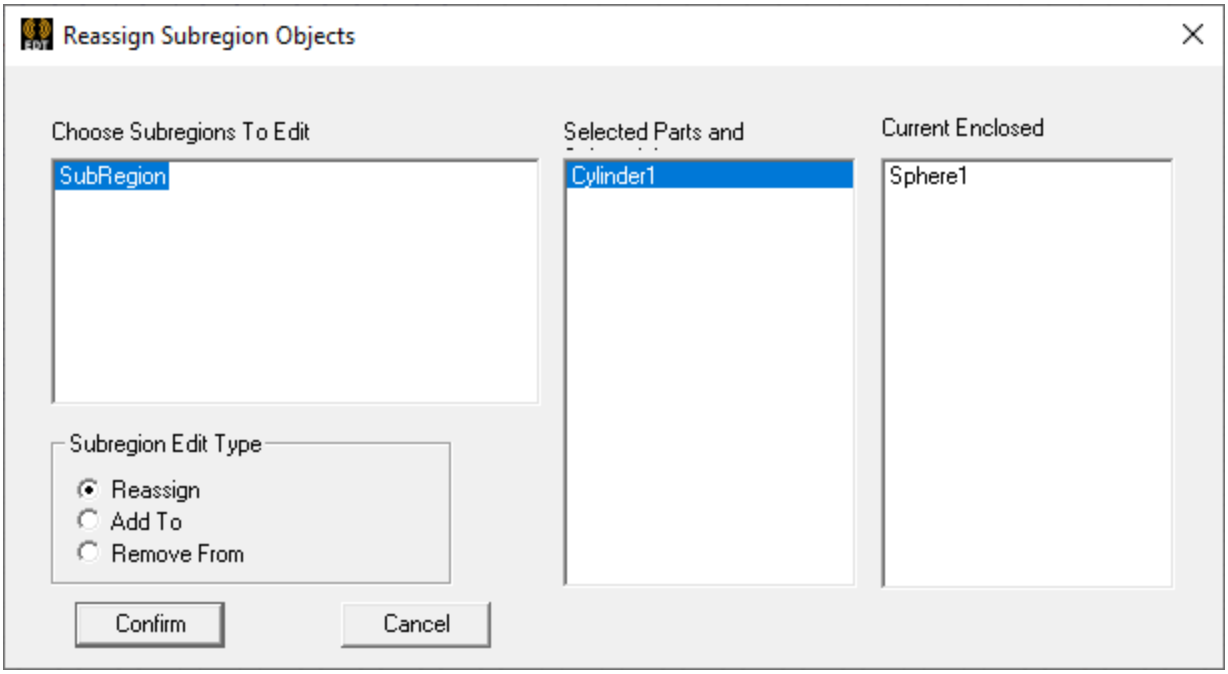


This displays the **SubRegion** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



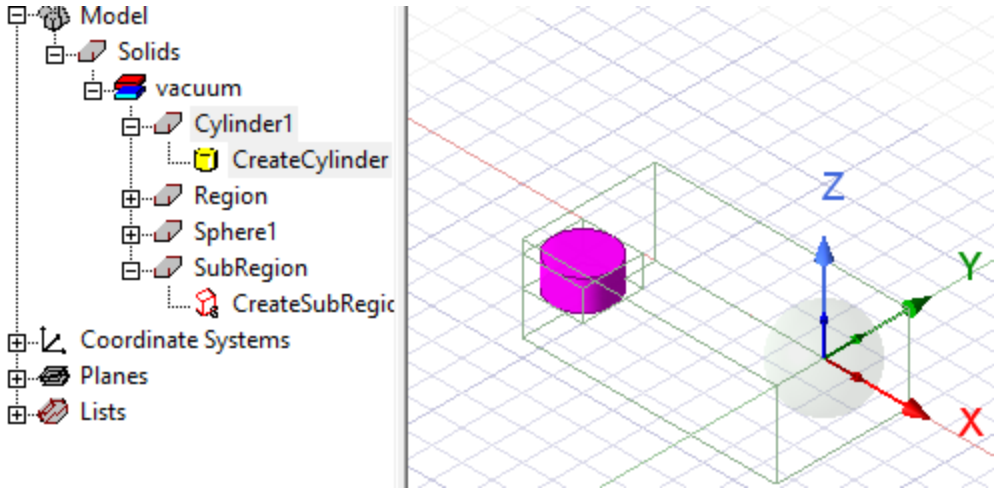
3. Select the SubRegion to Edit and the Parts to Reassign. If there are multiple SubRegions, these will be listed. When you have selected a SubRegion, the Currently Enclosed components are listed.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

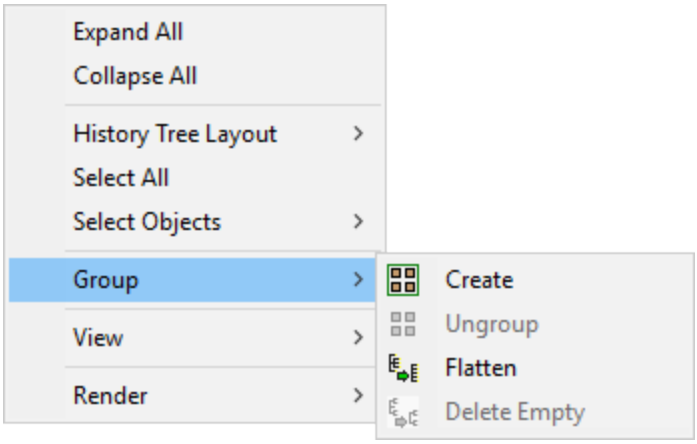
4. Specify whether to Reassign the selected parts to the SubRegion, or Add To the SubRegion, or Remove from the SubRegion.
5. Click **Confirm** to close the dialog complete the reassignments.

The sub-region is redrawn, selected, and displayed in the History tree. It is created using the current coordinate system.



Group Commands for Modeler Objects

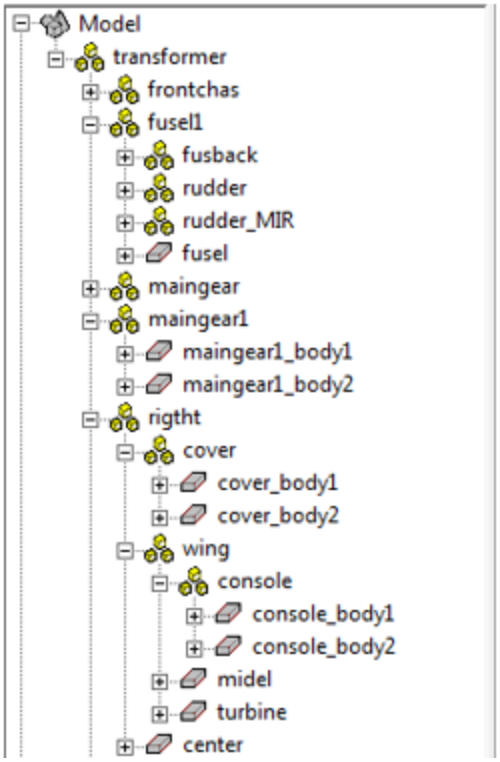
The 3D Modeler allows you to group objects in the History tree. Besides predefined folders under **History Tree Layout** for solids, sheets, and material assignments, a set of **Group** commands allows you to create groups, ungroup objects, flatten a selected group's hierarchy, and delete empty groups. Groups are not compatible with 3D components, although, you can organize 3D components by definition in the History tree.



The **Group** commands support the following features:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Group objects in the history tree. When objects are grouped, they show up under a sub-item in the history tree.
- Groups can contain sub-groups and sub-groups containing further sub-groups.
- Groups permit moving objects from one group to any other group at any time.
- Groups are purely for organization of history tree. They do not affect solution in any way.
- Groups permit you to bring in MCAD assemblies and sub-assemblies as groups. See: Importing 3D Model Files.



- Groups can contain objects, submodels (for example, UDM, 3D Component, CAD links) and groups. Coordinate systems, planes etc will not have any parent group.

Object groups have following important distinctions from 3D components:

- Groups do not encapsulate history of objects. They do not encapsulate parameters used by those objects.
- Groups do not have history tree operations. For example, Arrange operations apply to all objects of the group rather than the group.
- Groups are not independent. Delete of an object in one group could cause object in another group to be also deleted.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Accessing Group Commands

You can access Group commands in several ways: via **Modeler > Group**, via the right-click shortcut menu in the History tree, via the right-click shortcut menu in the Modeler window, and via the Group icons on the Model tab.

Group Command Descriptions

Group > **Create** works with or without a selection. If the History Tree Layout > Show Groups command is disabled, a dialog box reports that groups are currently hidden, and that if you OK to continue, groups will be shown. If there is no selection of objects for new group, an empty group is created under Model. If there are selections, a new group is created under same group as all the selections, provided all selections are under one group. Once a group is created, all selections move under the new group. If selections are under different groups, then a new group is created under Model. Selections could be objects, groups, 3D Components, or User-Defined Model. The **Group** > **Create** command tries to find a more suitable group name where possible, based on names of all selections. If not, default group names are Group1, Group2, and so on. You can edit group names later.

Group > **Ungroup** is enabled only when you select one or more groups. Upon ungroup, all the contents of the selected group move under group's parent and the selected group is deleted. Note that **Group** > **Ungroup** is not recursive; it only affects the selected group is and all its children groups remain intact. Ungroup is also different from **Delete** (under **Edit** > **Delete**), which deletes a selected group and everything under it.

Group > **Flatten** is enabled when you select a group. You can select multiple groups at different levels. All of the selected group's contents (objects, sub-groups, and so forth) come directly under selected group. In a sense this is a recursive ungroup operation as all the children and grandchildren groups of selected group are ungrouped. If the History Tree Layout > Show Groups command is disabled, a dialog box reports that groups are currently hidden, and that if you OK to continue, groups will be shown.

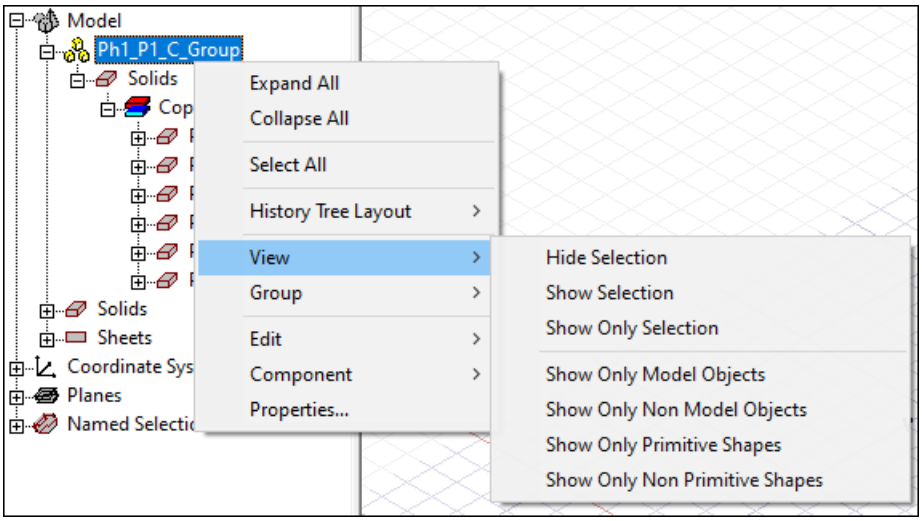
Group > **Delete Empty** deletes all empty groups under the selected group. If a selected group is empty it is deleted. This command is helpful to clean up empty groups after you have moved objects from one group to another. If the History Tree Layout > Show Groups command is disabled, a dialog box reports that groups are currently hidden, and that if you click **OK** to continue, groups will be shown.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Operations on Groups

View Options

For groups, you can hide or show the entire group. You can also opt to show only particular types of objects or shapes in that group. The right-click view options are shown below:



Edit/Delete

Delete operations delete group and all the children, including sub-groups, under the group.

Copy and Paste of Objects in Groups

When you copy and paste objects with groups to another design, group information is carried over, that is, new group in target design is created. If a group with the exact name exists, then the objects are moved under that group and no new group is created. When pasted in the same design, the pasted object moves under same parent group as original object.

Copy and Paste of Groups

When you copy and paste one or more groups in same design or to another design, a new group is created with unique name derived from original group name. Everything under that group, including any sub-groups, is copy/pasted as well.

Copy and Paste of Groups and Objects

If you multi-select a few objects and groups, all of them are pasted correctly. If you select a group and few objects under that group, copy and paste includes the complete group structure. This means that selection of objects under the selected group is ignored. If you want to only copy and paste few objects from group, you should only select those objects and NOT select the group.

Arrange Operations on Groups

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can select a group to enable arrange operations like move, rotate and mirror. Note that arrange operations works directly on objects under selected group. So after arrange operation is done, the History tree shows the arrange operation under every object of that group.

Duplicate Operations on Groups

You can select groups to enable duplicate operations. Afterward, the History tree shows a duplicate operation under every object of that group. Newly created objects appear under a new group. There will be new group for every instance of duplicate.

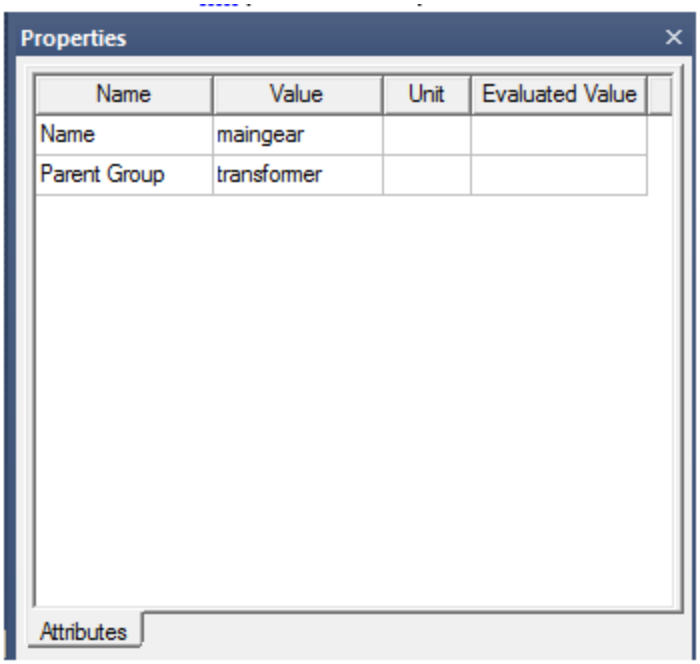
Other Operations on Groups and Objects in Groups

Modeling operations that create new objects, such as Create from Face or Edge, as well as Simplify and Separate are available for groups. The newly created objects appear under a new group created under the original object's group, based on the tool option settings. See 3D Modeler Options: Group Options.

If the group has hierarchy, you can also flatten it.

Group Properties

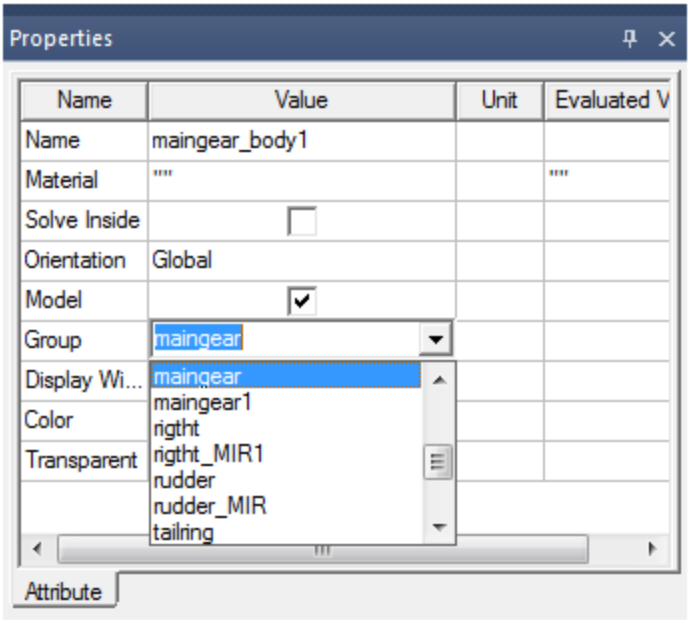
A Group's Property window is shown when you select a group in the history tree. The Group's properties include Name and parent Group.



Group Property Window

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Object, group and submodel properties have a Group property. You can edit the Group Property and this provides another way of setting an object's (or group's) group. You can select any of the existing groups listed or create a new group.



Group assignment could be changed in Object's property window

Model Analysis

For some models it may be beneficial to remove unnecessary small entities and to fix object misalignments to avoid potential mesh issues. The Modeler window includes Model Analysis functions to help you evaluate models you have imported or created. Select **Modeler** > **Model Analysis** to see the menu options. Depending on the design and the current selection, some features may not be enabled. The menu includes the following commands.

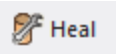
- **Analyze Objects**
- **Analyze InterObject Misalignment**
- **Analyze Surface Mesh**
- **Show Analysis dialog**

> **Note:**
>
> Before running model analysis, you must remove all command history for the selected object by using the Purge History command. If you need to save the object history, save a separate copy.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After import, you typically perform validation check. This lets you focus on objects and object pairs that have errors and or warnings. The objects that fail should be analyzed by using the **Modeler** > **Model Analysis** > **Analyze Objects** menu item.

1. Select the objects and invoke **Modeler** > **Model Analysis** > **Analyze Objects**; or, on the **Model** ribbon, click **Heal**.

   This displays the **Analysis Options** dialog box, which allows you to specify settings for entity check level and small feature detection.

   When you **OK** this dialog box, the initial analysis executes and the **Model Analysis** dialog box is displayed.

2. Choose the objects that have "Invalid Entities Found" and **Perform** > **Heal Objects**.

   In most cases, the objects will be healed and the errors fixed.

3. If errors still persist, choose the edges and faces and click on **Delete**.

This will replace the selected face/edge object by a tolerant edge/vertex respectively. In some cases the replacement of the face/edge by tolerant edge/vertex will fail.

When models pass the initial validity checks, mesh generation could still fail. The following errors can be present in models: (See Error Detection.)

- Non-manifold topology – These are non-manifold edges and vertices that are present in the model.
- Object pair intersection – This detects whether pairs of objects intersect.
- Small feature detection – small edge length, small face area and sliver face detection.
- Misaligned entities detection – detects pairs of faces from objects that can be aligned to remove object intersections. This improves the probability of mesh success.
- Mesh failure error display – This is available for single object, object pairs and last simulation run (all objects in a model). Errors reported by the meshing module are reported to the user.

Small feature errors and misaligned entities should be resolved before you invoke the meshing for the model.

By default, the **Heal** command is automatically applied to imported objects.

Analysis Options Dialog Box

To perform analysis on an object according to specified features and tolerance values:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the object you want to analyze and click **Modeler** > **Model Analysis** > **Analyze Objects**.

   This displays the **Analysis Options** dialog box, with the **Analysis Options** tab selected. Selecting **Modeler** > **Model Analysis** > **Heal** also displays this dialog box. If, during **Modeler** > **Import...** you select **Heal Imported Objects** and **Manual**, you also see this dialog box.

2. If desired, check the **Perform Entity Check Errors** check box.

   This enables the **Check Level** menu. The setting can be **Basic**, **Strict**, or **Comprehensive**. See: Modeler Validation Settings.

3. If desired, click the check boxes to enable and set the **Detect Feature** settings:
   - **Detect Holes**, and specify the Maximum Radius.
   - **Detect Chamfers**, and specify the Maximum Width.
   - **Detect Blends**, and specify the Maximum Radius.
4. If desired set the **Detect Small Entities** features and tolerance values.
   - **Small Edges**, length less than
   - **Small Faces**, area less than
   - **Sliver Faces**, which enables:
     - Object Bounding Box Scale Factor
     - Sliver Edge Width
5. Click the **Properties** tab to see a listing of the geometric properties of the selected object.
6. Clicking **OK** on this dialog displays the **Model Analysis** dialog box, which contains the results of the analysis.

Analyzing the Surface Mesh

To set the options to analyze the surface mesh:

1. Select an object of interest.

   This enables the Analyze Surface Mesh command in the menu.

2. Click **Modeler** > **Model Analysis** > **Analyze Surface Mesh**.
   The **Surface Mesh Analysis Options** dialog box appears. This dialog box allows you to set parameters to remove.

   You can also open the **Surface Mesh Analysis Options** dialog box from the **Model Analysis** dialog box via the **Perform** drop-down menu on the **Objects** tab.

3. Select or clear the **Perform Object Pairs Analysis** check box. Selecting this option evaluates the mesh for all combinations of the selected objects.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Select or clear the **Ignore Objects Separated by greater than** check box, and enter a value in the text box. Selecting this option means that object pairs are disregarded from analysis if their separation is greater than the specified value.
5. Click **OK** to perform the analysis with the selected options.
   The **Model Analysis** dialog box appears, displaying the results of the analysis.

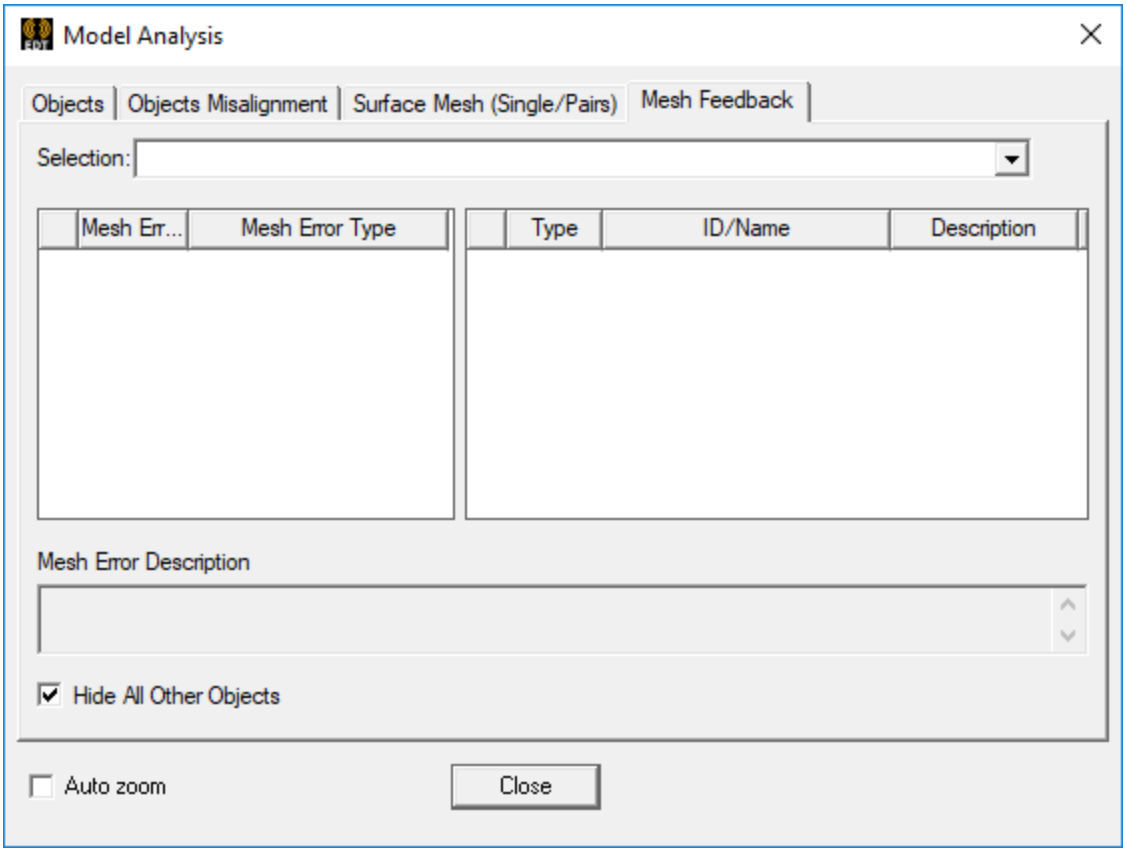
Model Analysis Dialog Box

This dialog box contains results for all model analysis, including diagnostic information relating to mesh issues. To view the analysis options:

1. Click **Modeler** > **Model Analysis** > **Show Analysis Dialog**.
   A submenu appears.
2. Select one of the following from the submenu:
   - **Objects**
   - **Objects Misalignment**
   - **Surface Mesh**
   - **View Mesh Feedback**

   The **Model Analysis** dialog box appears (This dialog box also appears automatically after clicking **OK** in the **Analysis Options** dialog box).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. You can select **Hide All Other Objects** to suppress display of objects, faces, or surfaces in the Modeler window not selected in the **Mesh Feedback** panel
4. Select the **Auto zoom to selection** check box to automatically zoom to the item selected on the **Objects** tab.
5. Make the desired changes on each tab in the **Model Analysis** dialog box.
    - **Objects** tab
    - **Objects Misalignment** tab
    - **Surface Mesh (Single/Pairs)** tab
    - **Mesh Feedback** tab
6. Click **Close** to close the **Model Analysis** dialog box.

Objects Tab

All results relating to model analysis of specific objects are presented under the **Objects** tab.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. The results table contains the following information.
    - **Object** – column listing the objects in the current design.
    - **Last Analysis status** – column giving the analysis status of the listed objects. Objects can have the following status:
        - **Good** – the object contains no invalid geometry entities given the tolerance values specified in the **Analysis Options** dialog box.
        - **Null Body** – the object is non-existent.
        - **Analysis not performed** – the object was not selected for analysis.
        - **Invalid entity errors** – these are api_check_entity() errors and non-manifold errors which Ansys EM recommends that you fix before meshing.
        - **Small entity errors** – small faces, sliver faces and small edges that are optionally detected based on the tolerance limits specified in the Analysis Options dialog box.
2. Select any object name in the table which contains errors to display a table with a list of Error types, corresponding Entity List IDS for associated faces, edges and vertices, and error descriptions. Selecting the row for each error causes the Modeler window to highlight the associated entity.

    > **Note:**
    >
    > **Auto Zoom to Selection** – if this option is checked, the modeler automatically zooms to the item selected in the **Model Analysis** dialog box.

3. Select the face, edge or vertex entity from the list to view the error description in the **Description** field.
4. Select the **Delete** button if you want to remove a selected face or edge entity.
5. Select the **Perform** button to list the commands that you can execute on the selected objects in the Results table.
    - **Heal Objects** – repairs invalid geometry entities for the selected objects within the specified tolerance settings. The **Healing Analysis** dialog box will appear.
    - **Analyze Objects** – evaluates the object status. Selecting this displays the Analysis Options dialog.
    - **Analyze Surface Mesh** – invokes a mesh for each selected object and reports analysis results under the **Surface Mesh (Single/Pairs)** tab. Selecting this option displays a dialog with radio buttons to select.
        - **Perform Object Pairs Analysis** – evaluates mesh for all combinations of the selected objects.
        - **Ignore objects separated by greater than a specified value** – object pairs are disregarded from analysis if their separation is greater than the specified

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

value.

- Click **OK** to perform the analysis with the selected options.

- **Analyze Interobject Misalignment** – determines any misalignments between two selected objects in the results table. The results are reported under the **Objects Misalignment** tab.
- **Display Healing Log** – checking this causes the **Model Analysis** dialog box to display a healing log which includes information about operations performed on an object during the healing process.

Surface Mesh (Single/Pairs) Tab

The panel displays the results of a surface mesh analysis.

1. You can display results for
   - Individual Objects
   - Object Pairs

   > **Note:**
   >
   > **Auto Zoom to Selection** – if this option is checked, the modeler automatically zooms to the object or object pair selected.

2. The results table contains the following information:
   - **Object** – column listing object name or a pair of object names.
   - **Last Analysis Status** – column stating the meshing status of the object or object pair.
     - Mesh Success
     - Mesh Failure
   - **Error Type** – this column gives the category of error that caused the mesh failure.
   - **Error Detail** – provide specific geometry information regarding mesh error location.

   Display options include

   - **Display Mesh Analysis** – checking this displays further details concerning each error to be listed.
   - **Auto Zoom to Selection** – checking this causes the modeler to automatically zoom to objects or faces corresponding to the error.

Object Misalignment Tab

The table in this panel displays results of an **Interobject Misalignment** analysis. It contains a list of Alignable Faces, described in a list of Object Sets, and corresponding Misaligned Faces.

All misaligned face pairs corresponding to the analyzed objects are listed in the table.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Align Faces** – select a face pair in the table and click **Align Faces** to align selected faces.

- **Clear All Analysis Data** – this button removes all information from the tables.
- **Display Log for Object Set** – checking this causes the **Model Analysis** dialog box to display a healing log which includes information about operations performed on an object set during the healing process.
- **Auto Zoom to Selection** – if this option is checked, the modeler automatically zooms to the item selected in the table.

After validation check is performed, the pairs of objects that intersect are chosen for analysis. Use the analysis results to find whether objects have faces that can be aligned.

Choose all the bodies that intersect with another body.

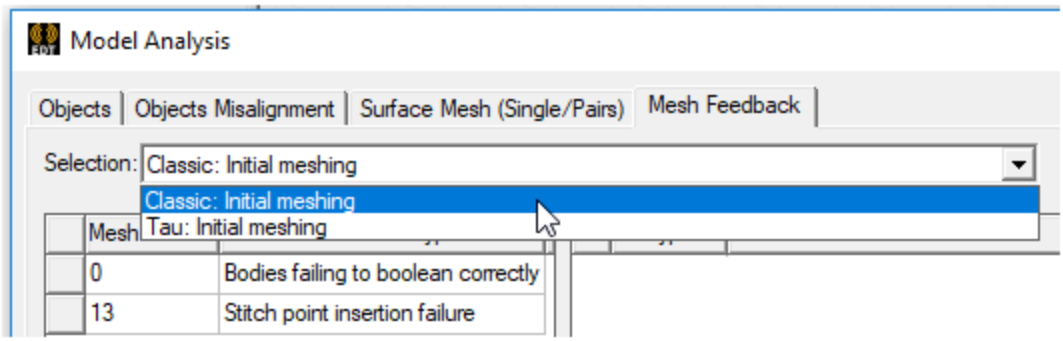
1. From the **Model Analysis** dialog box choose perform/Analyze Interobject misalignment. Or you can run **Modeler** > **Model Analysis** > **Analyze Interobject Misalignment**.

   If the analysis finds object pairs that can be aligned, they will be displayed in the **Objects Misalignment** tab.

2. You can select individual or multiple rows and perform Align Faces. In some cases, face alignment will fail if the topology of the body changes by a large factor after alignment.
3. Identify individual bodies and body pairs that fail to mesh.
4. Perform Mesh analysis on individual objects and object pairs.
5. Review the reports and fix the errors.

Mesh Feedback Tab

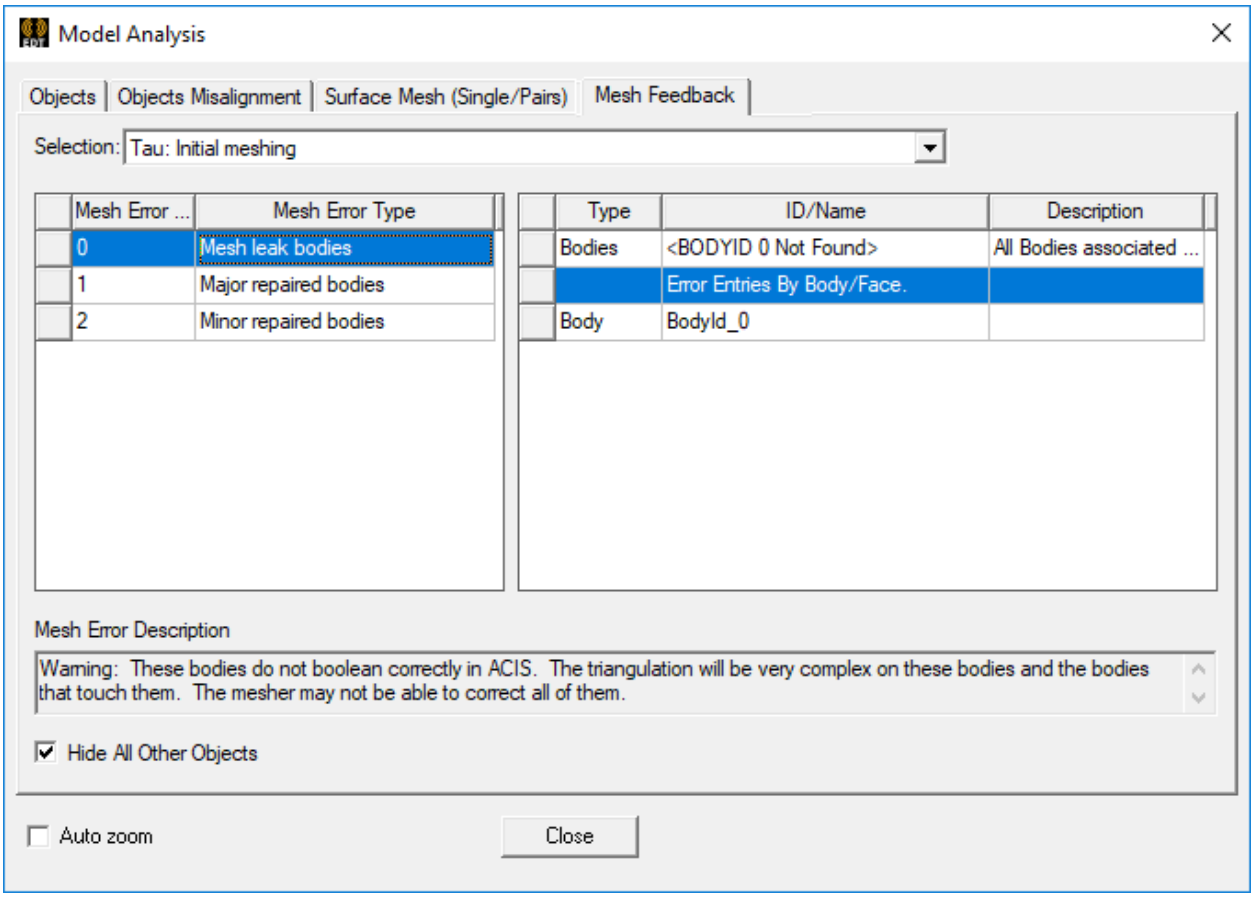
The tables in this panel list model errors as viewed by the mesher. If the project has solutions by different solvers, the **Selection** drop-down list lets you select which to view:



The left-side table has columns for Mesh Error Serial ID and Error Type.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Mesh Error Serial ID** – the 0 item is the body with which the subsequently listed entries are associated.
- **Mesh Error Type** – this column gives the category of error that caused the mesh failure, for example, Non-Manifold Edge, or Point Insertion Failure.



The display in the table on the right side of the **Model Analysis** window depends on your selections from the left hand table. When you select a row in errors panel (the left panel as shown in above figure), all the faces are selected (second row in right panel) and if you enable **Auto Zoom**, the view fits to the errors:

- **Type** – whether the error applies to Bodies, Faces, or Triangles. If you select a particular error type, the columns for Type, ID/Name and Description are filled in.
- **ID/Name** – the object name or object ID for the error type.
- **Description** – the first three rows are for Bodies, Faces, and mesh Triangles. Subsequent rows, are of Error Entities By Body.Face.

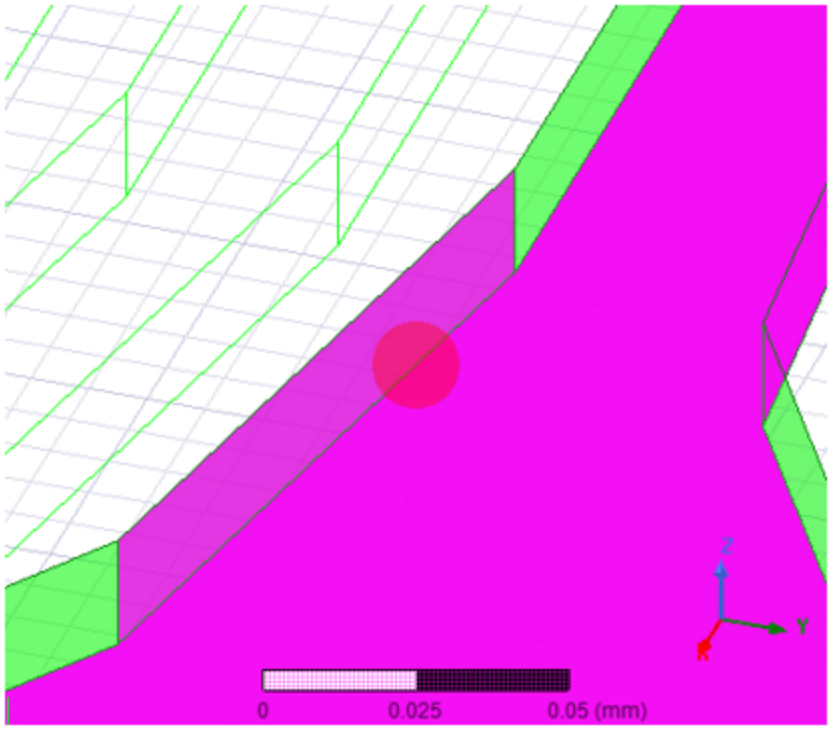
The **Mesh Error Description** field describes error message describing the nature of the selected Mesh Error Serial ID, the implications, and provides a recommended response.

Display options include

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Hide All Other Objects** – hides objects in the Modeler window not associated with the selected Bodies, Faces, or Triangles.
- **Auto Zoom to Selection** – checking this causes the modeler to automatically zoom to objects or faces corresponding to the selected Mesh Error Serial ID (on the left side table) or the ID/Name (on the right side table).

Use of these selections let you more easily view and respond to the errors. If you enable **Auto Zoom**, any selection whether it is body(ies), face(s) or error triangle(s) or error segment(s) will fit to view. If all the faces are also selected (second row in right panel), then errors and selected entities will fit to view.



The error shown in the figure above is a contact error between the faces. Focusing only on the errors when an error is selected in right panel, improves visualization of these meshing errors and provides you with precise information in order to take corrective action quickly.

Mesh Error Markers

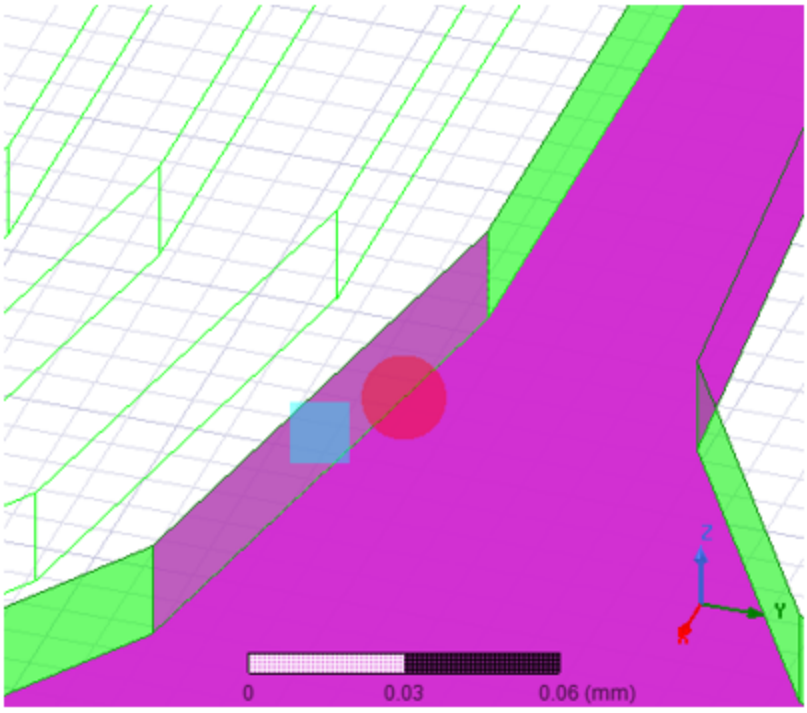
As shown above the mesh errors are marked by ‘Red Circles’. The following table describes all such markers:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Mesh errors
	Mesh warnings
	Mesher information
	Selected Faces

These markers are located at the center of errors or center of faces and their size correspond to the maximum side size of the entity, such as error triangles, error segments or faces. The size of these markers is clamped to a maximum and a minimum size. Additionally they have some transparency. The size and transparency of these markers cannot be changed by the user.

If you select an individual face in the right panel, then the face is marked using a blue square as shown below:



If many individual faces are selected in the right panel, each face is marked by a blue square. In order to find which blue square corresponds to which face in the right panel in the dialog box,

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

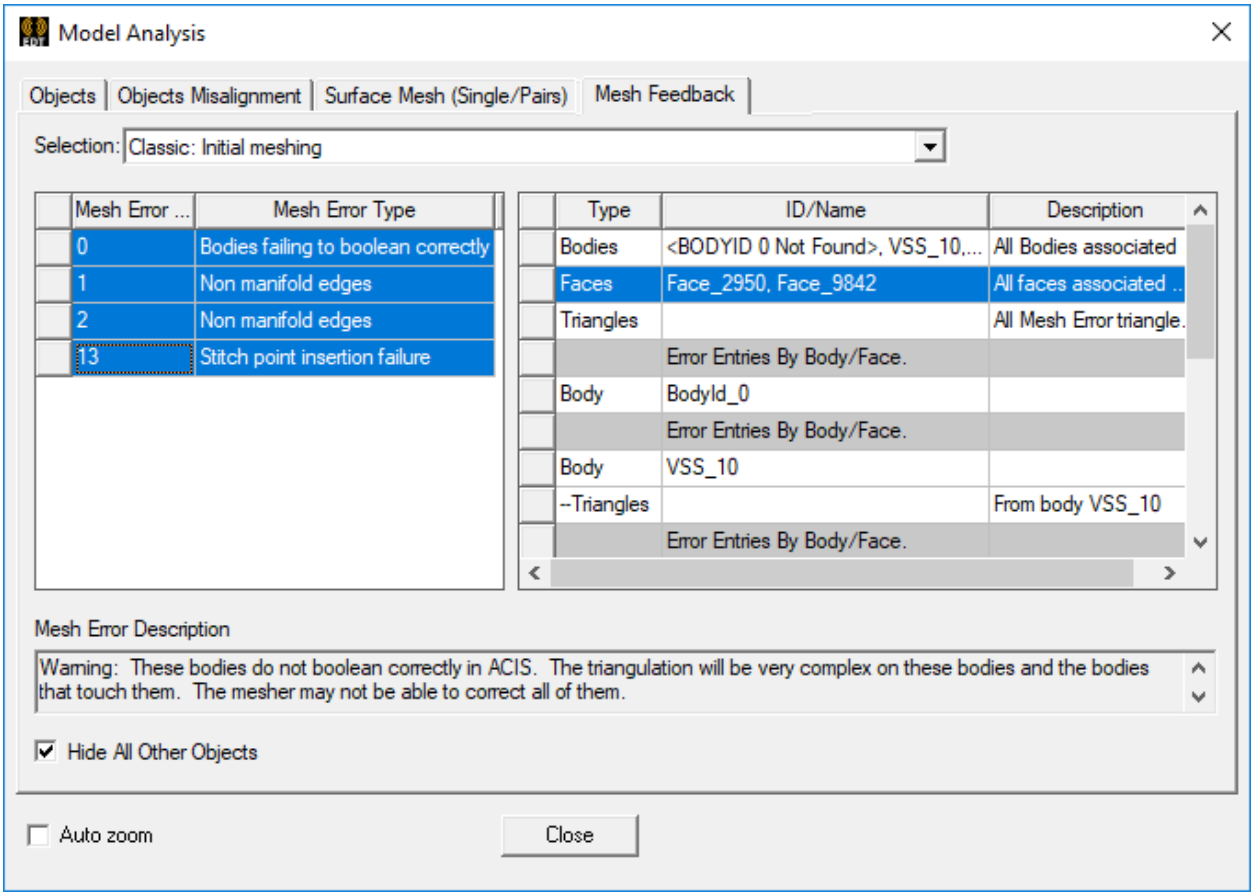
click on the blue square. This action selects the corresponding face row in the right panel of the dialog.

Face Selection

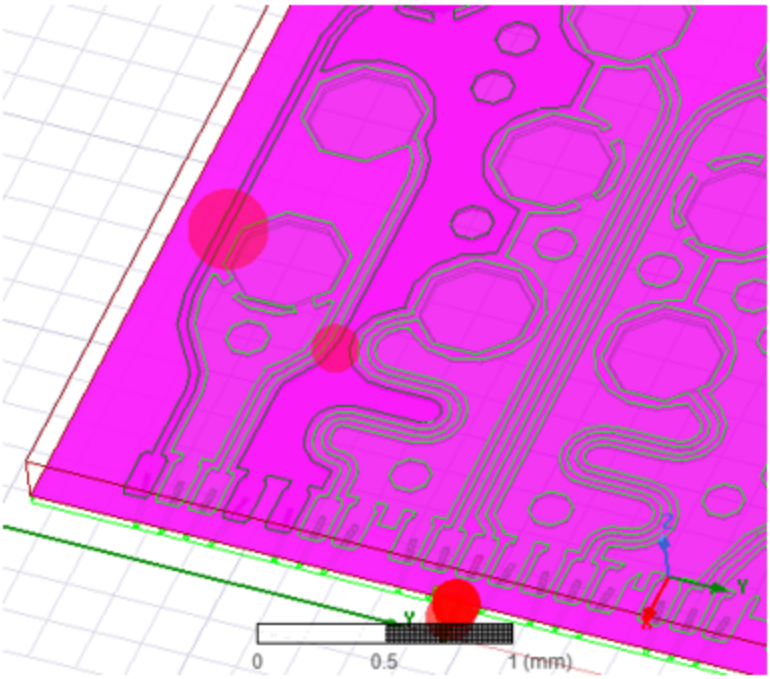
When the all faces row or individual face rows are selected, the Modeler selects the face(s). This enables the face related commands in the Modeler window. Also when you select faces this way, it makes the corresponding bodies of the selected faces visible, even when 'Hide all other objects' is enabled. This provides some contextual information about the face location with respect to its body.

Multiple Selection of Mesh Errors

You can select multiple rows in the right pane of the **Mesh Feedback** tab. To view all the meshing errors, select all the rows and enable **Auto Zoom**, as shown in figures below:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



With **Auto Zoom** enabled, the view fits to all the errors, providing an overview of all the mesh errors, as shown in above figure.

Also, when multiple errors are selected, right panel shows the merged information about all bodies, faces, error triangles etc. And Mesh Error Description text box shows description of all the errors.

Model Preparation Commands

Use the **Modeler** > **Model Preparation** commands to prepare a model.

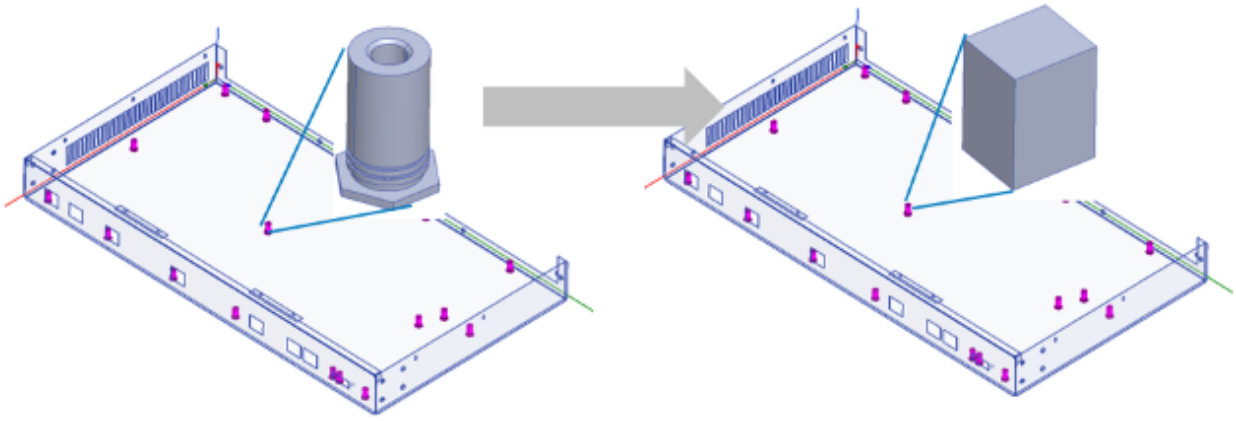
These commands include:

- Simplify
- Heal
- Stitch Sheets
- Align Faces
- Remove Faces
- Remove Edges
- Perform Explicit Subtractions

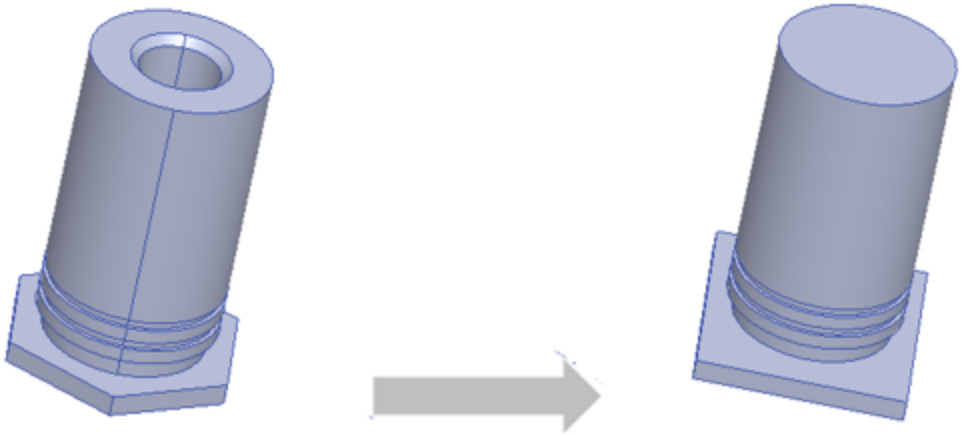
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Simplify Command

Use the **Modeler** > **Model Preparation** > **Simplify** command to converts a complex MCAD object into simpler primitives which are easy to mesh and solve. The operation can be applied on any selected object and not just imported objects. You can specify the type of simplification as Bounding Box, Primitive Fit, or Polygon Fit.



Bounding Box Simplification

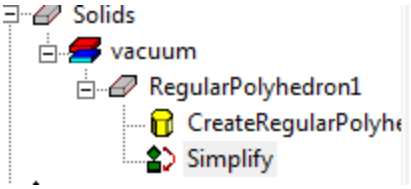


Primitive Fit Simplification

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

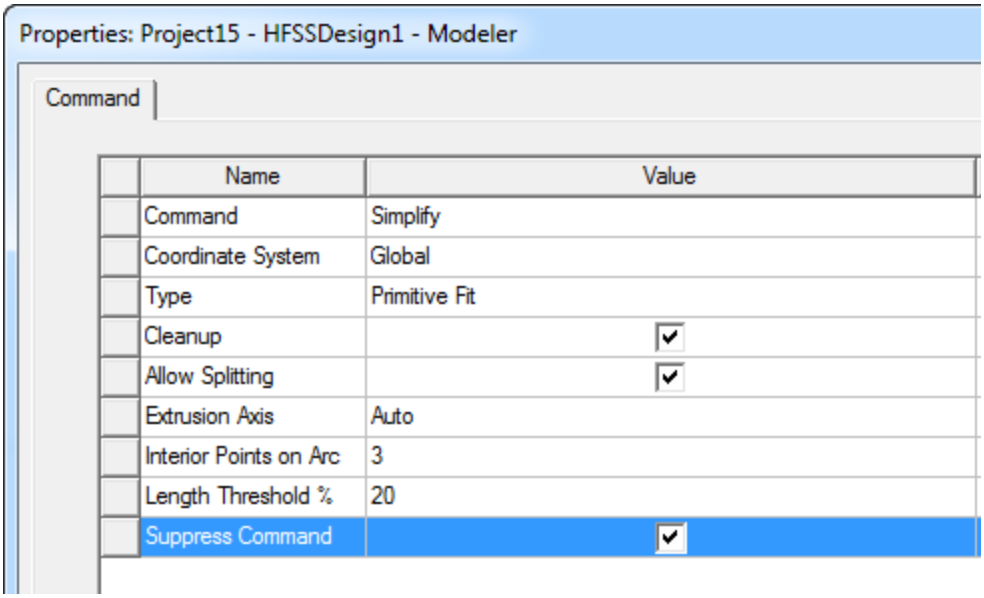
Polygon Fit Simplification

The History tree shows the **Simplify** command as being applied to the object.



Any parameters you specify in the dialog you can also edit in the Properties dialog for the History tree **Simplify** command to get a different simplification. The properties display in the docked properties window and include only the parameters that are used by your currently selected **Simplify** command type. None of the parameters of simplify operation accept variables.

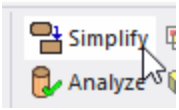
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



1. Select an object.

   This enables the Simplify command on the **Modeler** > **Model Preparation** submenu.

2. Click **Modeler** > **Model Preparation** > **Simplify** or on the **Model** ribbon, select **Simplify**.



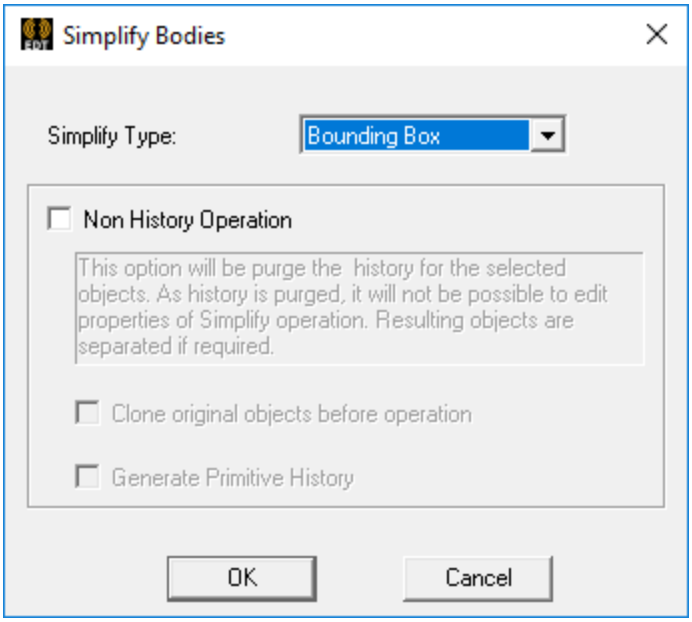
This opens the **Simplify Bodies** dialog box. A drop-down menu lets you select the degree of simplification, from Bounding Box, Primitive Fit, or Polygon Fit.

Bounding Box Simplify Type

If you select Bounding Box as the Simplify Type, there are no additional parameters. The object is replaced by its exact bounding box as computed in operation's coordinate system. You can change the operation CS to get a bounding box in an appropriate orientation. For the **Non**

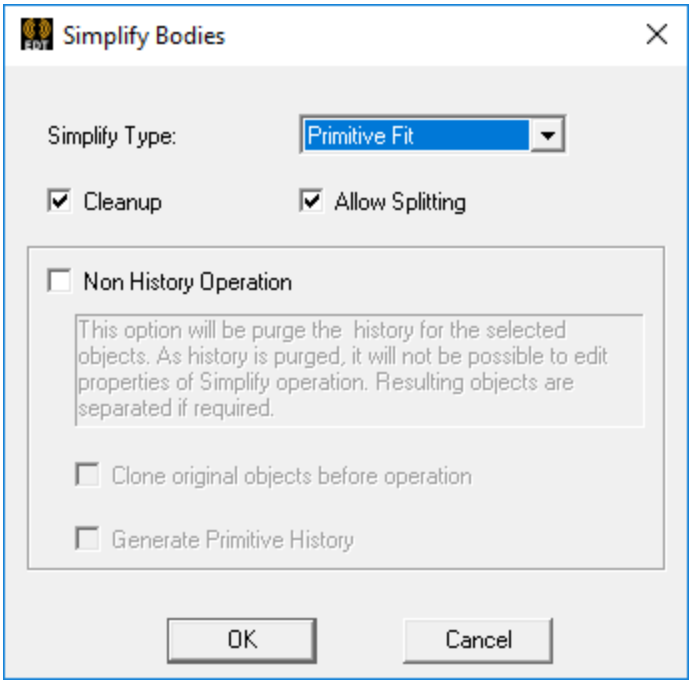
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

History Operation options, see below.



Primitive Fit Simplification

For Primitive Fit simplification the object is replaced by a set of primitives shapes like Prism, Cylinder, Cone and so forth. This simplification type typically produces the simplest geometry with highest number of primitives. Primitive fit has the following options: Cleanup and Allow Splitting, whether to Separate Bodies and Purge History, and if so, whether to clone original objects before the operation.



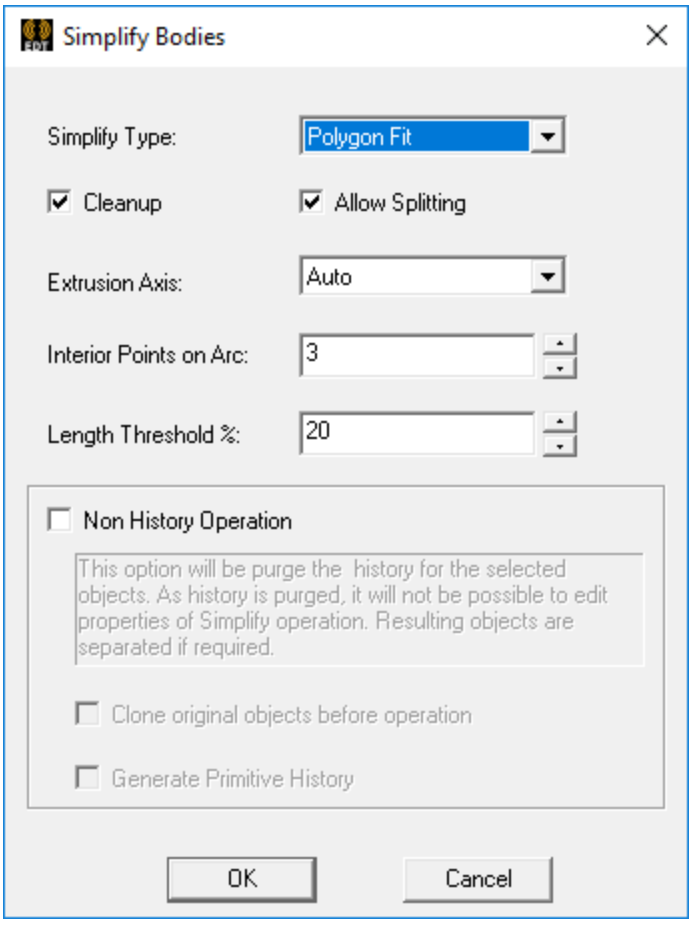
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For Primitive Fit, the parameters are:

- **Cleanup** - This option allows you to clean the model before performing simplification. Cleanup include removing small features like rounds, fillets, chamfers in a solid body. It is recommended to set this option to get more simplified result.
- **Allow Splitting** - This option controls if object should be split during simplification. If this option is selected, complex object will first split into multiple pieces and each piece will be further simplified. Setting this option will result in an object which will match more closely with original object. It is recommended to set this option.
- For the **Non History Operation** options, see below.

Polygon Fit Simplification

For Polygon Fit simplification, the object is replaced by set of polygon swept along normal and other primitives like box and cylinder. This simplification type typically produces geometry closest to original object with highest number of primitives. If you select Polygon Fit as the Simplify Type, the parameters are Clean Up, Allow Splitting, Extrusion Axis, Interior Points on Arc, and Length Threshold. For the **Non History Operation** options, see below.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

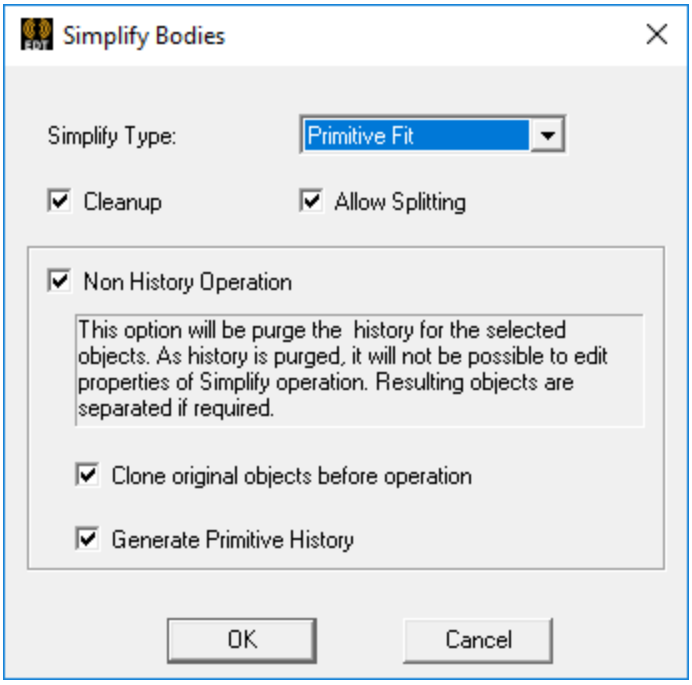
For Polygon Fit, you specify parameters for

- **Cleanup** - This option allows you to clean the model before performing simplification. Cleanup include removing small features like rounds, fillets, chamfers in a solid body. It is recommended to set this option to get more simplified result.
- **Allow Splitting** - This option controls if object should be split during during simplification. If this option is selected, complex object will first split into multiple pieces and each piece will be further simplified. Setting this option will result in an object which will match more closely with original object. It is recommended to set this option.
- **Extrusion Axis**, whether Auto selected, or X, Y, or Z. - This option specified a normal plane in which polygon profile is looked for. The default is Auto which allows an algorithm to determine best possible plane for polygon profile. You can specify a axis to help the algorithm, particularly when there is a draft or chamfer in the extrusion direction. The polygon profile is then swept in the extrusion direction.
- **Interior Points on Arc** - This specifies number of interior points created when a curve on the polygon profile is represented by line segments. The range is 1 to 100 with default being 3. Total number of points used to represent a curve is 5 when number of interior points is 3. This number specifies the maximum number of interior points to add. The number of points added could be less if the Length Threshold is already met.
- **Length Threshold %** - This specifies the length of edge as percentage of maximum length of an edge in the profile. It is specified as percentage of maximum edge length. If length of edge is more than the specified threshold, more interior points are added until the number of “Interior Points on Arc” is met.
- For the **Non History Operation** options, see below.

Non History Operation Options

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Non History Operation options are available for all Simplify Types.



With this option selected, the input body is simplified into multiple primitives (or polygon shapes) and a new part is created for each simplified shape. This option also purges the history of original part and it is not possible Edit Properties of the Simplify operation. You can choose to clone original object before simplification to retain history of original part. You can also choose to generate Primitive History.

Simplify result with separate bodies option

Use of Coordinate System

By default, the current working CS is used as the Simplify operation CS, but you can also change operation CS through the **Simplify** command property window or Property dialog.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Heal

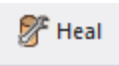
The **Heal** command provides a way to correct geometric violations and to remove specific kinds of small features. When models are imported, two types of errors can occur - geometry errors and topology errors. Geometry errors are errors in definition of the underlying geometry while topology errors are errors in how the underlying components like faces, edges and vertices are connected. Ansys Electromagnetics recommends that these be fixed before you invoke mesh generation.

Imported objects which have only one operation on the history tree, can be healed. (Use the Purge History command to remove unwanted history operations before using **Heal**.)

> **Note:**
>
> If you need to save the object history, save a separate copy for that purpose before you heal the object.

Healing can be invoked in different ways.

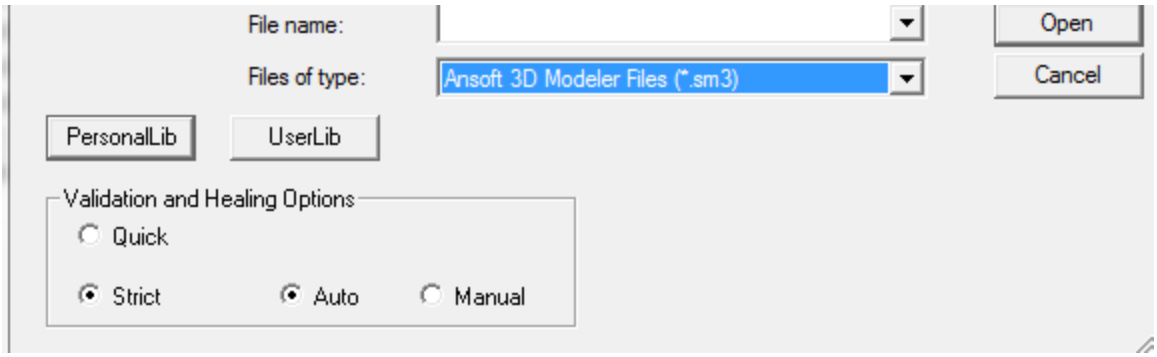
- The menu command **Modeler** > **Model Preparation** > **Heal** command, or the **Model** ribbon **Heal** icon applies to a selected object.

- Some formats permit healing during **Modeler** > **Import**. These are
  - 3D Modeler file (*.sm3),
  - SAT file (*.sat),
  - STEP file (*.step,*. stp),
  - IGES file (*.iges, *.igs),
  - CreoParametric files (*.prt, *.asm),
  - CATIA (*.model, *.CATpart), and
  - Parasolid file (*.x_t, and *.x_b).

  Selecting these formats enables a check box at the bottom of this window, **Validation and Healing Options**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



See this table for details. The Quick option allows you to switch off healing to speed up the import process.

- The **Model Analysis** dialog box that appears after running **Modeler** > **Model Analysis** > **Analyze Objects**, or **Modeler** > **Model Analysis** > **Show Analysis** dialog includes a **Perform** action menu with **Heal Objects** as a selection.

Any of these approaches leads to the same heal process.

Basic Steps in the Heal Process

There are several steps that are performed on selected objects.

1. Entity check, according to the **Analysis Options** settings.
2. Basic healing. This is done for all selected objects. Basic healing consists of fixing surface normals in the object and updating the orientation (to avoid having an object with negative volume).
3. Advanced healing. This is auto-heal. This is invoked on objects that require healing, that is, bodies that have errors, including have non-manifold errors.
4. Feature Removal. If you choose in the **Healing Options** to remove small holes, chamfers, blends, small edges, small faces and/or sliver faces, the actions are performed on all selected objects. There is no guarantee that small feature removal will be successful. (Also see Specifying the Model Resolution for defeaturing through the Auto Simplify and Model Resolution settings there.)

The above actions are performed on the selected objects. If you choose objects for healing which have not been analyzed, analysis is performed to determine its state (that is, whether it has invalid entities, small entities, and so forth). Invalid objects have all the above steps performed. Advanced healing is not performed on objects that do not require it.

While working on analyzing complex bodies, it is sometimes useful to examine faces, edges and vertices. In particular it is useful to find the connected faces for a face or edge or vertex, connected edges for a face/edge/vertex and connected vertices for a face/edge/vertex. The additional selection modes are available under **Edit** > **Selection Mode**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Validation and Healing Options for Import File

The **Import File** dialog box contains shows different Validation and Healing options for the seven formats listed here. The Quick option allows you to import these formats without healing.

Format	Options Available			
3D Modeler file (*.sm3),	Quick	Strict		
SAT file (*.sat),	Quick	Strict	Auto or manual	
STEP file (*.step,*.stp),	Quick	Strict	Auto or Manual	Stitch Tolerance
IGES file (*.iges, *.igs)	Quick	Strict	Auto or Manual	Stitch Tolerance
Creo Parametric files (*.prt, *.asm)	Quick	Strict	Auto or Manual	Import Free Surfaces. This imports such surfaces as well as parts.
CATIA (*.model, *.CATpart)	Quick	Strict	Auto or Manual	Stitch Tolerance
Parasolid file (*.x_t, and *.x_b)	Quick	Strict	Auto or Manual	

Healing Stage One: Validation Check

After import an object, you should perform a validation check. This lets you focus on objects and object pairs that prevent the mesh from being invoked. The objects that fail api_check_entity() should be analyzed via the **Analyze Objects** menu item.

1. Select the objects, and click **Modeler** > **Model Analysis** > **Analyze Objects**.
   The **Analysis Options** dialog box appears, allowing you to perform small feature detections. Once you have completed this, click **OK**. The **Model Analysis** dialog box appears.
2. In the **Model Analysis** dialog box, select the objects marked with "Invalid Entities Found", and click **Perform** > **Heal Objects**. The **Healing Options** dialog box appears, allowing you to remove parameters. Click **OK**. The **Model Analysis** dialog box reappears.

   > **Note:** In most cases, the objects are healed, and the errors are fixed.

3. If errors persist, select the edges and faces still containing errors, and click **Delete**.
   This replaces each selected face/edge object by a tolerant edge/vertex, respectively. In some cases, the replacement of the face/edge by tolerant edge/vertex fails.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When models pass the initial validity checks, mesh generation could still fail. The following errors can be present in models: (See Detecting Errors.)

1. Non-manifold topology. These are non-manifold edges and vertices that are present in the model.
2. Object pair intersection. This detects whether pairs of objects intersect.
3. Small feature detection – small edge length, small face area and sliver face detection.
4. Misaligned entities detection – detects pairs of faces from objects that can be aligned to remove object intersections. This improves the probability of mesh success.
5. Mesh failure error display. This is available for single object, object pairs and last simulation run (all objects in a model). Errors reported by the meshing module are reported to the user.

Errors of type 3 and 4 must be resolved before the mesh can be invoked on the model.

By default, the **Heal** command is automatically applied to imported objects.

Healing Stage Two: Fix Intersections Between Objects

The second stage in healing is to fix intersections between objects. After validation check is performed, the pairs of objects that intersect are chosen for analysis. Use the analysis results to find whether objects have faces that can be aligned.

Choose all the bodies that intersect with another body.

1. From the **Objects** tab of the **Model Analysis** dialog box, click **Perform** > **Analyze Interobject Misalignment**.

   Alternatively, you can click **Modeler** > **Model Analysis** > **Analyze Interobject Misalignment**.

   If the analysis finds object pairs that can be aligned, they are displayed on the **Objects Misalignment** tab.

2. You can select individual or multiple rows, and align the faces. In some cases, face alignment fails if the topology of the body changes by a large factor after alignment.
3. Identify individual bodies and body pairs that fail to mesh.
4. Perform a mesh analysis on individual objects and object pairs.
5. Review the reports and fix the errors.

Healing Stage Three: Fix Object Pair Intersections

The next stage of healing is to fix object-pair intersections. The healing process in stage two might fix some of these errors (by alignment).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

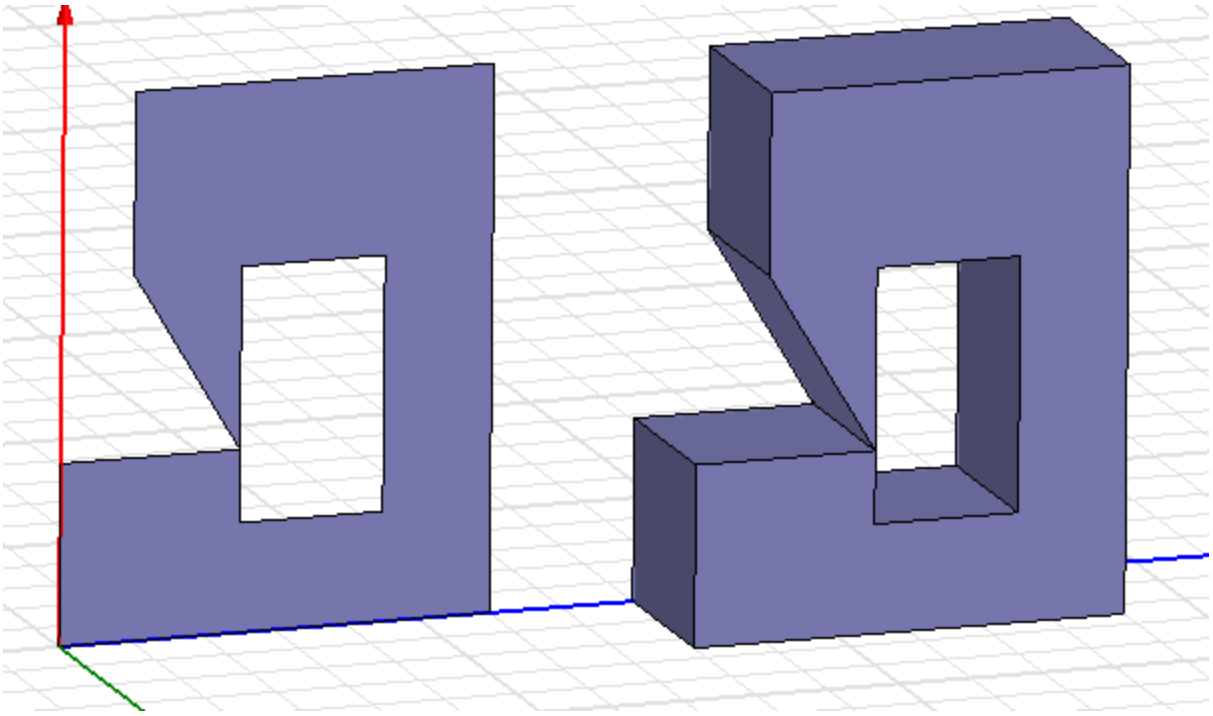
1. If these errors are still present, you must remove them either by using the **Remove Faces** command (**Modeler** > **Model Analysis** > **Remove Faces**) or by performing a Boolean subtraction.
2. Overlap between objects is too large to be fixed by healing or by face alignment. Boolean intersect shows the common portion between the bodies. In this case, use a subtract operation to remove overlaps.

Healing Stage Four: Analysis Phase

If the last simulation run contains errors, they are displayed in the **Model Analysis** dialog box, on the **Mesh Feedback** tab.

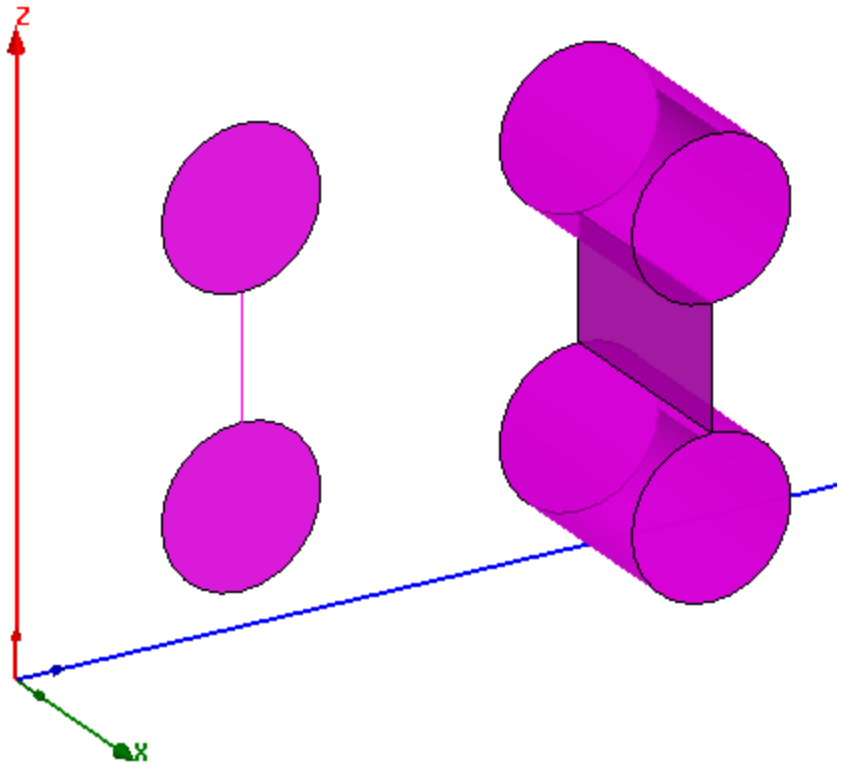
Healing Non-Manifold Objects

Non-manifold objects, in simple terms, are non-physical objects or objects that cannot be manufactured. For example, objects that intersect themselves (like the symbol for infinity in 2D) are clearly non-manifold. In addition objects that touch themselves may be non-manifold such as when a 2D object touches itself at a vertex, or a 3D object touches itself at a point or edge. These cases are shown in the following figure.



Another type of non-manifold object has mixed dimensionality. For example, a pair of 2D objects connected by a 1D line segment, or a pair of 3D objects connected by a 2D sheet object. These cases are illustrated below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The criteria for manufacturability is a simple manifestation of a complex mathematical concept that must be adhered to in the solid modeling system. When creating geometry, either directly, or through boolean operations, you should always consider whether or not the resulting operation will result in an object that could not be manufactured. If this is the case, then the object will cause an error in the modeler or in the meshing system.

To heal non-manifold objects:

1. Identify an edge that is non-manifold.
2. Select the connected faces.

   You can use the face selection toolbar icons.

3. Create a face coordinate system on the planar face.
4. Create a small box to cover the non-manifold edge.
5. Either do a union or a subtraction to remove the faces that contain the non-manifold edge.

   The non-manifold edge is now removed. You may also remove or add a small portion of the model.

6. Repeat for all non-manifold edges.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Healing Options

The **Healing Options** let you control how healing proceeds with respect to a variety of features and issues.

1. Click **Modeler** > **Model Healing** > **Heal** to open the **Healing Options** dialog box. You can also open the **Healing Options** dialog box from the **Model Analysis** dialog via the **Objects** tab drop down menu.

   The **Healing Options** dialog contains three tabs:

   - **Healing Options**
   - **Feature Removal Options**
   - **Properties**, which lists the geometric properties of the currently selected object.
2. Select the **Healing Options** tab to specify the following:
   - **Heal Type** as: **Auto Heal** (default), **Manual Heal**, or **No Heal**. Selecting **Manual Heal** enables the **Manual Heal Options**:
     - **Perform Tolerant Stitching** check box: This enables a field for the **Stitch Tolerance** value, and a check box to **Stop After First Error**.
     - **Perform Geometry Simplification**: This enables fields for **Simplification Tolerance** and **Maximum Generated Radius** values. You can also select radio buttons to **Simplify Curves**, **Surfaces**, or **Both**.
     - **Tighten Gaps** settings: A check box to select **Perform Tighten Gaps**. A field to specify **Tighten Gaps Within** a given value in mm.
3. Select the **Feature Removal Options** tab to specify the following:

   Feature Removal Options:

   - **Remove Holes** check box and **Maximum Radius** value.
   - **Remove Chamfers** check box and **Maximum Width** value.
   - **Remove Blends** check box and **Maximum Radius** value.

   Remove Small Entity Options:

   - **Small Edges, Length Less Than**, less than a specified value.
   - **Small Faces Area Less Than**, less than a specified area.
   - **Sliver Face Width Less Than**, less than either:
   - **Object Bounding box Scale Factor**, less than a specified scale factor

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Sliver Edge Width**, less than a specified value.

  > **Note:** Sliver faces have a maximum distance among the long edges that is smaller than the specified tolerance and have at least one short edge and at most three long edges. A short edge has a length less than the specified tolerance. A long edge has a length greater than the specified tolerance. You can give the tolerance as a absolute value or a factor of the bounding box containing the face.

Control Object Properties Change options:

- **Allowable Change in Surface Area** check box, and percent value.
- **Allowable Change in Volume** check box, and percent value.

4. Select the **Properties** tab to view the geometric properties of the currently selected object.

5. Click **OK** to apply the specified Healing options and to open the Analysis dialog box.

Stitch Sheets

Use the **Modeler** > **Model Preparation** > **Stitch Sheets** command to stitch selected sheets.

1. Select two or more sheet objects.

   This enables the **Stitch Sheets** command on the **Modeler** > **Model Preparation** submenu.

2. Click **Modeler** > **Model Preparation** > **Stitch Sheets**; or, on the **Model** ribbon, click the **Stitch Sheets** icon.

   This displays a **Stitch** dialog box with a **Maximum Stitch Tolerance** field. The default value (**Auto**) comes from the **Healing** dialog box's **Options** tab with Manual Healing selected. You may edit the value in the **Stitch** dialog box or in the **Healing Options**.

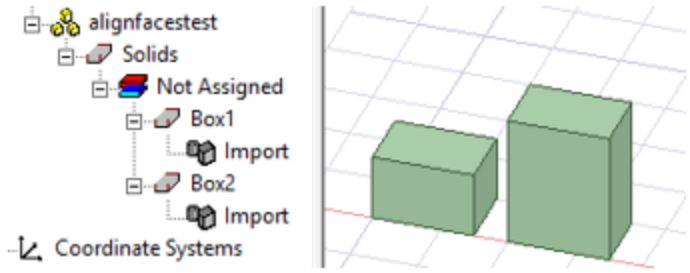
3. Click **OK**.

   This closes the dialog box and attempts to perform stitching on the selected sheets. If the sheets are separated beyond the stitch tolerance, stitching is not performed and a warning is issued.

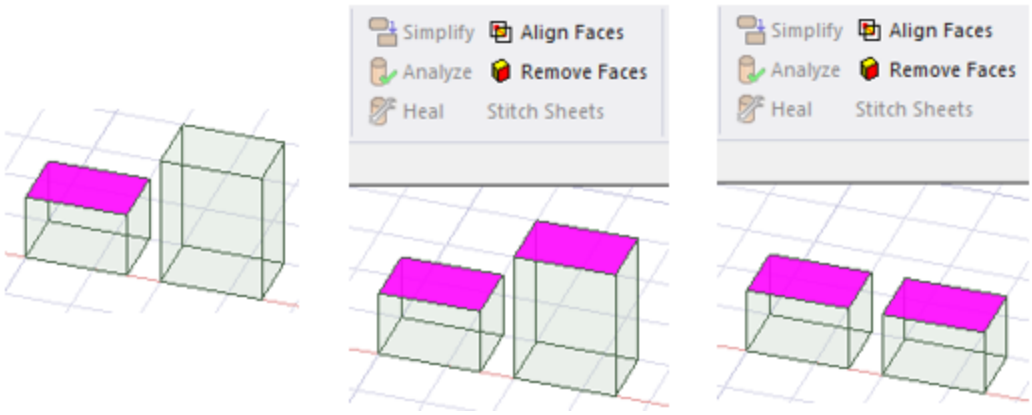
Align Selected Faces

Use the **Modeler** > **Model Preparation** > **Align Faces** command to align the adjacent selected faces of imported objects that have only one operation in their History Tree. You can use **Undo/Redo** on this command.

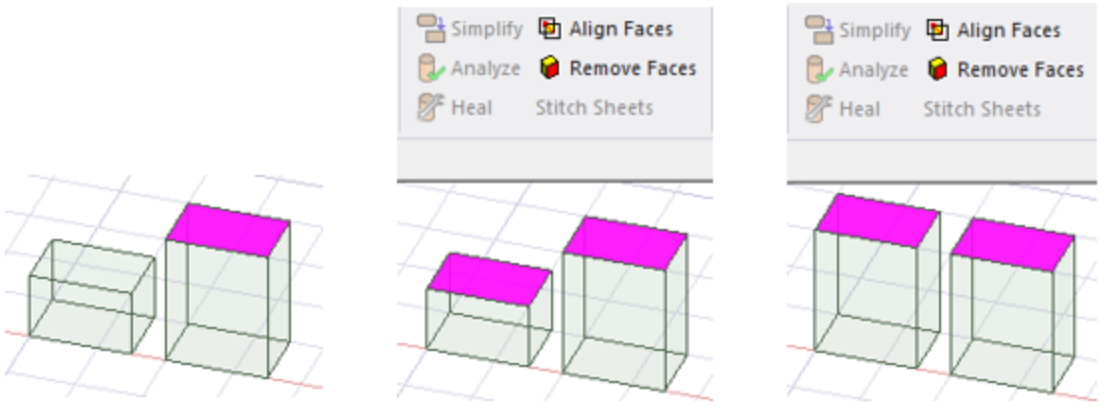
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Model** ribbon also includes an **Align Faces** icon, which is enabled when you make appropriate face selection. The first selected face defines the plane of alignment for subsequent selection, that is, tool faces to blank faces.

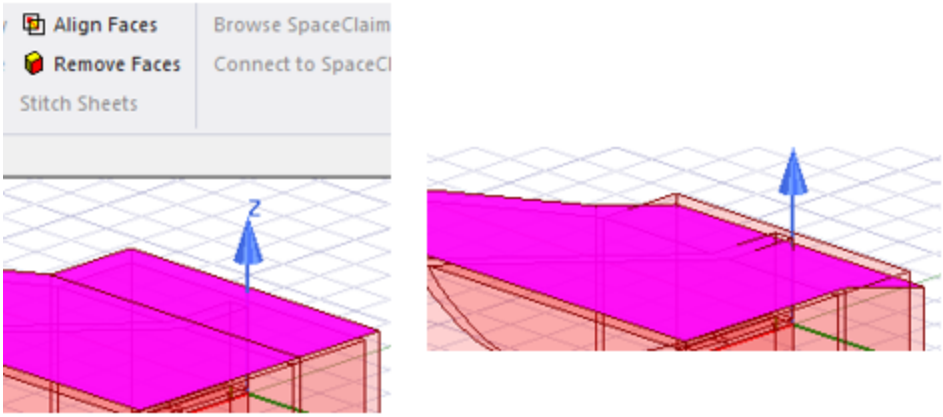


Selecting faces in a different order changes the alignment.



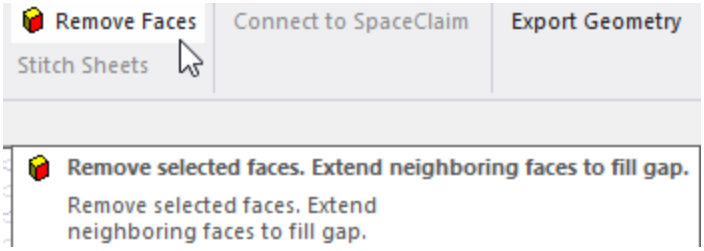
The selected faces can touch.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Remove Selected Faces

Use the **Modeler** > **Model Preparation** > **Remove Faces** command to remove the selected faces, extending the remaining faces to fill gaps. You can also use the **Model** ribbon **Remove Faces** icon when you have made an appropriate face selection.



If you find object-pair intersections that healing does not fix, or that can be fixed (by alignment), you can correct the problem by one of the following methods.

1. Use the **Remove Faces** command (**Modeler** > **Model Preparation** > **Remove Faces**) or by performing Boolean subtract.
2. If overlap between objects is too large to be fixed by healing or by face alignment. Boolean intersect shows the common portion between the bodies. In this case, use a subtract operation to remove overlaps.

Remove Selected Edges

Use **Modeler** > **Model Preparation** > **Remove Edges** to remove selected edges. You can also use the toolbar icon when you have made an appropriate edge selection.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

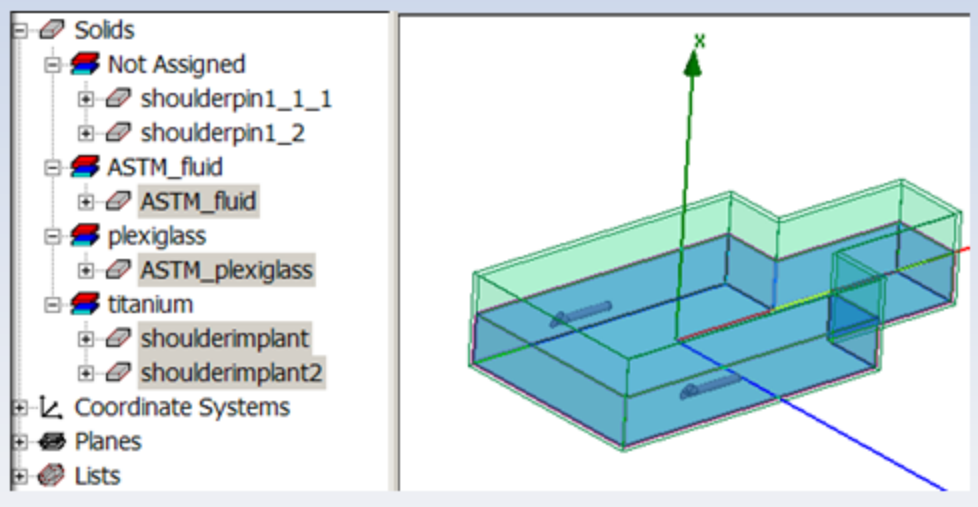
Perform Explicit Subtractions

Use **Modeler** > **Model Preparation** > **Perform Explicit Subtractions** to subtract selected History Tree objects that are fully contained by other parts.

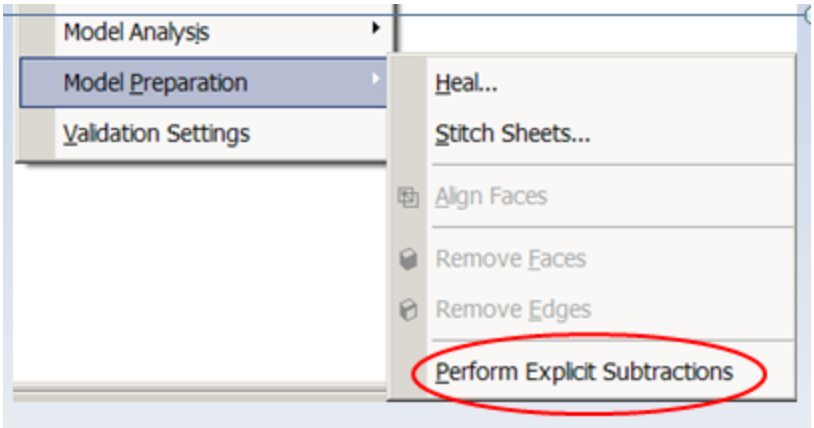
1. Select two or more history objects.

   This enables the **Modeler** > **Model Preparation** > **Perform Explicit Subtractions** submenu.

   For example, consider the following model.

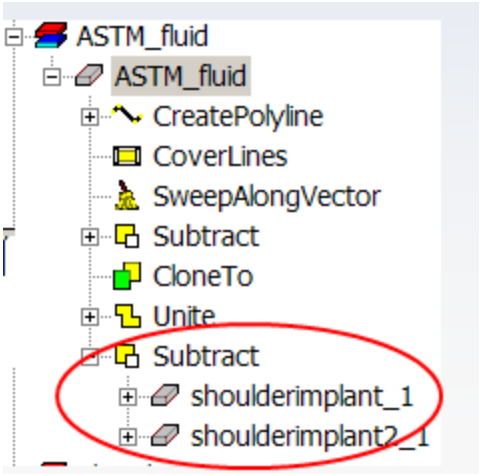


2. **Perform Explicit Subtraction** can be performed.

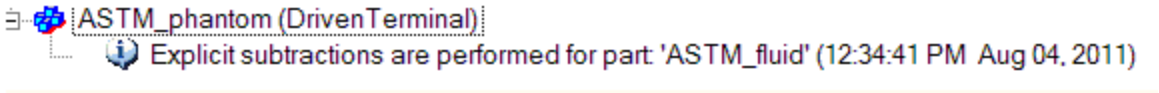


   The results appear in the History tree as shown:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Message window reports this successful action.



If no parts are fully contained by other parts, a message explains that no explicit subtraction is needed.

User Customization through User-Defined Primitives (UDPs)

User-Defined Primitives (UDPs) allow users to add customized geometric modeling commands to the Ansys Electronics Desktop. UDPs are C++ compiled or Python libraries that can be added to the desktop interface and shared between users with common modeling needs.

To create a UDP, see Creating a User-Defined Primitive for requirements and the procedure for building a proper DLL or Python library.

In order to share UDPs between users, an existing DLL or Python library may be copied into the **UserLib** > **User Defined Primitives** subdirectory which can be given the paths specified in **Tools** > **General Options** under *General* > *Directories*. Placing an appropriately constructed DLL or Python library in this subdirectory and executing **Draw** > **User Defined Primitives** > **Update Menu** adds a new menu item in the **Draw** > **User Defined Primitives** menu to allow access to the UDP.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating a User-Defined Primitive

The modeler allows you to generate user-defined primitives, primitives customized to suit any application. User-defined primitives are accessed using DLLs or IronPython scripts that you build and compile. When user-defined primitives exist in your UserLib or PersonalLib directory (given the paths specified in **Tools** > **General Options**, Project Options tab), they appear in the menu bar under **Draw** > **User Defined Primitives**. Newly created UDPs will appear after a restart or after executing the command **Draw** > **User Defined Primitive** > **Update Menu.**

For Python-based primitives, see the **Maxwell Scripting Guide**.

The modeler includes example C++ source and header files that can be used to generate DLLs. The files are located in the **UserDefinedPrimitives/Examples** subdirectory under the directory.

As an example, create the primitive **myUDP.dll** using Microsoft Visual C++ Developer Studio:

1. Create a directory to store all of the workspace information, call it UDPDir.
2. Use the sample workspace **RectangularSpiral.dsw** as a template:
    a. Copy **RectangularSpiral.dsw** and **RectangularSpiral.dsp** from the **UserDefinedPrimitives/Examples** directory to this new directory.
    b. Make sure the new files have write permissions.
    c. Rename the files to **myUDP.dsw** and **myUDP.dsp** respectively.
    d. Open the **.dsw** and **.dsp** files in a text editor, and replace every occurrence of **RectangularSpiral** with **myDLL**.
    e. Save **myUDP.dsp** and **myUDP.dsw**.
3. In the **UDPDir** directory, create a **Headers** subdirectory.
4. Copy the **UserDefinedPrimitiveStructures.h** and **UserDefinedPrimitiveDLLInclude.h** files from the **UserDefinedPrimitives/Headers** directory.

> **Note:**
>
> The header files include information on the methods that are available for use in your source code. They must be included when you compile the DLL.

5. In the **UDPDir** directory, create a **Source** subdirectory.
6. Use the sample source file **RectangularSpiral.cpp** as a template:
    a. Copy **RectangularSpiral.cpp** from the **UserDefinedPrimitives/Examples** directory to this new directory.
    b. Make sure the new file has write permission.
    c. Rename the file to **myUDP.cpp**.

The resulting directory structure will resemble the following:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
UDPDir/
        myUDP.dsw
        myUDP.dsp
        Headers/
          UserDefinedPrimitiveDLLInclude.h
          UserDefinedPrimitiveStructures.h
        Sources/
          myUDP.cpp
```

7. Open **myUDP.dsw** using Microsoft Visual C++ Developer Studio, and edit the source code to create your desired primitive. You may also add additional headers and source files as appropriate.

   The UDP dll contains a data structure called UDPPrimitiveTypeInfo. This contains information about the udp, its purpose, company/author who created it, date created and the version number. When you select a primitive from your library, you see the **Create Primitive** dialog with a **Parameters** tab for setting the parameters, and an **Info** tab with the information from this data structure.

8. Build **myUDP.dll** using the **Win32 Release** configuration.
9. Copy the resulting file **myUDP.dll** to the **[*ProductInstallationPath*]/userlib/UserDefinedPrimitives** directory or the **[*PersonalLibPath*]personallib/UserDefinedPrimitives** directory.
10. To view your primitives, click **Draw** > **User Defined Primitive** > **Update Menu** and then click **Draw** > **User Defined Primitive** > **[*PrimitiveToDraw*]**.

> **Note:**
>
> On Linux, you may use the same example directory structure, source, and header files to build and compile a shared library using C++. The resulting shared library will have a **.so** extension and needs to be placed in the same */userlib/UserDefinedPrimitives* directory.
>
> As with the Windows DLL, the compiled library will work only on the operating system on which it was built.

Specifying UDP Parameters

The **User Defined Primitive Operation** dialog box allows you to set the specific parameters required for a UDP.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the **Parameters** tab, a list of the parameters associated with the UDP is displayed along with the value, unit, evaluated value, and description for each parameter. To change a parameter:

1. Select the **Value** cell in the row corresponding to the parameter to be changed. The cell becomes editable. Enter the desired **Value** for the parameter.
2. Click on the cell under **Unit** to obtain a list of the available units for the parameter. Select the desired **Unit**.
3. When the desired changes have been made, click **OK** to create the geometry defined by the UDP and the parameters you have entered.

The **Info** tab provides information about the UDP.

Updating the User-Defined Primitives Menu

When new user-defined primitives have been created or provided for your use, they should be placed in the *Maxwell/userlib/UserDefinedPrimitives* directory. Optionally, user-defined primitives can be placed in your **PersonalLib**. The **PersonalLib** directory path is specified on the **Directories** panel under **General** in the **Options** window. The **Update Menu** command scans the disk and creates menu items for new UDPs. To create menu items for new UDPs:

1. Place new UDPs in either of the directories listed above. Organizing UDPs using subdirectories is permitted.
2. Select **Draw** > **User Defined Primitive** > **Update Menu**.

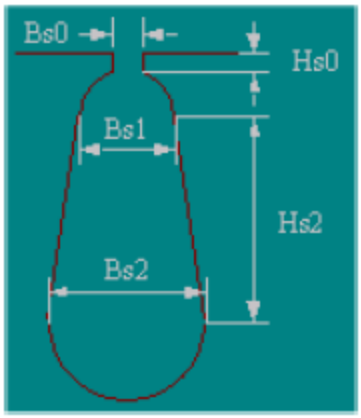
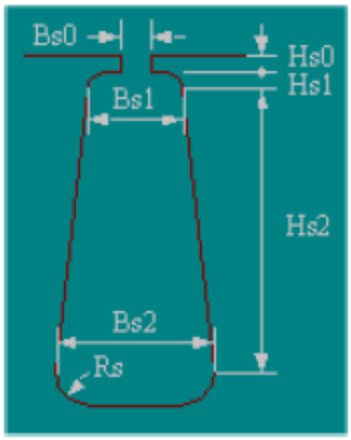
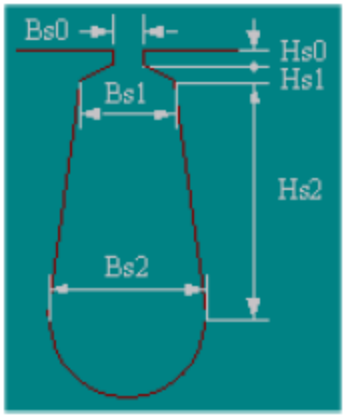
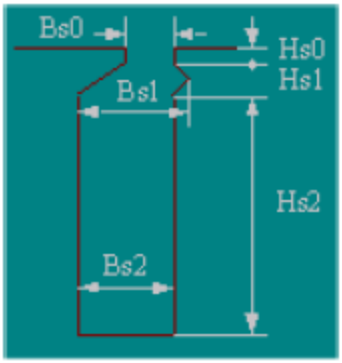
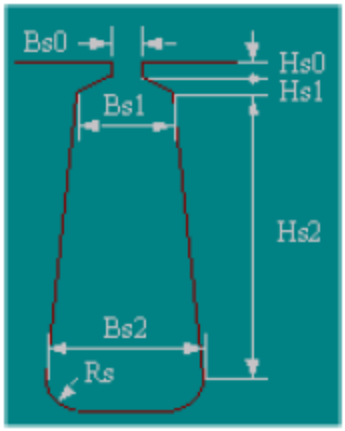
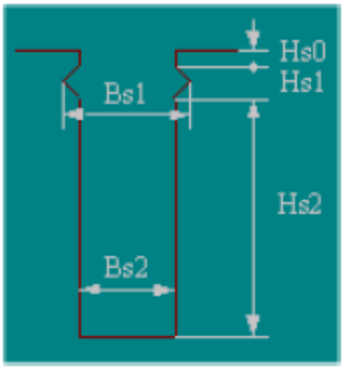
The new UDPs may now be run directly from the **Maxwell** menu.

User-Defined Primitives (UDPs) for Motor Geometries

When designing a motor or other machine in either the Maxwell 2D or 3D application or the RMxprt application, multiple stator, rotor, and pole geometries are available.

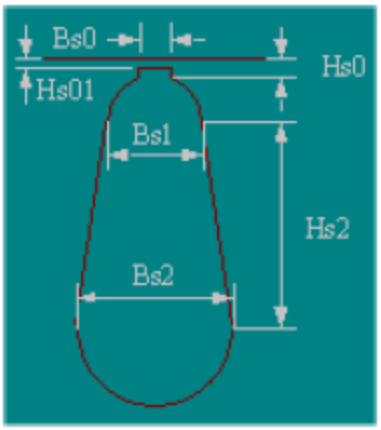
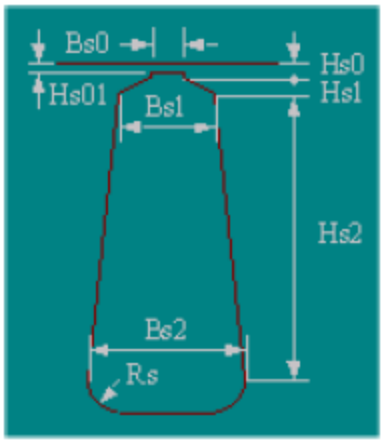
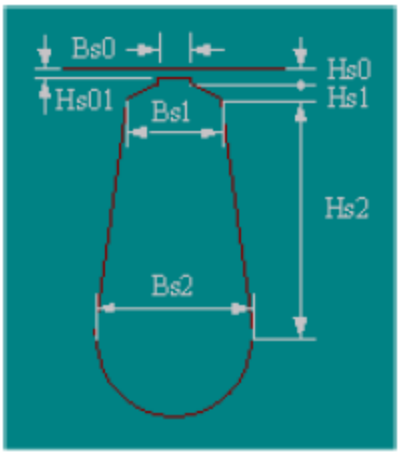
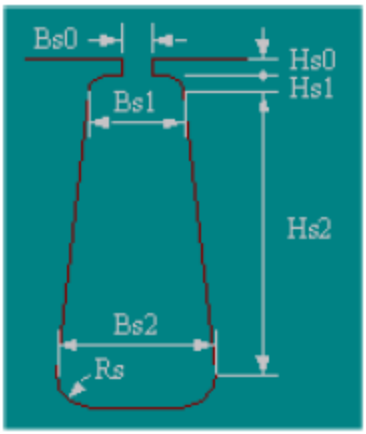
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stator Geometries

Slot Type	Slot Dimensions	Slot Type	Slot Dimensions
1		4	
2		5	
3		6	

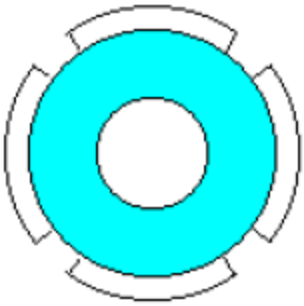
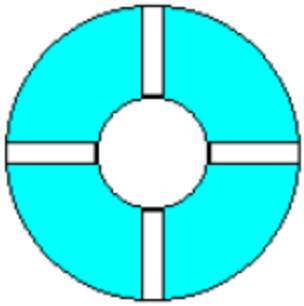
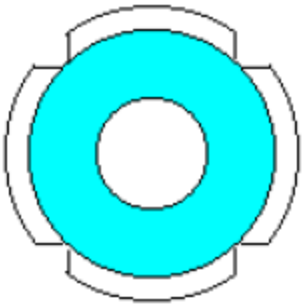
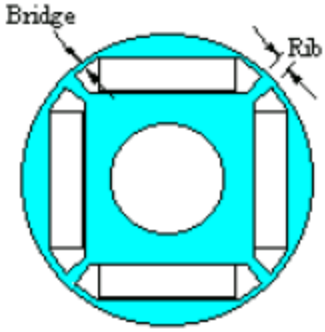
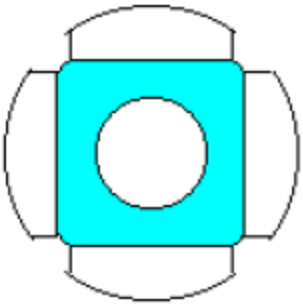
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rotor Geometries

Slot Type	Slot Dimensions	Slot Type	Slot Dimensions
1		3	
2		4	

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pole Geometries

Slot Type	Slot Dimensions	Slot Type	Slot Dimensions
1		4	
2		5	
3			

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameters for RMxprt User-Defined Primitives

In the **Draw/User Defined Primitive/syslib/RMxprt** menu, the following user-defined primitives exist for creating elements of motors.

Click the primitive of interest to see the parameters required.

Band	**LapCoil**	**SRMCore**
ClawPoleCore	**LinearMCore**	**SynRMCore**
ConCoil	**NonSalientPoleCore**	**TransCoil**
DCMCore	**PMCore**	**TransCore**
DiskCoil	**PMDamperCore**	**UnivMCore**
DiskPMCore	**RacetrackSlotCore**	**VentSlotCore**
DiskSlotCore	**SalientPoleCore**	**WaveCoil**
DoubleCage	**SlotCore**	
IPMCore	**SquirrelCage**	

RMxprt/Band

Property	Description
DiaGap	Band diameter in gap center, DiaGap < DiaYoke for outer band
DiaYoke	Band diameter on yoke side, DiaYoke < DiaGap for inner band
Length	Band Length
SegAngle	Angle per segment of band (0.1 ~ 5 degrees)
Fractions	Number of circumferential fractions, 1 for circular region
InfoCore	0: band; 100: outer region or shaft

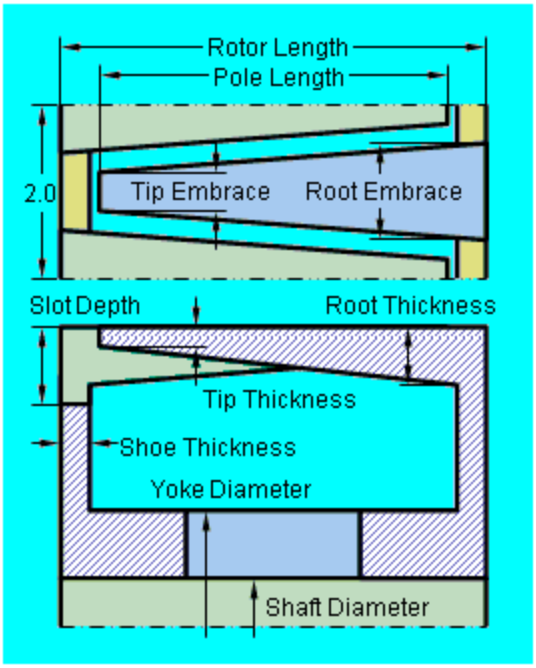
RMxprt/ClawPoleCore

See graphic below for definitions.

Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaShaft for outer cores
DiaShaft	Core diameter on shaft side, DiaShaft < DiaGap for inner cores

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Length	Core Length
Poles	Number of poles
EmbraceTip	Embrace of pole tip
EmbraceRoot	Embrace of pole root
ThickTip	Thickness of pole tip
ThickRoot	Thickness of pole root
ThickShoe	Thickness of side shoes
DepthSlot	Depth of slot between two poles
ThickYoke	Thickness of yoke
LengthPole	Length of pole from tip to tip
LengthMag	Length of magnet or the second air-gap
LenRegion	Region length
InfoCore	0: core; 1: core&coil; 2: coil; 3: magnet; 100: region



RMxprt/ConCoil

Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaYoke for outer cores

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DiaYoke	Core diameter on yoke side, DiaYoke < DiaGap for inner cores
Length	Core Length
Skew	Skew angle in core length range
Slots	Number of slots
SlotType	Slot Type: 1 to 7
Hs0	Slot opening height
Hs1	Slot wedge height
Hs2	Slot body height
Bs0	Slot opening width
Bs1	Slot wedge maximum width
Bs2	Slot body bottom width, 0 for parallel teeth
Rs	Slot body bottom fillet
FilletType	0: a quarter circle; 1: tangent connection
LayerLoc	0: whole; 1: middle; 2: top; 3: bottom
CoilPitch	Coil pitch measured in slots
EndExt	One-side end extended length
LayerExt	Span layer extension in the axial direction
LayerDiff	Span layer difference in the radial direction
AltEnd	0: same end layers; 1: alternate end layers
LenRegion	Region length
InfoCoil	0: coil; 1: terminal1; 2: terminal2; 100: region

RMxprt/DCMCore

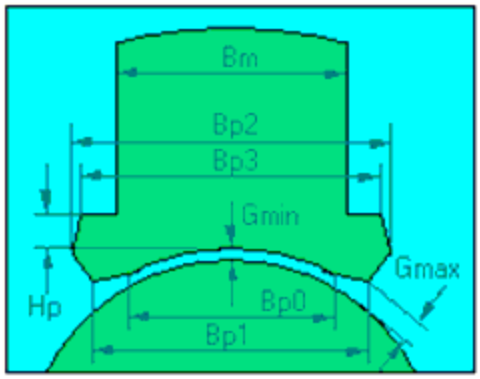
See graphics below for definitions.

Property	Description
DiaGap	Core diameter on gap side, or Dmin, the minimum inner diameter
DiaYoke	Core diameter on yoke side, or frame outer diameter
Length	Main pole length, 0 for 2D geometry
Skew	Skew angle in pole length range
FrameWidth	Overall width of a racetrack frame
FrameThick	Frame thickness
FrameLength	Frame length
Poles	Number of poles
PoleType	Pole Type: 1 or 2

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dmax	Diameter of shoe tip with maximum air gap length
Bp0	Pole arc width with uniform air gap, 0 for eccentric air gap
Bp1	Total pole width (the width between shoe tips)
Bp2/Rp0	Bp2 (max shoe width for pole type 1), or Rp0 (shoe fillet radius for pole type 2)
Bp3/Rp1	Bp3 (min shoe width for pole type 1), or Rp1 (pole fillet radius for pole type 2)
Hp	Pole body height
Bm	Pole body width
FieldWndgs	Number of field windings, 2 for both series and shunt windings
EndExt	Coil one-side end extended length
SlotsPerPole	Compensating slots per pole distributed under pole arc surface
Bc0	Opening width of compensation slots
Hc0	Opening height of compensating slots
Bc2	Width of compensating slots
Hc2	Height of compensating slots
CmpEndExt	One-side end extended length of compensating coils
ComPoleWidth	Width of the commutating poles
ComPoleHeight	Height of the commutating poles
ComPoleLength	Length of the commutating poles
ComShoeWidth	Shoe width of the commutating poles
ComShoeHeight	Shoe height of the commutating poles
ComShoeLength	Shoe length of the commutating poles
ComGap2	Air gap length between commutating poles and the frame
CmtEndExt	One-side end extended length of commutating coils
LenRegion	Region length
InfoCore	0: core&coil; 1: poles; 2: frame; 3: com poles; 4: shunt coil; 5: series coil; 6: com coil; 100: region
InfoTerm	0: whole coil; 1: terminal1; 2: terminal2

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



RMxprt/DiskCoil

Property	Description
DiaOuter	Core outer diameter
DiaInner	Core inner diameter
Thickness	Core thickness
Gap	Gap between the core and xy plane, <0 for lower core
Skew	Skew angle
Slots	Number of slots
SlotType	Slot type: 1 to 7
Hs0	Slot opening height
Hs1	Slot wedge height
Hs2	Slot body height
Bs0	Slot opening width
Bs1	Slot wedge maximum width
Bs2	Slot body bottom width. 0 for parallel teeth
Rs	Slot body bottom fillet
FilletType	0: a quarter circle; 1: tangent connection
Layers	Number of winding layers
CoilPitch	Coil pitch measured in slots
EndExt	One-side end extended length
EndClr	Clearance between two end spans; 0 for no span
InfoCoil	0: all coils; 1: one coil only
SegAngle	Angle per segment of end span: 5~15 degrees

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RMxprt/DiskPMCore

Property	Description
DiaOuter	Core outer diameter
DiaInner	Core inner diameter
Thickness	Core thickness
Gap	Gap between the core and xy plane, < 0 for lower core
Skew	Skew angle
Poles	Number of poles
Embrace	Pole embrace
ThickMag	Magnet thickness
InfoCore	0: core only; 1: all PMs; 2: one PM only

RMxprt/DiskSlotCore

Property	Description
DiaOuter	Core outer diameter
DiaInner	Core inner diameter
Thickness	Core thickness
Gap	Gap between the core and xy plane, < 0 for lower core
Skew	Skew angle
Slots	Number of slots
SlotType	Slot type: 1 to 6
Hs0	Slot opening height
Hs01	Slot closed bridge height
Hs1	Slot wedge height
Hs2	Slot body height
Bs0	Slot opening width
Bs1	Slot wedge maximum width
Bs2	Slot body bottom width. 0 for parallel teeth
Rs	Slot body bottom fillet
FilletType	0: a quarter circle; 1: tangent connection
HalfSlot	0: symmetric slots; 1: half slots
RingLength	One-side radial ring length
RingHeight	Axial ring height

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

InfoCore	0: core only; 1: squirrel cage only

RMxprt/DoubleCage

Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaYoke for outer cores
DiaYoke	Core diameter on yoke side, DiaYoke < DiaGap for inner cores
Length	Core Length
Skew	Skew angle in core length range
Slots	Number of slots
SlotType	Slot Type: 1 to 4
Hs0	Slot opening height
Hs01	Slot closed bridge height
Hs1	Slot wedge height
Hs2	Slot body height
Bs0	Slot opening width
Bs1	Slot wedge maximum width
Bs2	Slot body bottom width, 0 for parallel teeth
Rs	Slot body bottom fillet
FilletType	0: a quarter circle; 1: tangent connection; 2 & 3: arc bottom
HalfSlot	0: symmetric slots; 1: half slots
BarEndExt	One-side bar end extended length
RingLength	One-side radial ring length
RingHeight	Radial ring height
RingDiaGap	Ring diameter on gap side
DoubleCage	0: normal squirrel cage; 1: double squirrel cage
BSlotType	Bottom slot type: 1 to 4
BHs0	Slot opening height
BHs1	Slot wedge height
BHs2	Slot body height
BBs0	Slot opening width
BBs1	Slot wedge maximum width
BBs2	Slot body bottom width, 0 for parallel teeth
BRs	Slot body bottom fillet
CastRotor	0: insert-bar; 1: cast-rotor
LenRegion	Region length

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

InfoCoil	0: bar & rings; 1: bars; 2: rings; 100: region

RMxprt/IPMCore

Property	Description
DiaGap	Core diameter on gap side, or outer diameter
DiaYoke	Core diameter on yoke side, or inner diameter
Length	Core Length
Poles	Number of poles
PoleType	Pole Type: 1 or 3
D1	Limited diameter of PM ducts
O1	Bottom width for separate or flat-bottom duct
O2	Distance from duct bottom to shaft surface
B1	Duct thickness
Rib	Rib width
HRib	Rib height
DminMag	Minimum distance between side magnets
ThickMag	Magnet thickness
WidthMag	Total width of all magnet per pole
LenRegion	Region length
InfoCoil	0: core; 1: magnets; 2: ducts; 100: region

RMxprt/LapCoil

Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaYoke for outer cores
DiaYoke	Core diameter on yoke side, DiaYoke < DiaGap for inner cores
Length	Core Length
Skew	Skew angle in core length range
Slots	Number of slots
SlotType	Slot Type: 1 to 7
Hs0	Slot opening height
Hs1	Slot wedge height
Hs2	Slot body height

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Bs0	Slot opening width
Bs1	Slot wedge maximum width
Bs2	Slot body bottom width, 0 for parallel teeth
Rs	Slot body bottom fillet
FilletType	0: a quarter circle; 1: tangent connection
Layers	Number of winding layers
CoilPitch	Coil pitch measured in slots
EndExt	One-side end extended length
SpanExt	Axial length of end span; 0 for no span
SegAngle	Angle per segment of end span: <5 for true-surface end span
LenRegion	Region length
InfoCoil	0: winding; 1: coil; 2: terminal1; 3:terminal2; 100:region

RMxprt/LinearMCore

Property	Description
WidthCore	Core width in motion direction
ThickCore	Core thickness
Length	Core length
SlotPitch	The distance between two slots
Slots	Number of slots
SlotType	Slot Type: 1 to 7
Hs0	Slot opening height
Hs1	Slot wedge height
Hs2	Slot body height
Bs0	Slot opening width
Bs1	Slot wedge maximum width
Bs2	Slot body bottom width, 0 for parallel teeth
Rs	Slot body bottom fillet
Layers	Number of winding layers
CoilPitch	Coil pitch measured in slots
EndExt	One-side end extended length
SpanExt	Axial length of end span; 0 for no span
InfoCore	0: core only; 1: core & all coils; 2: one coil only

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RMxprt/NonSalientPoleCore

Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaYoke for outer cores
DiaYoke	Core diameter on yoke side, DiaYoke < DiaGap for inner cores
Length	Core Length
Skew	Skew angle in core length range
IndexingSlots	Number of indexing slots for slot pitch calculation
ActualSlots	Number of actual slots: ActualSlots <= IndexingSlots
SlotType	Slot Type: 1 to 7
Hs0	Slot opening height
Hs1	Slot wedge height
Hs2	Slot body height
Bs0	Slot opening width
Bs1	Slot wedge maximum width
Bs2	Slot body bottom width, 0 for parallel teeth
Rs	Slot body bottom fillet
FilletType	0: a quarter circle; 1: tangent connection; 2 & 3: arc bottom
Poles	Number of poles
TangentDucts	Number of surface tangential vent ducts
TDuctWidth	Width of surface tangential vent ducts
TDuctDepth	Depth of surface tangential vent ducts
TDuctPitch	Pitch of surface tangential vent ducts
AxialDucts	Number of axial vent ducts per pole
ADuctWidth	Width of axial vent ducts in main tooth
ADuctDepth	Depth of axial vent ducts in main tooth
LenRegion	Region length
InfoCore	0: core; 100: region

RMxprt/PMCore

Property	Description
DiaGap	Core diameter on gap side, or outer diameter
DiaYoke	Core diameter on yoke side, or inner diameter
Length	Core Length

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Poles	Number of poles
PoleType	Pole Type: 1 or 5
Embrace	Pole embrace (not for type 4)
ThickMag	Max thickness of magnets
WidthMag	Magnet width (for types 4 & 5)
Offset	Pole arc offset (for types 1, 2, & 3)
Bridge	Bridge thickness (for type 5 only)
Rib	Rib width (for type 5 only), Rib=0 for rectangular ducts
LenRegion	Region length
InfoCoil	0: core; 1: magnets; 2: magnet; 100: region

RMxprt/PMDamperCore

Property	Description
DiaGap	Core diameter on gap side, or outer diameter
DiaYoke	Core diameter on yoke side, or inner diameter
Length	Core Length
Skew	Skew angle in core length range
Slots	Number of damper slots per pole
SlotType	Slot type: 1 to 4
Hs0	Slot opening height
Hs01	Slot closed bridge height
Hs1	Slot wedge height
Hs2	Slot body height
Bs0	Slot opening width
Bs1	Slot wedge maximum width
Bs2	Slot body bottom width, 0 for parallel teeth
Rs	Slot body bottom fillet
FilletType	0: a quarter circle; 1: tangent connection; 2 & 3: arc bottom
Poles	Number of poles
PoleType	Pole Type: 1 or 8
D1	Limited diameter of PM ducts
O1	A locating dimension of PM ducts, for pole types 3, 4, 7 & 8
O2	A locating dimension of PM ducts, for pole types 1, 3, 5, 6, 7 & 8
B1	Barrier width, for pole types 1, 2 & 3
Rib	Rib to hold PM ducts, for pole types 1 to 7

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ThickMag	Magnet thickness
WidthMag	Total width of all magnet per pole
BarEndExt	One-side damper bar end extended length
RingLength	One-side axial ring length
RingHeight	Radial ring height
LenRegion	Region length
InfoCore	0: core; 1: magnets; 2: damper; 100: region

RMxprt/RacetrackSlotCore

Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaYoke for outer cores
DiaYoke	Core diameter on yoke side, DiaYoke < DiaGap for inner cores
Length	Core Length
Skew	Skew angle in core length range
Slots	Number of slots
SlotType	Slot type: 1 to 7
Hs0	Slot opening height
Hs01	Slot closed bridge height
Hs1	Slot wedge height
Hs2	Slot body height
Bs0	Slot opening width
Bs1	Slot wedge maximum width
Bs2	Slot body bottom width, 0 for parallel teeth
Rs	Slot body bottom fillet
FilletType	0: a quarter circle; 1: tangent connection; 2 & 3: arc bottom
Poles	Number of poles
WidYoke	Racetrack width for 2/4-pole cores
MidSlots	Number of middle-size slots for 2/4-pole cores
MidHs2	Middle-size slot body height for 2/4-pole cores
MidBs2	Middle-size slot body bottom width for 2/4-pole cores
SmlSlots	Number of small-size slots for 2/4-pole cores
SmlHs2	Small-size slot body height for 2/4-pole cores
SmlBs2	Small-size slot body bottom width for 2/4-pole cores
LenRegion	Region length

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

InfoCore	0: core; 100: region

RMxprt/SalientPoleCore

Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaYoke for outer cores
DiaYoke	Core diameter on yoke side, DiaYoke < DiaGap for inner cores
Length	Core Length
Skew	Skew angle in core length range
Slots	Number of damper slots per pole
SlotType	Slot type: 1 to 4
Hs0	Slot opening height
Hs01	Slot closed bridge height
Hs1	Slot wedge height
Hs2	Slot body height
Bs0	Slot opening width
Bs1	Slot wedge maximum width
Bs2	Slot body bottom width, 0 for parallel teeth
Rs	Slot body bottom fillet
FilletType	0: a quarter circle; 1: tangent connection; 2 & 3: arc bottom
SlotPitch	Slot pitch in mechanical degrees, refer to offset point
CenterPitch	Center slot pitch in mechanical degrees, refer to offset point
Poles	Number of poles
WidthShoe	Pole shoe width
HeightShoe	Pole shoe height (maximum)
WidthBody	Pole body width
HeightBody	Pole body height
AirGap2	Second air gap length
Offset	Pole arc offset
Off2_x	The second pole arc offset perpendicular to the pole-center line
Off2_y	The second pole arc offset parallel with the pole-center line
CoilEndExt	One-side coil end extended length
EndRingType	0: whole press board; 1: pole press board; 2: pole ring; 3: whole ring
BarEndExt	One-side damper bar end extended length, for types 2 & 3 only
RingLength	One-side axial ring length, or conductor press board thickness
RingHeight	Radial ring height, for types 2 & 3 only

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LenRegion	Region length
InfoCore	0: core; 1: core & coils; 2: coil; 3:damper; 4:terminal1; 5: terminal2;

RMxprt/SlotCore

Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaYoke for outer cores
DiaYoke	Core diameter on yoke side, DiaYoke < DiaGap for inner cores
Length	Core Length
Skew	Skew angle in core length range
Slots	Number of slots
SlotType	Slot type: 1 to 6
Hs0	Slot opening height
Hs01	Slot closed bridge height
Hs1	Slot wedge height
Hs2	Slot body height
Bs0	Slot opening width
Bs1	Slot wedge maximum width
Bs2	Slot body bottom width, 0 for parallel teeth
Rs	Slot body bottom fillet
FilletType	0: a quarter circle; 1: tangent connection; 2 & 3: arc bottom
HalfSlot	0 for symmetric slot, 1 for half slot
LenRegion	Region length
InfoCore	0: core; 100: region

RMxprt/SquirrelCage

Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaYoke for outer cores
DiaYoke	Core diameter on yoke side, DiaYoke < DiaGap for inner cores
Length	Core Length
Skew	Skew angle in core length range
Slots	Number of slots
SlotType	Slot type: 1 to 4

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Hs0	Slot opening height
Hs01	Slot closed bridge height
Hs1	Slot wedge height
Hs2	Slot body height
Bs0	Slot opening width
Bs1	Slot wedge maximum width
Bs2	Slot body bottom width, 0 for parallel teeth
Rs	Slot body bottom fillet
FilletType	0: a quarter circle; 1: tangent connection; 2 & 3: arc bottom
HalfSlot	0 for symmetric slot, 1 for half slot
BarEndExt	One-side bar end extended length
RingLength	One-side axial ring length
RingHeight	Radial ring height
RingDiaGap	Ring diameter on gap side
CastRotor	0:insert-bar; 1: cast-rotor
LenRegion	Region length
InfoCore	0: bars & rings; 1: bars; 2: rings; 100: region

RMxprt/SRMCore

Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaYoke for outer cores
DiaYoke	Core diameter on yoke side, DiaYoke < DiaGap for inner cores
Length	Core Length
Poles	Number of poles
ThkYoke	Toke thickness
Embrace	Pole embrace (the ratio of pole arc to pole pitch)
EndExt	Coil one-side end extended length
LenRegion	Region length
InfoCore	0: core; 1: core & coils; 2: coil; 3: terminal1; 4:terminal2; 100: region

RMxprt/SynRMCore

Property	Description
DiaGap	Core diameter on gap side, or outer diameter

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DiaYoke	Core diameter on yoke side, or inner diameter
Length	Core Length
Poles	Number of poles
PoleType	1: ALA; 2: arc; 3: hyperbolic; 4: hyperbolic line
Barriers	Barriers per pole, for PoleType 2 & 3 only
H	Bridge thickness, for PoleType 2 & 3 only
W	Rib width, for PoleType 2 & 3 only
R	Barrier fillet radius, for PoleType 2 & 3 only
R0	Radius of the bottom barrier arch, for PoleType 2 & 3 only
Rb	Barrier bottom minimum radius
Y0	Yoke bottom thickness
B0	Barrier bottom thickness, for PoleType 2 & 3 only
LenRegion	Region length
InfoCore	0: core; 1: one barrier; 100: region

RMxprt/TransCoil

Property	Description
DistLeg	Leg center to center distance
CoilType	Coil type: 1 for solenoid coil; 2 for pancake coil
WidthIn	Coil width between two inner sides
DepthIn	Coil depth between two inner ends
RadiusIn	Coil inner fillet radius
ThickCoil	Coil thickness of one side
HighCoil	Coil Height
Layers	Number of layers
GapLayer	Gap between two layers
InfoCore	0: all coils; 1: one coil only

RMxprt/TransCore

Property	Description
DiaLeg	Outer diameter of leg cross-section
DistLeg	Leg center to center distance

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

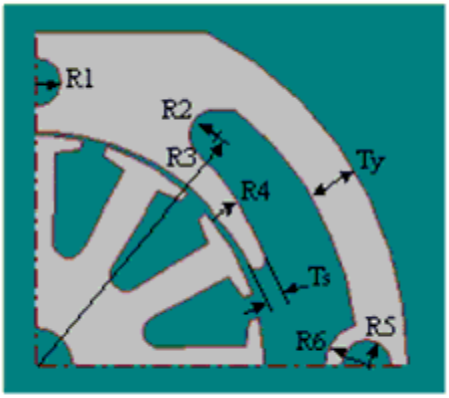
DistYoke	Yoke center to center distance
Stages	Number of stages of leg cross-section
ThickCore	Core thickness, only used for Stages=1
WidthYoke	Yoke width, =0 for same cross-section as leg's
InfoCore	0: whole core; 1: legs only; 2: yokes only

RMxprt/UnivMCore

See graphic below for definitions.

Property	Description
DiaGap	Core diameter on gap side, or outer diameter
DiaYoke	Core diameter on yoke side, or inner diameter
Length	Core Length
Skew	Skew angle in core length range
Poles	Number of poles
PoleType	Pole type: 1 to 2
WidthCore	Overall width of a racetrack core
Embrace	Pole embrace (the ratio of pole arc to pole pitch)
Offset	Pole arc offset
WidthPole	Minimum width at pole root
Ty	Yoke thickness
Ts	Shoe-tip thickness
R1	Radius of the screw holes in pole center
R2	Radius of the side fillet arc at pole root
R3	Radius of the side fillet arc center layout circle
R4	Radius of the shoe connecting arc, 0 for auto-design
R5	Inner radius of the screw holes in between two poles
R6	Outer radius of the screw holes in between two poles
EndExt	Coil one-side end extended length
LenRegion	Region length
InfoCore	0: core; 1: coils; 2: one coil; 3: terminal1; 4: terminal2: 100: region

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



RMxprt/VentSlotCore

Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaYoke for outer cores
DiaYoke	Core diameter on yoke side, DiaYoke < DiaGap for inner cores
Length	Core Length
Skew	Skew angle in core length range
Slots	Number of slots
SlotType	Slot type: 1 to 7
Hs0	Slot opening height
Hs01	Slot closed bridge height
Hs1	Slot wedge height
Hs2	Slot body height
Bs0	Slot opening width
Bs1	Slot wedge maximum width
Bs2	Slot body bottom width, 0 for parallel teeth
Rs	Slot body bottom fillet
FilletType	0: a quarter circle; 1: tangent connection; 2 & 3: arc bottom
HalfSlot	0 for symmetric slot, 1 for half slot
VentHoles	Number of axial vent holes per row
HoleDiaIn	Diameter of inner vent holes
HoleDiaOut	Diameter of outer vent holes
HoleLocIn	Diameter of inner vent hole center layout circle

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HoleLocOut	Diameter of outer vent hole center layout circle
VentDucts	Number of radial vent ducts
DuctWidth	Axial width of radial vent ducts
DuctPitch	Center-to-center distance between two adjacent ducts
LenRegion	Region length
InfoCore	0: core; 100: region

RMxprt/WaveCoil

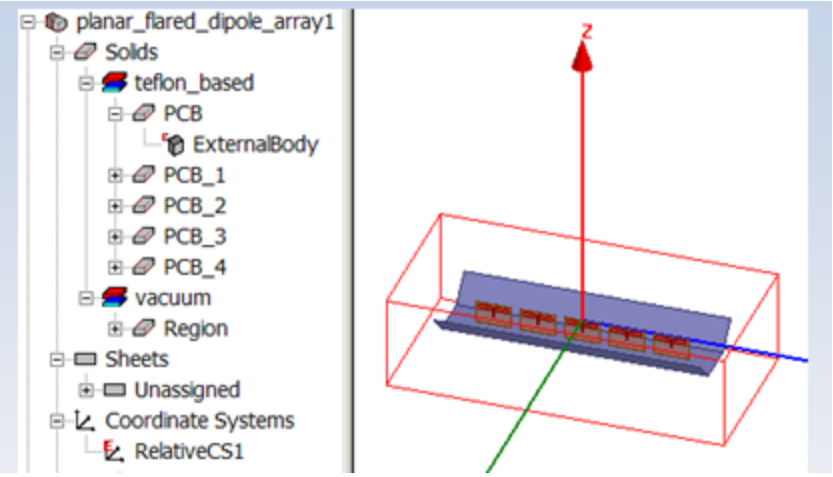
Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaYoke for outer cores
DiaYoke	Core diameter on yoke side, DiaYoke < DiaGap for inner cores
Length	Core Length
Skew	Skew angle in core length range
Slots	Number of slots
SlotType	Slot type: 1 to 7
Hs0	Slot opening height
Hs1	Slot wedge height
Hs2	Slot body height
Bs0	Slot opening width
Bs1	Slot wedge maximum width
Bs2	Slot body bottom width, 0 for parallel teeth
Rs	Slot body bottom fillet
FilletType	0: a quarter circle; 1: tangent connection
Layers	Number of winding layers
CoilPitch	Coil pitch measured in slots
EndExt	One-side end extended length
SpanExt	Axial length of end span; 0 for no span
SegAngle	Angle per segment of end span; 0 for true-surface end span
PolePitch	Pole pair pitch measured in slots
LenRegion	Region Length
InfoCoil	0: winding; 1: one coil; 100: region

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

User-Defined Model (UDM)

A User-Defined Model (UDM) is collection of externally defined parts imported into an Ansys EM 3D Modeler or created using a C or Python script.

- UDM includes part attributes (like name, color etc) and material assignment
- UDM can also have external coordinate systems and corresponding planes
- UDM parts can be parameterized and manipulated in Ansys EM modeler just like any other part



UDM can either represent static geometry models or dynamic links to models of external geometry editors. It is used for supporting CAD integration in WorkBench

UDM uses same plugin technology as User Defined Part (UDP).

See the following sections:

UDM compared to User Defined Primitives

Insert UDM Command on Draw Menu

UDM Properties

Library of Models for CAD Integration

Ansys EM to Ansys Geometry Transfer

Material Assignment Transfer

Geometry Transfer through Ansys DesignModeler (DM)

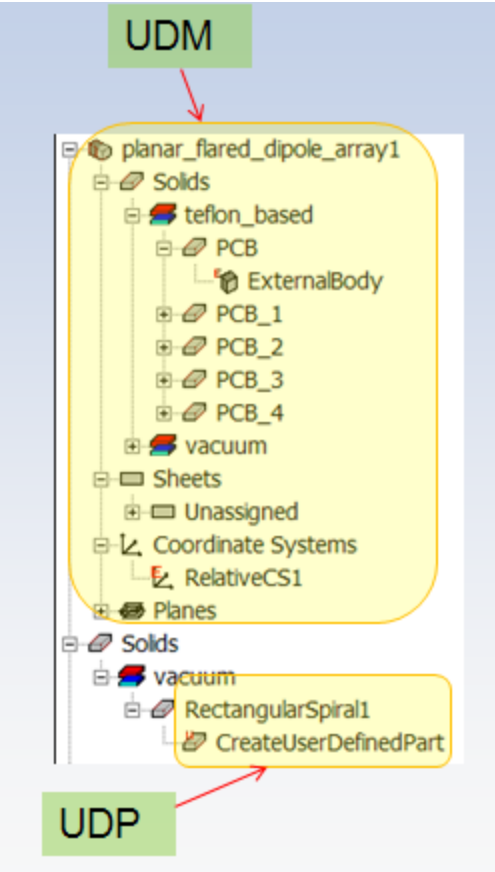
CAD Integration Functionality

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See "Creating User-Defined Primitives and User-Defined Models in Python Scripts" in the **Maxwell Scripting Guide** for information.

UDM Compared to User-Defined Primitives

User-Defined Models (UDM) resemble User-Defined Primitives (UDP):



- Ansys EM products can be extended by users through new UDMs.
- UDM plugins are discovered by searching standard directory paths .
- Plugins for static UDM can build model using 'callback interfaces' (like create-box, create-cylinder, subtract etc) similar to UDP.
- UDMs run inside Ansys EM applications.
- UDMs provide geometry, topology, persistence and parameters

In contrast to UDP:

- UDM provides multiple Parts/CS/etc.

  UDP provides primitive operation only for a single Part.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

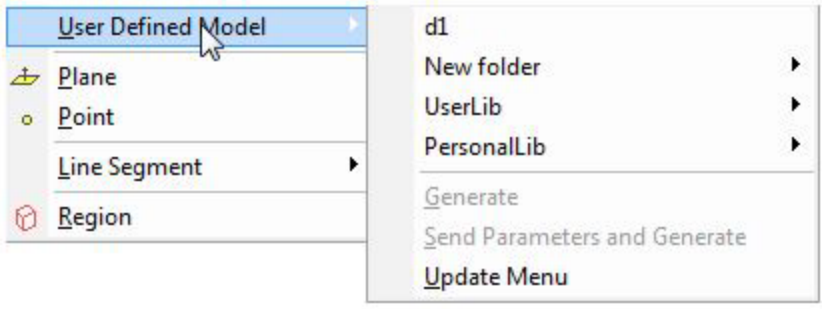
- UDM provides part attributes and material assignment.
- UDP does not define part attributes or material.

UDM Properties have four tabs - Definitions, Parameters, Options and Info.

- Definition tab has:
  - UDM name
  - Coordinate system used to position UDM
  - May have external reference to file
- Info tab has:
  - UDM dll or .py name, dll or .py location, version etc
- Option tab:
  - may have options if any

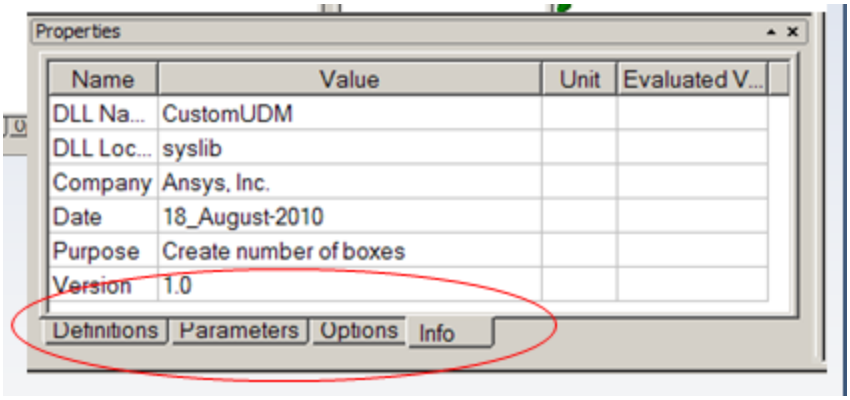
Insert UDM Command on Draw Menu

To insert a UDM into a design, use the User Defined Model command on the Draw menu for the Modeler window.



UDM Properties

UDM Properties have four tabs - Definitions, Parameters, Options and Info.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Definition tab has

- UDM name
- Coordinate system used to position UDM
- May have external reference to file

Info tab has:

- UDM dll or .py name, dll or .py location, version etc

Option tab:

- may have options if any

UDM Parameters

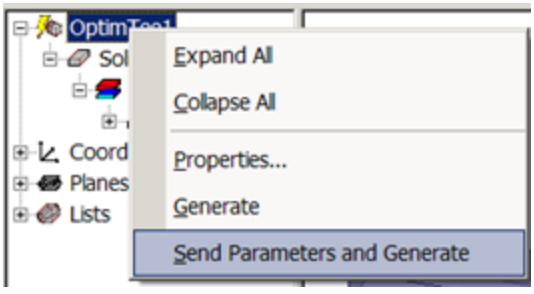
UDM geometry can be manipulated through its parameters.

- Can be mapped to design or project variable for animation, parametric analysis.
- IDs are persisted (allowing to retain boundaries) during parameter edits.

  UDM geometry in HFSS invoked through Workbench is not 'dynamic updated' upon parameter edits.

- In such cases, UDM shows a lightning bolt icon by the model name when parameters are edited.

- You must run the **Send Parameters and Generate** command to synchronize parameters with geometry



UDM Part Edits

Several modeling operations are allowed on UDM parts:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Operations will be part of history tree and retained during model refresh

The following operations are not allowed:

- Non history tree operations like healing, defeature.
- Operations which use UDM parts as tool, such as sweep or boolean (but allowed when clone tool option is selected)

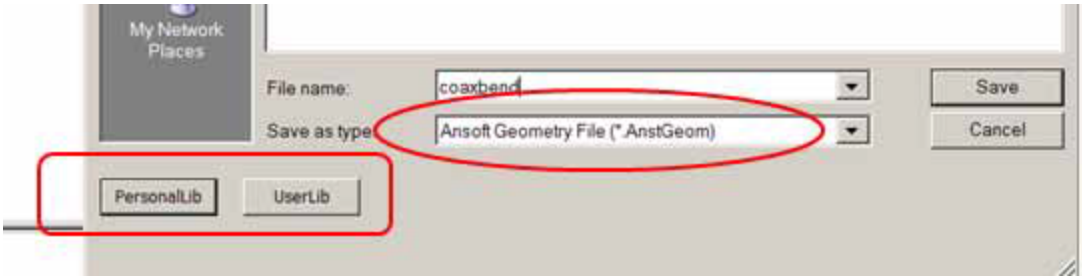
The following part attributes can be modified for UDM parts:

- Model/Non Model flag
- Solve Inside flag
- Part orientation

Library of Models for CAD Integration

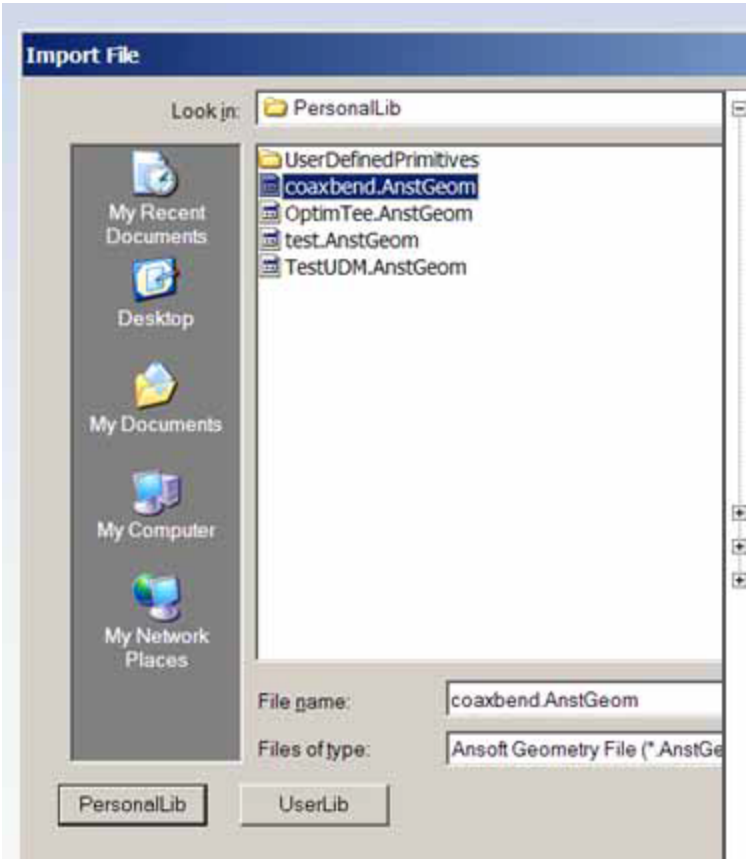
UDM technology allows a library of models

- Any Ansys EM model can be exported as 'Ansoft Geometry File'

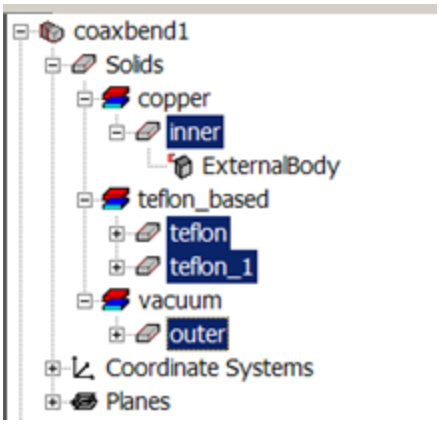


- An Ansoft Geometry File can be imported back as a UDM.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Any Design/project variables associated with model are brought in as UDM parameters.



Geometry computation for UDM (and also UDP) can be specified in the Modeler Options as either done on.

- Engine side (default):

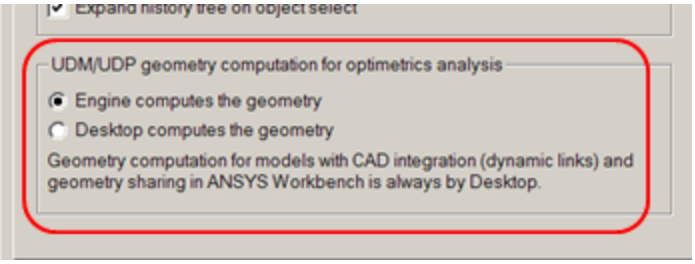
  Requires deployment of UDM on each node

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Desktop side:

  UDM need not be deployed on each engine

  Desktop will be busy during parametric analysis



3D Modeler Discovery Link

The 3D Modeler Discovery Link allows you to create a User-Defined Model (UDM) from the geometry model in an Ansys Discovery project. Discovery Link can be used only if Discovery with the same version as Ansys Electronics Desktop is installed on the same machine. Discovery Link is only available on Windows.

Discovery Link provides following functionality:

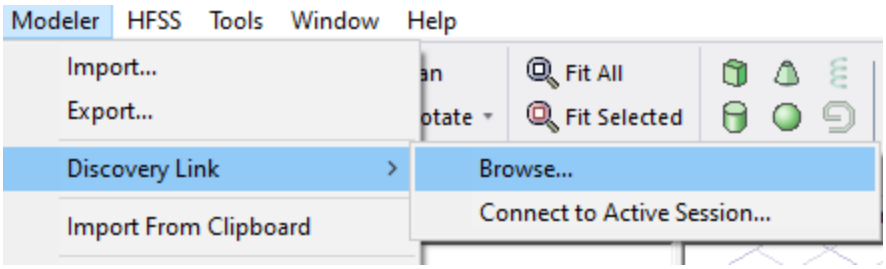
- Browse and select Discovery project files (*.dsco) to import geometry directly.
- Import geometry by connecting to an active Discovery session.
- The Discovery geometry is imported as a UDM along with geometry parameters with units, rendering attributes and material assignments. Object, material, and parameter names with non-ASCII characters are not supported.
- The Discovery geometries can be imported as faceted bodies if the design supports lightweight geometries.
- A Discovery document referenced by a Discovery link is included in the project archive automatically.
- You can break a Discovery link. In this case, all the parts in the UDM are converted to imported objects.

Importing Discovery Geometry into Ansys Electronics Desktop

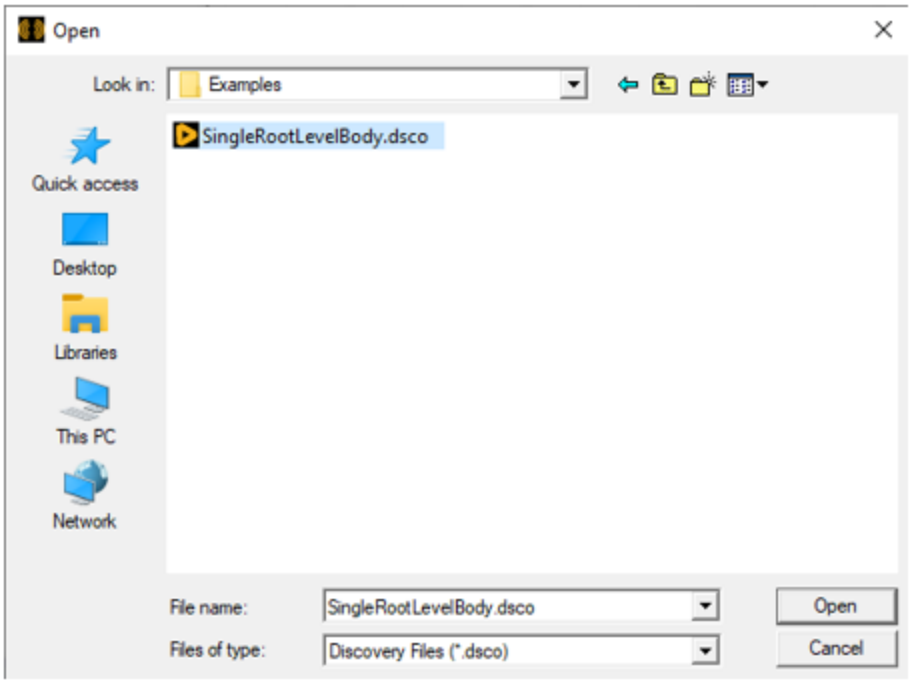
To select a Discovery file:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select **Modeler** > **Discovery Link** > **Browse** or **Browse Discovery** in the **Model** ribbon.



2. Navigate to and select a *.dsco project to load into the Modeler window.

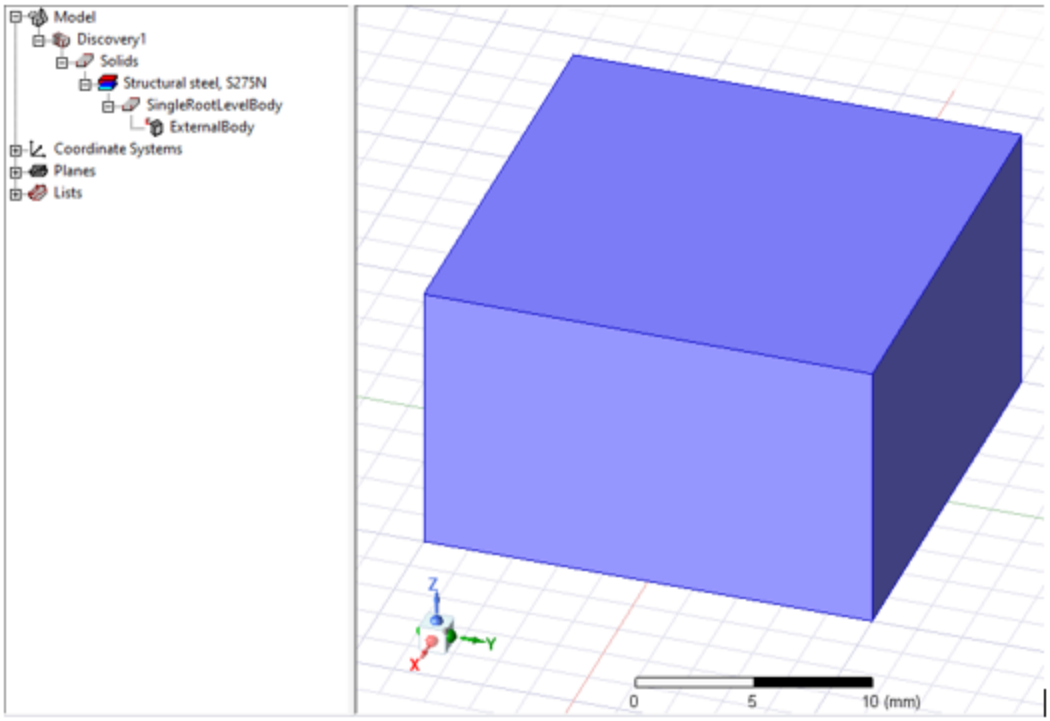


3. Click **Open**.

   The progress bar displays import progress. When the model has loaded, it appears in the Modeler window, and in the history tree as a User-Defined Model (UDM) listed under

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

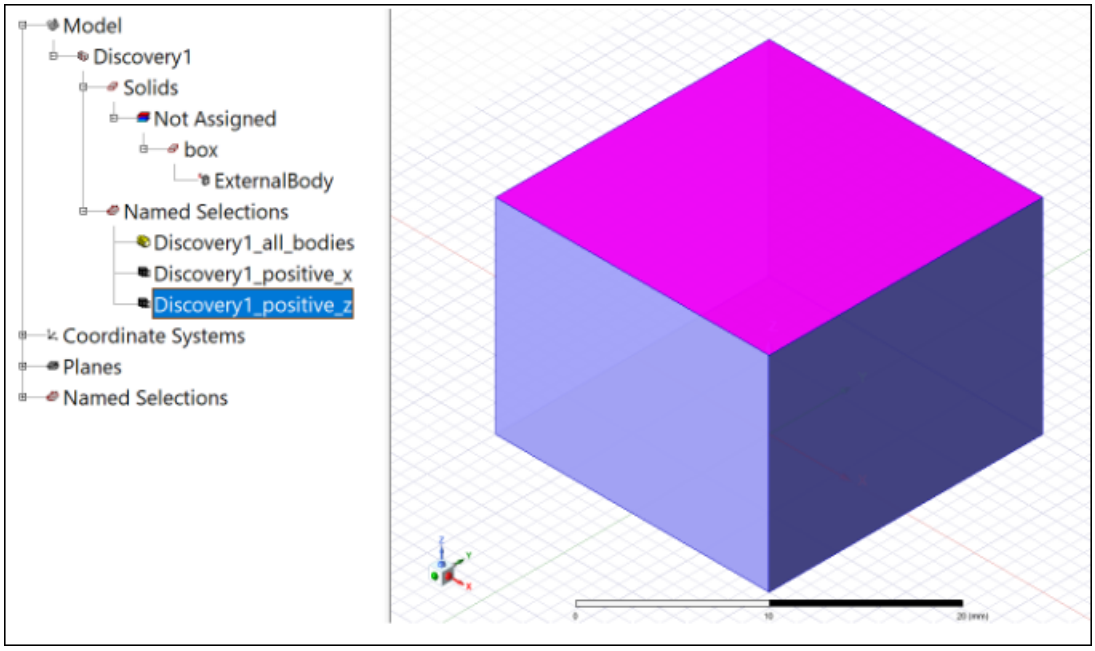
Discovery.



If there is an active Discovery session which is using the selected document, the created UDM will be linked to the active Discovery session. If there is not one, a new non-graphical Discovery session will be used, which makes the creating and refreshing slow. Afterwards, if an active Discovery session is launched and opens the document, the UDM will be linked to the active session.

Note that if the **Import Named Selections** option is checked in the **Tools** > **Options** > **General Options** > **3D Modeler** > **Discovery Link** window, named selections created in Discovery will be listed in the **Named Selections** branch of the history tree:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To synchronize imported named selections with Discovery, right-click on the UDM (*Discovery1* in the preceding example), and select **Generate** or **Send Parameters and Generate**. Within Ansys Electronic Desktop, the imported named selections cannot be directly modified or deleted without deleting the whole UDM.

Preserving IDs During Parametric Variations

When a model is imported, IDs in Ansys Electronic Desktop depend on IDs available from Discovery or other source CAD software. If parameters in Discovery are set up such that changing their values causes existing parts to be edited, then IDs in Discovery and Ansys Electronic Desktop remain constant across parametric variations.

However, if IDs in the source CAD or Discovery change during a parametric update, then Ansys Electronic Desktop cannot match object/faces across variations. Ansys Electronic Desktop might delete original objects and create new objects, causing deletion of boundary conditions and failures in parametric simulations. Such a situation can commonly occur when an embedded script is used in Discovery, and the script clears all parts at the beginning of script.

Named selections can be used in such situations to preserve IDs of objects/faces so that parametric simulations can proceed. In such cases, use the following guidelines:

1. The total number of objects must be kept the same across parametric variations.
2. If part IDs are changing in Discovery, users must include every part in at least one object named selection and keep the name of each part constant across parametric variations.

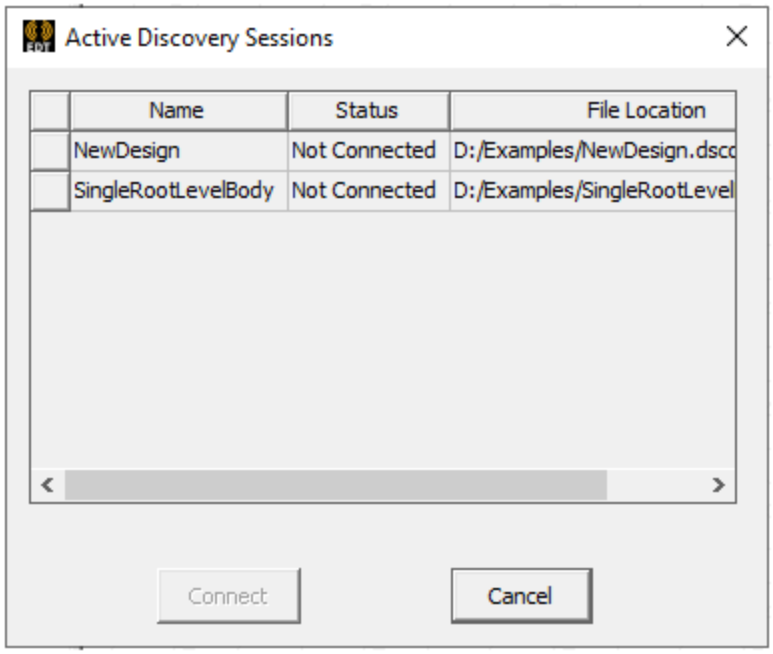
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Discovery can have multiple parts with a same name. Ansys recommends assigning a unique name to every part and creating an object named selection that includes all parts in the model. If duplicate names cannot be avoided, then multiple named selections can be created so that parts with the same name are in different named selections. Ansys Electronic Desktop will match objects in a named selection by using their names and preserve IDs across parametric variations. This will preserve boundary conditions or other assignments in Ansys Electronic Desktop.
4. If a named selection has a single face, then the ID of that face in Ansys Electronic Desktop will be kept constant across parametric variations even if it changes on Discovery side. If a named selection has multiple faces and the IDs of these faces change in Discovery, then Ansys Electronic Desktop cannot guarantee that the ID of each face in that named selection will be preserved consistently. It is possible that IDs may switch among faces in that named selection. Ansys recommends creating a separate named selection for each face on which boundary/excitation is added in Ansys Electronic Desktop.
5. If the number of faces in a named selection changes across parametric variations, then the face IDs may not remain constant.

Connecting to an Active Discovery Session

If one or more Discovery sessions are running on the machine, Discovery link can connect to them to import geometry.

When you select **Modeler** > **Discovery Link** > **Connect to Active Session** or **Connect to Discovery** in the Model ribbon, the Active Discovery Sessions dialog box appears.



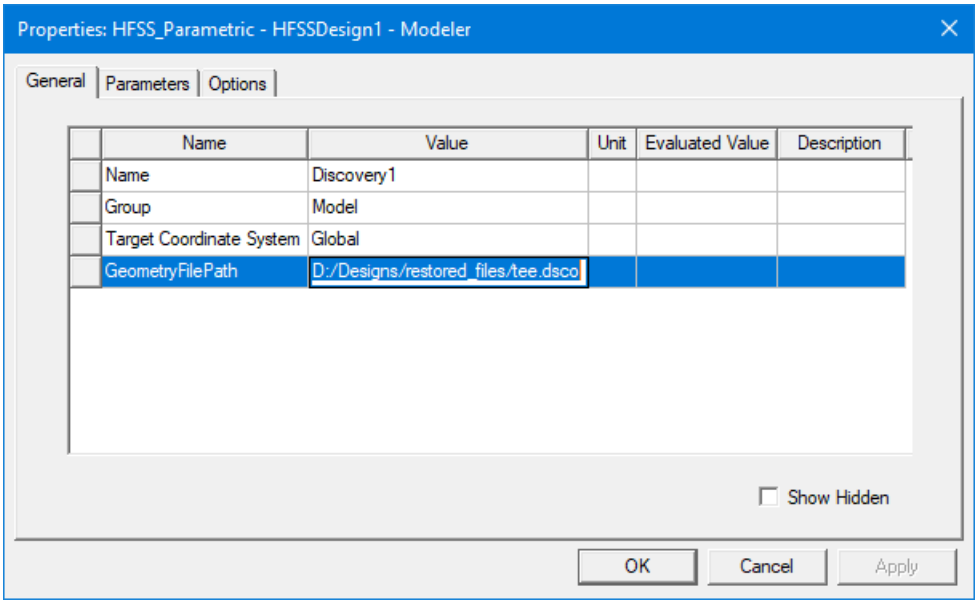
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

It shows all active Discovery sessions with their document names and connection status. Sessions which are not using a saved document are not included. You can select a session and click **Connect** to import geometry from that session. Once connected, that session will be used to regenerate the corresponding Discovery link UDM.

Property Window of Discovery Link UDM

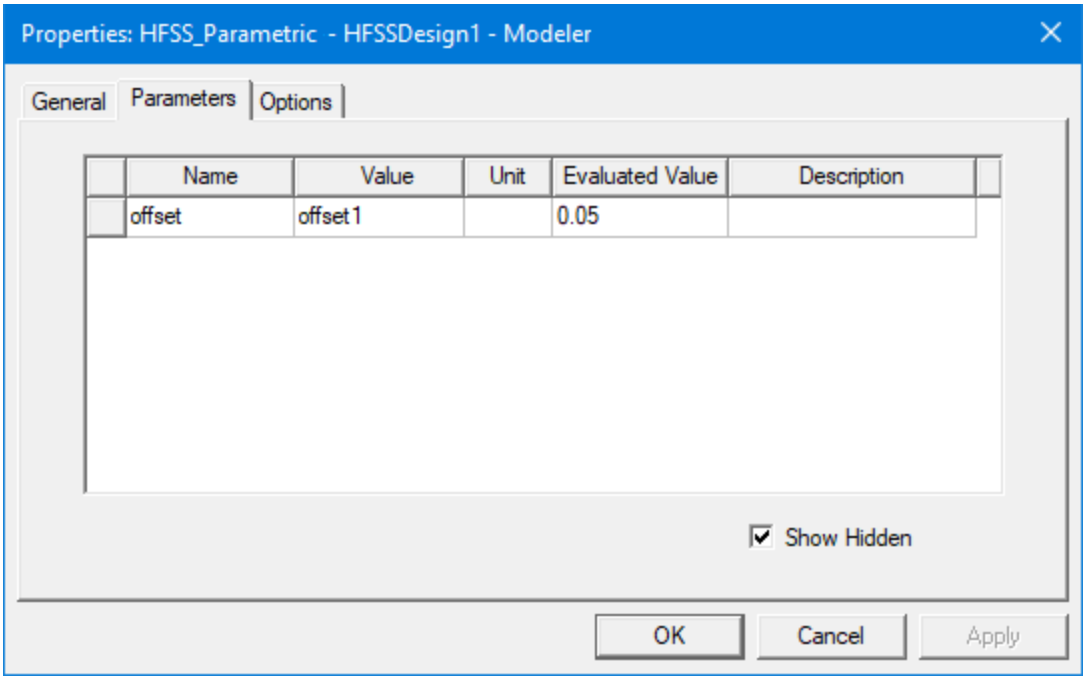
The Property window of a Discovery link UDM has three tabs:

- The **General** tab shows general information such as Discovery document path. Note that the Geometry File Path is editable; users can update the path if the original file is moved.



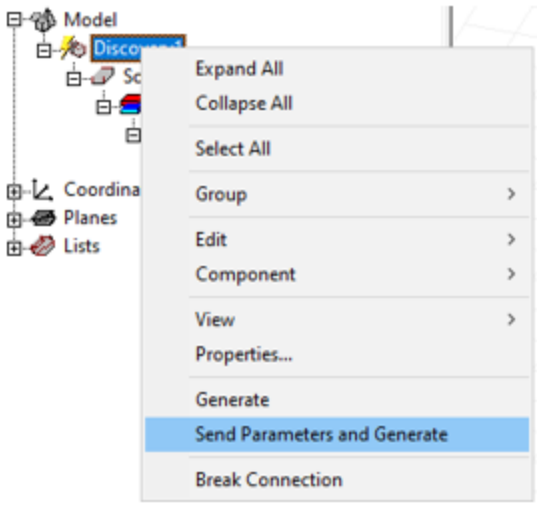
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **Parameters** tab shows all parameters.



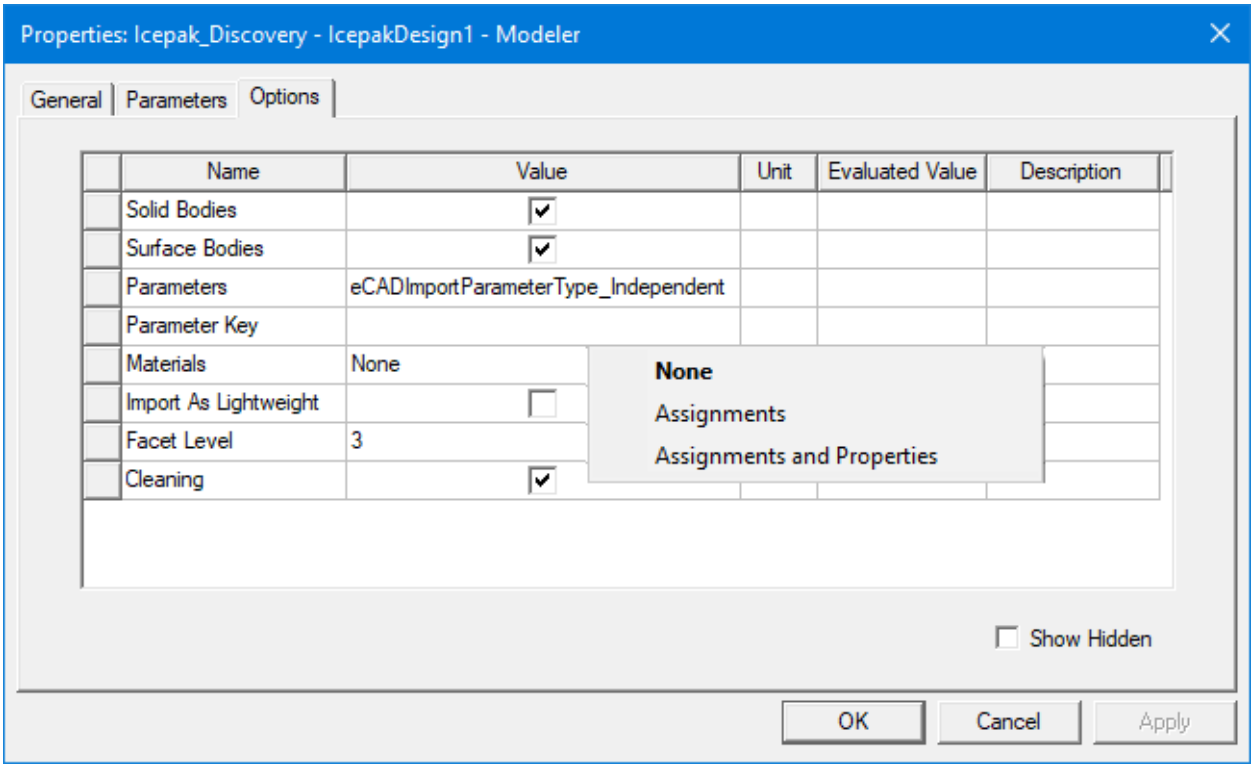
You can edit the parameters and send them back to Discovery to get the geometry updated accordingly.

After editing the parameter, you will need to use **Send Parameters and Generate** in the context menu of the Discovery link UDM. If there is an active Discovery session linked, its model will be updated but not saved. If there is not an active Discovery session linked, the Discovery document will not be updated.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **Options** tab shows supported options. Note that Materials**Assignments and Properties** is only available for Icepak and Mechanical designs. See Discovery Link Options for details on the options in this window.



Archive a Project Containing Discovery Link

If you want to share a project with Discovery link with other users, you need to archive the project, because the linked Discovery file is not included in the project file. When archiving, the Discovery documents corresponding to the links will be included in the project archive automatically. Once the archive file is unarchived, referenced Discovery documents will be put in restored_files subfolder in the project directory. All definition paths are updated to refer to the reference files in restored_files folder.

Break Connection with Discovery Link

A **Break Connection** command is available in the context menu of the Discovery link UDM.

- **Break Connection** converts the UDM bodies to imported objects. The UDM will be removed.
- Any operations that you have performed on UDM bodies are preserved.
- Parameters of the UDM are lost. You lose the ability to create parametric variations with

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discovery parameters.

- Attributes such as rendering attributes, materials, *etc.*, are preserved.

SpaceClaim Integration

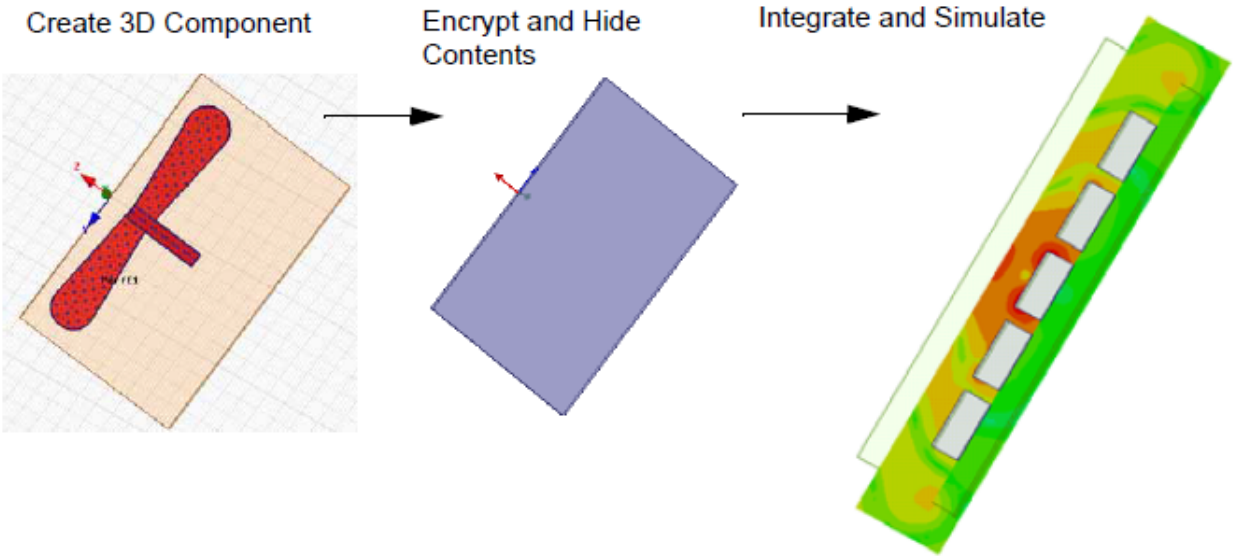
As of 2025R1, the direct link between the Ansys Electronics Desktop software and the SpaceClaim application has been discontinued. Users are encouraged to convert SpaceClaim documents to the Discovery application file format and to use the Discovery link (**Modeler** > **Discovery Link**) to import geometry into AEDT.

Legacy AEDT projects containing SpaceClaim links can be opened. Any regeneration of SpaceClaim geometry will open the SpaceClaim file in background session of the Discovery application. We recommend that users convert source *.scdoc* files to *.disco* files by opening and saving them in the Discovery application. Users can then modify the source file path under the UDM link to point to the Discovery file.

3D Component Library

The 3D Modeler lets you access predefined 3D component libraries and to create 3D components using:

- Geometry (objects and coordinate systems)
- Design data (boundaries, excitations and mesh operations)
- Material assignment and definition for components
- Component parameter, including variables used by geometries, design data, and material properties
- Optional logo or icon image to display with components in the modeler window
- Optional encryption with password definition or internal key as a password



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3D Component Libraries may be included as part of Electronics Desktop, downloaded from third-party vendors, or created from scratch.

The ability to create components from models lets you easily share and reuse components. You can add components to a library and share components with other users. Once you insert a component into the target design, you can directly manipulate whole components using **Arrange** operations such as **Move**, **Rotate** and **Mirror** or **Duplicate** operations like **Duplicate Along Line**, **Duplicate Around Axis** and **Duplicate Mirror**.

> **Note:**
>
> 3D Component Libraries can contain components created using the **Create 3D Component** menu command in both Maxwell 2D and Maxwell 3D projects. 3D Components created in Maxwell 2D projects cannot be used in Maxwell 3D projects, and vice-versa.

Instances of an inserted component are grouped by definition in the model history. Multiple instances of identical components permit mesh reuse.

> **Note:**
>
> Except where noted, the following sections apply to component libraries and components created in both Maxwell 2D and Maxwell 3D projects.

The documentation includes the following sections:

Creating a 3D Component from an Existing Model

Replace Selection with 3D Component

Layout Component in HFSS 3D (Beta)

Inserting a Component into a Design

Using the Component Libraries Window

Viewing Properties of 3D ComponentsViewing

3D Components in the Project Manager Window

3D Components in the History Tree

Editing 3D Component Properties

Editing a 3D Component Definition

Update 3D Component Definition

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Additional **Show**, **Show Only**, **Hide** and **Fit** features, helpful in working with 3D components are described here:

Fit All Objects in a View Window.

Fit Selected objects in a View Window.

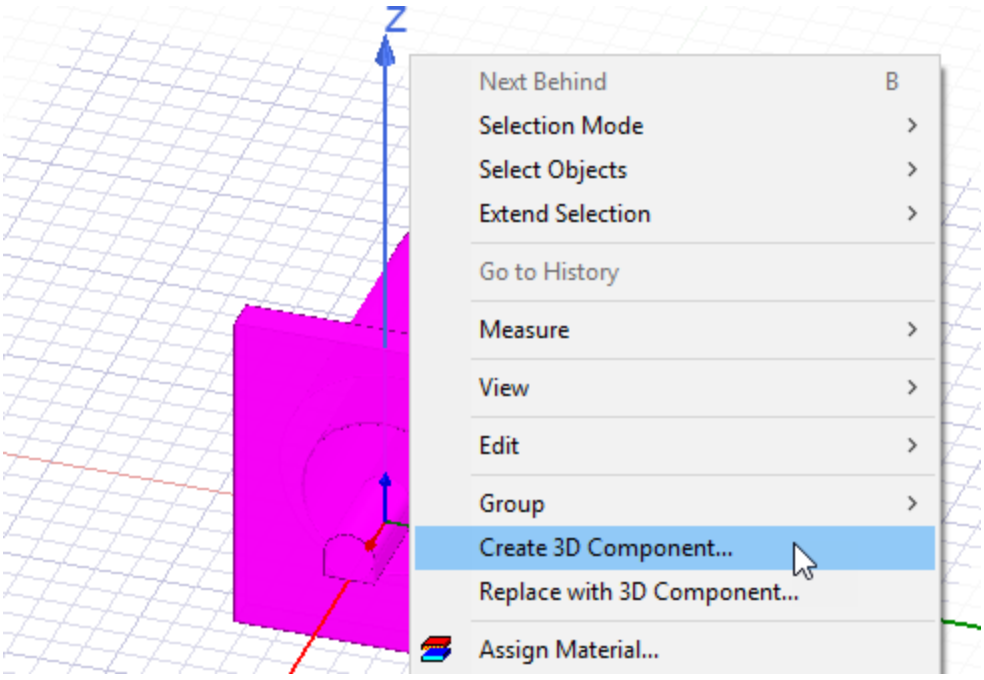
Showing Only Selected Objects in All or Active Views

Hiding Objects from View

Creating a 3D Component from an Existing Model

To create a 3D component, select an existing object or model, right-click, and select **Create 3D Component**. You can also click **Draw** > **3D Component Library** > **Create 3D Component**. You can also go to the **Model** tab of the ribbon and select **Create 3D Component**.

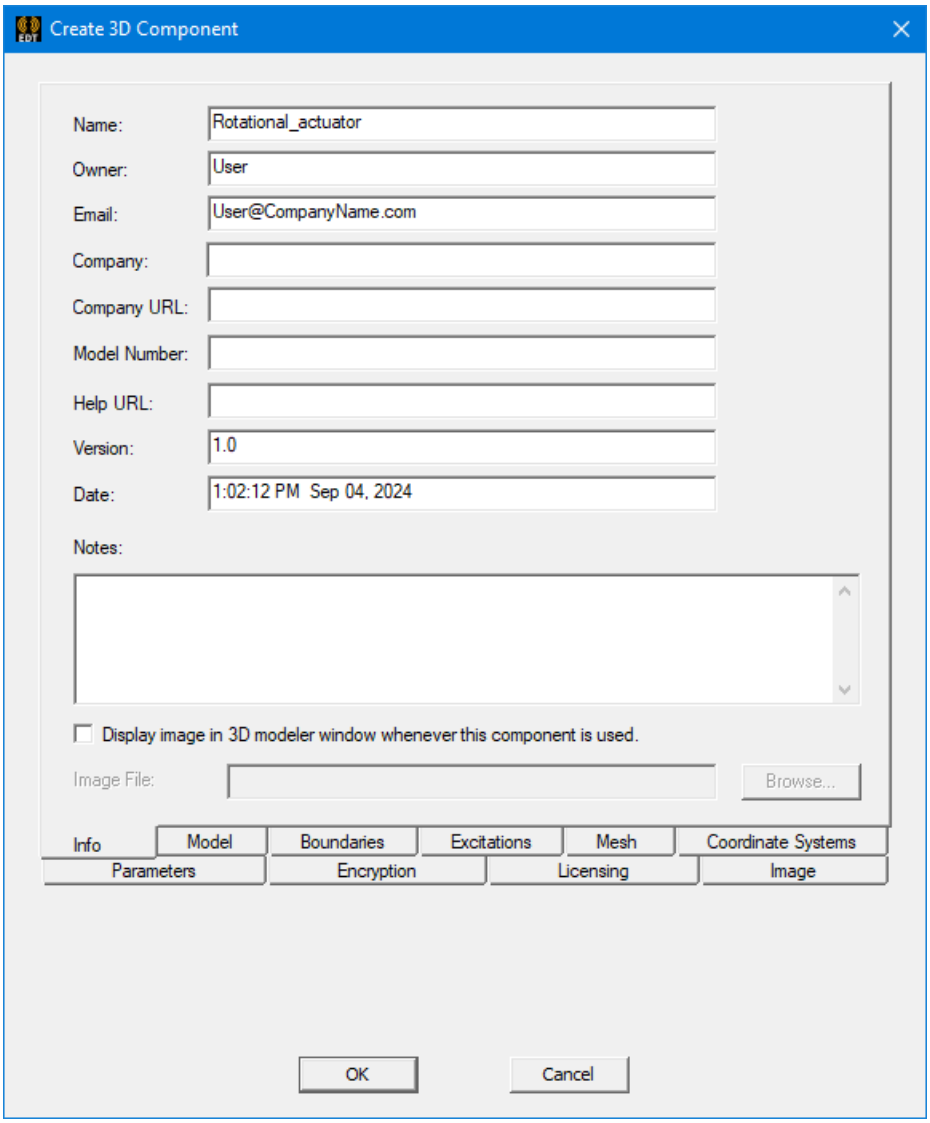
> **Note:**
>
> The Create 3D Component menu command can be used to create components in both Maxwell 2D and Maxwell 3D projects. 3D components created in Maxwell 2D cannot be used in Maxwell 3D projects, and vice-versa.



This opens the *Create 3D Component* dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You cannot create components from a design with an array setup. However, you can create 3D component from a unit cell and use that to create a Multi-Unit Cell array.



The tabs let you view the component features organized according to geometry, design data, and settings:

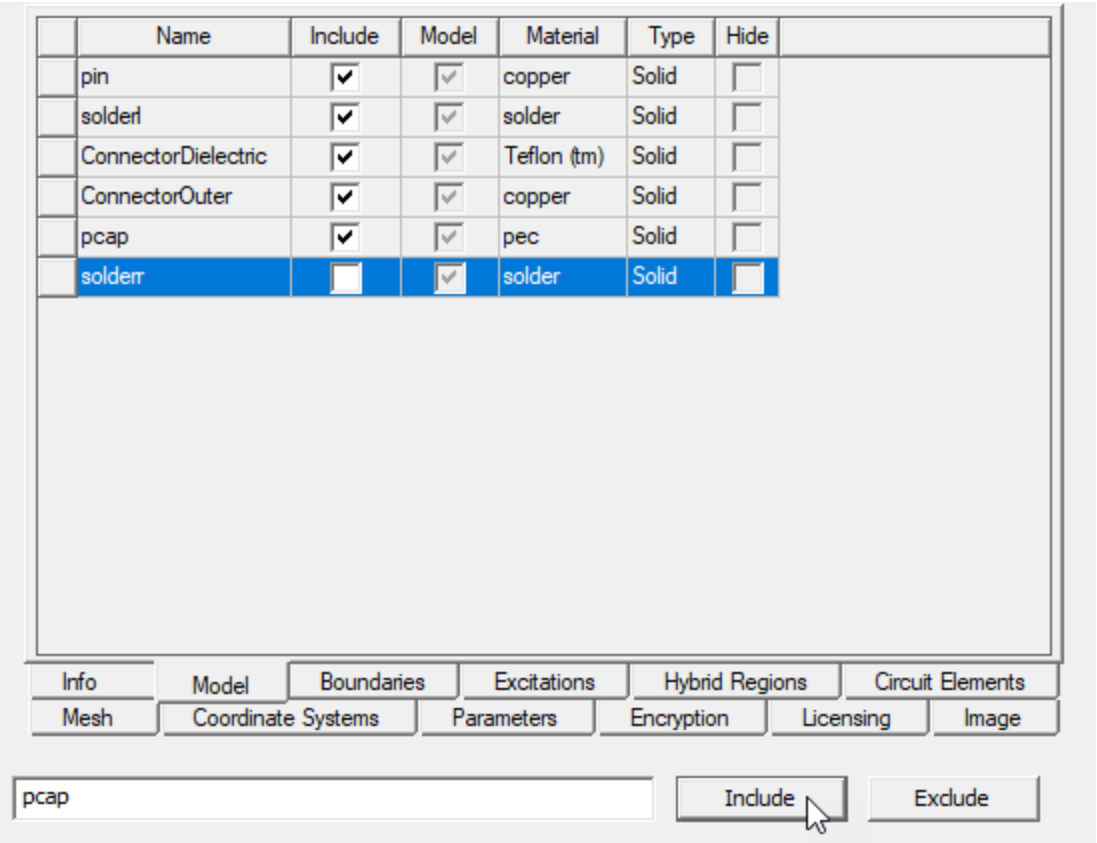
- Info tab
- Model tab
- Boundaries tab
- Excitations tab
- Circuit Elements tab
- Mesh tab

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Coordinate Systems tab
- Parameters tab
- Encryption tab
- Licensing Tab
- Image tab

Include or Exclude Feature from Component

You can view the various tabs and select which features to **Include** or **Exclude** in creating the component. You can use the check boxes in the Include column for this purpose. You can also use the text field. Type a feature Name that you can then use the command buttons to Include or Exclude.

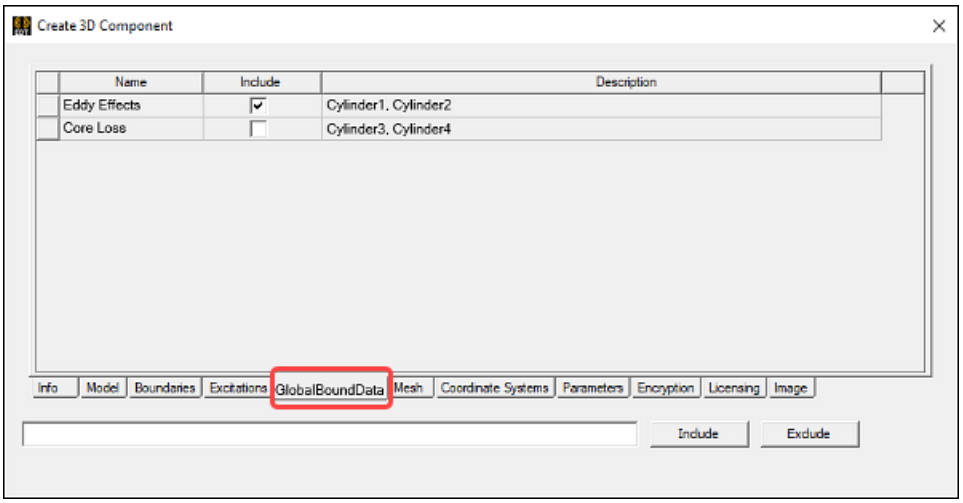


Object selection for **Include** or **Exclude** impacts selection in other tabs. For example, if you exclude an object from the component creation, the design data, parameters, and coordinate systems corresponding to that object are also excluded.

To include eddy effect or core loss in a created 3D component, you must first turn on the **3D Component Enhancement** Beta feature. To access the beta options, select **Tools** > **Options** > **General Options**, and in the window that opens, click **Beta Options**. Check this beta feature in the **Maxwell** section of the Beta Options dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When this beta option is activated, the Create 3D Component dialog box includes a **GlobalBoundData** tab, which has entries for eddy effect and/or core loss, if applicable.



Sorting Columns Using the Headers

The features listed for each tab include columns for the Name, check boxes for whether to include, and properties. You can click on the column headers to sort by Name, Model, or other feature listed for each tab. Click the column header to selects the column to sort. Click again to invert the column.

You can sort lists by using all columns except Include.

Design Data Notes:

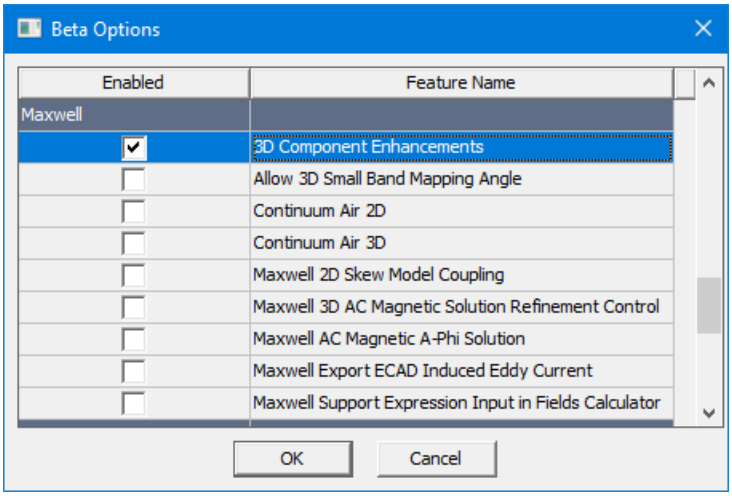
- DC thickness on selected objects is always included in components. It is not listed in **Boundaries** tab.
- The **Create 3D Component** dialog does not list design data without an assignment(for example, a winding in Maxwell).
- The parent of any included boundary/excitation is included, as long as the parent does not require assignment.
- Design settings like material overrides are included.

Once you have made the Include and Exclude settings, and have specified the Image and Info you click **OK** to Save 3D Component File. When you click OK, the component is validated for everything that is included in the component. For example, you cannot include boundaries if the object on which the boundaries have been created is not included. After validation, you see a dialog for saving the component to a location.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating a 3D Component with an External Circuit

If you are creating 3D component that has an external circuit, you can maintain windings and external circuit information by turning on the **3D Component Enhancement** Beta feature. To access the beta options, select **Tools** > **Options** > **General Options**, and in the window that opens, click **Beta Options**. Check this beta feature in the **Maxwell** section of the Beta Options dialog box:

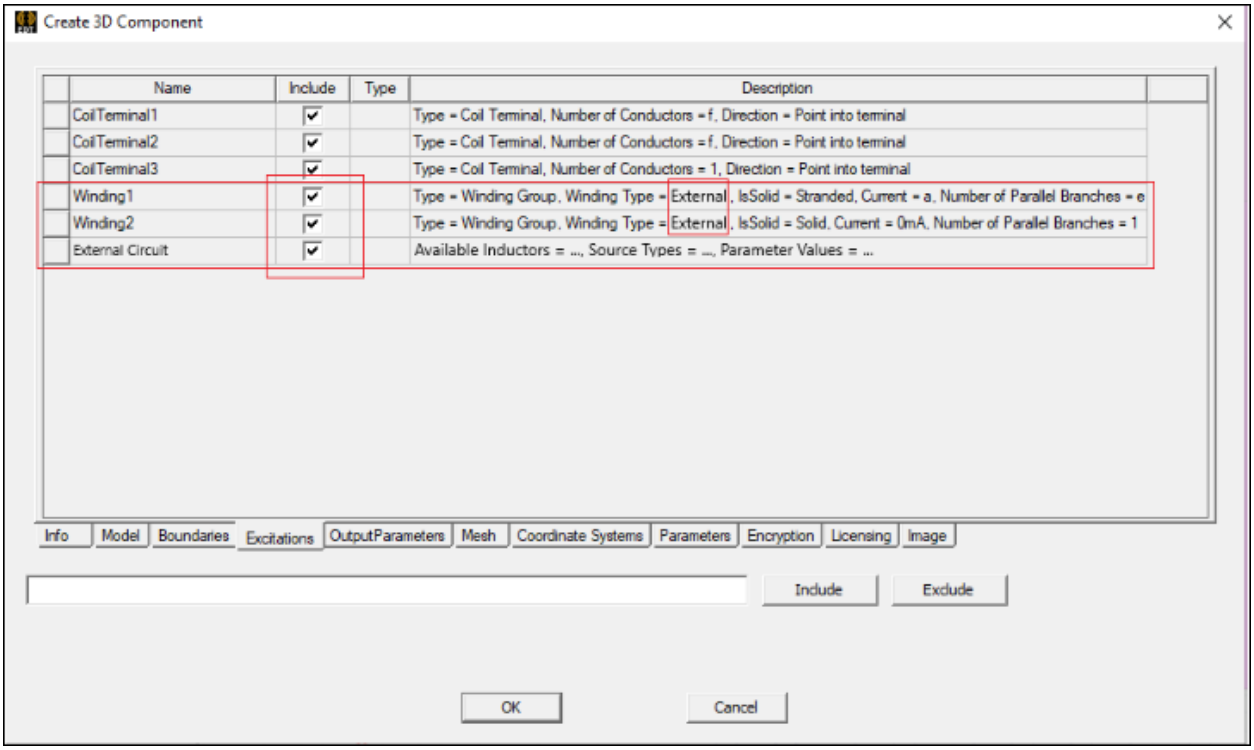


When creating a 3D component that includes an external circuit, note the following:

- The external circuit is limited to one created by Maxwell Circuit.
- Only one external circuit is allowed.
- A 3D component with an external circuit is NOT supported for design with a Simulink circuit.
- If the design the 3D component is used in has an external circuit linked to it, it will be replaced by the circuit from the 3D component.

For a design with an external circuit, the circuit information will be included in the created 3D component only if **all** windings of the external circuit are included during component creation:

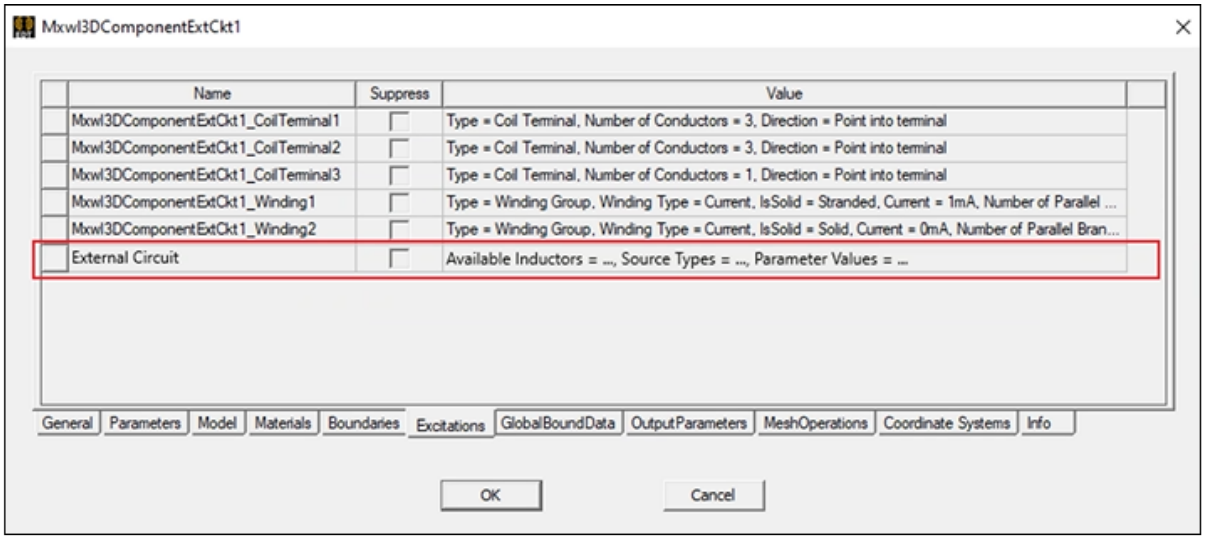
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

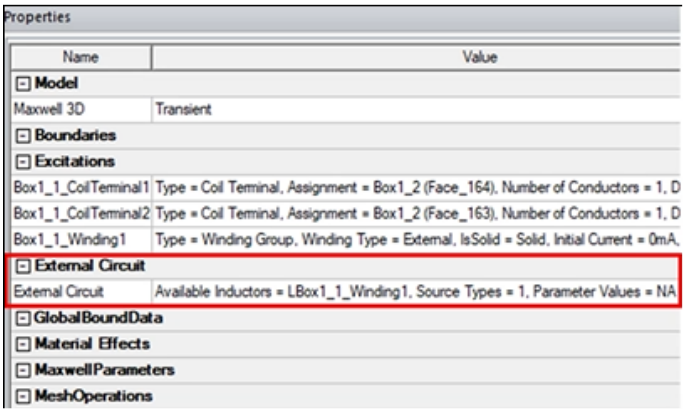


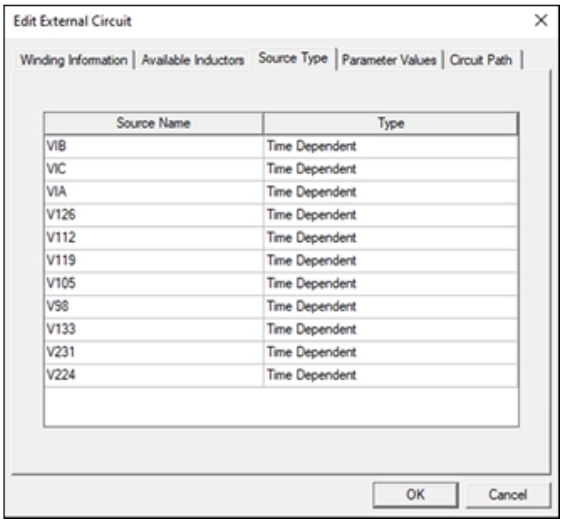
If the 3D component with an external circuit is not encrypted:

- The external circuit information is included in the 3D component's Properties window.
- The inductor names are shown in the Properties windows **Component Data** tab.
- The user can view the circuit information in the Edit External Circuit dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



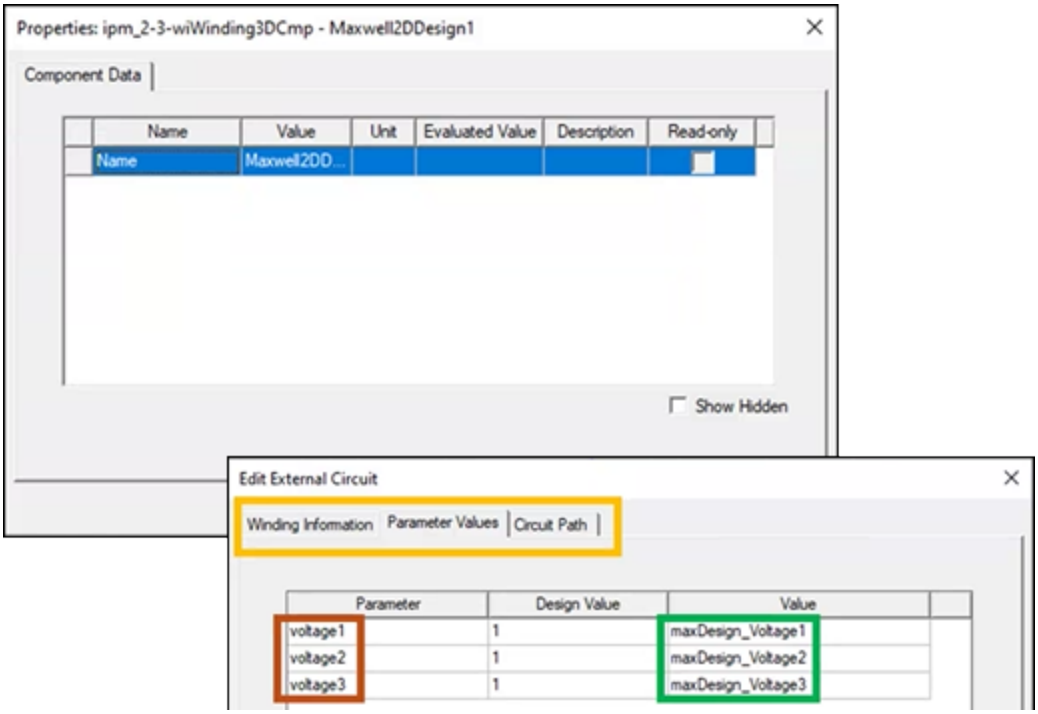




If the 3D component with an external circuit is encrypted:

- Only properties at the winding level will be shown in the Project Manager tree.
- Read-only attributes will be hidden in the winding's Properties window.
- The user can only see circuit information in the **Winding Information**, **Parameter Values**, and **Circuit Path** tabs of the Edit External Circuit window.

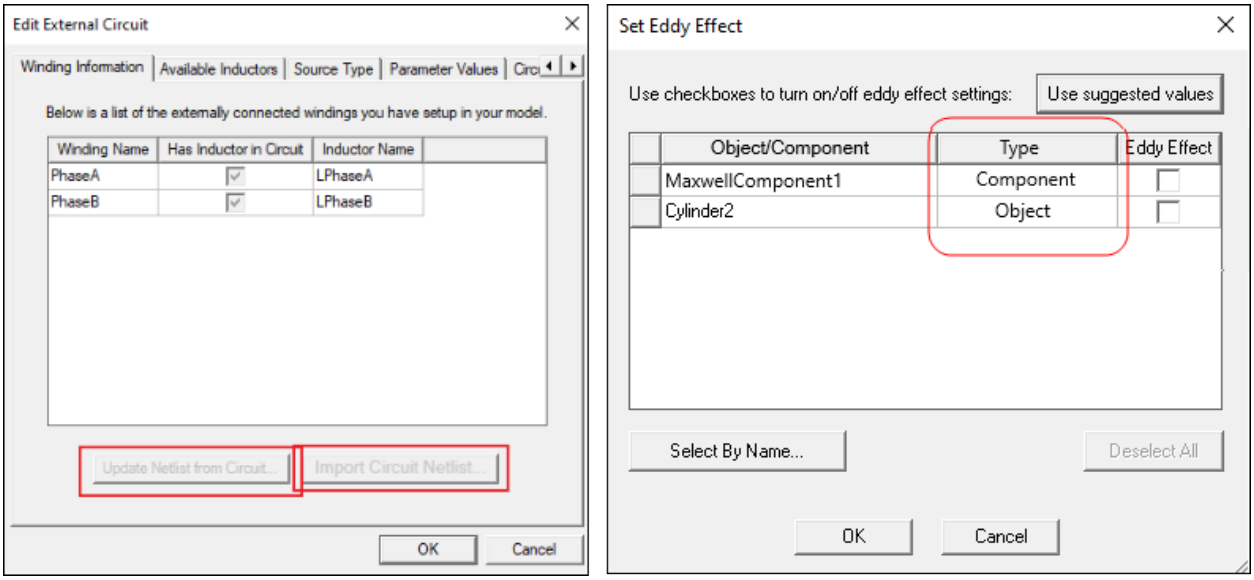
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For both non-encrypted and encrypted components with an external circuit:

- The user can control component data (*i.e.,* circuit data) through variables defined in the Maxwell design.
- The **Import Circuit Netlist** and **Update Netlist from Circuit** buttons will always be disabled in the Edit External Circuit dialog box.
- The user can modify the eddy effect and core loss selection on the entire component.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Create 3D Component: Model Tab

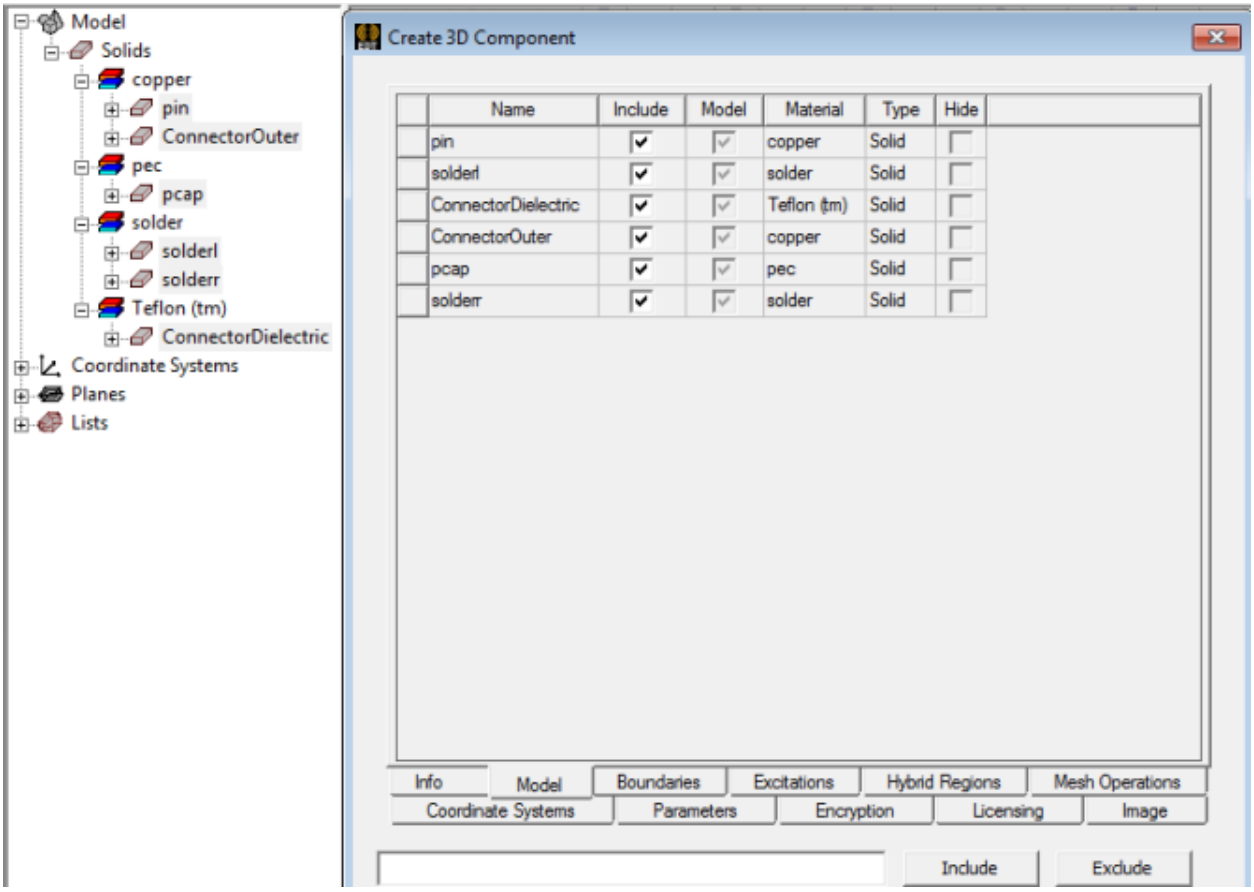
The **Model** tab lists all of objects selected before you invoked **Create 3D Component**. These correspond to the selected objects listed in the history tree.

- Use the **Include** check box to include objects in the 3D component (selected), or to Exclude them from the component (cleared).

  Alternatively, you can type an object **Name** into the text box and click the **Include** or **Exclude** button. Wildcards (? and *) are supported when you use this method, so you can change the inclusion state of multiple objects with similar names in a single operation.

- The **Material** and **Type** columns are read only.
- For Encrypted components, you can also chose which objects to hide.
- The **Hide** column is only enabled if you have the **Hide Selected Contents** option selected on **Encryption** tab.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Create 3D Component Boundaries Tab

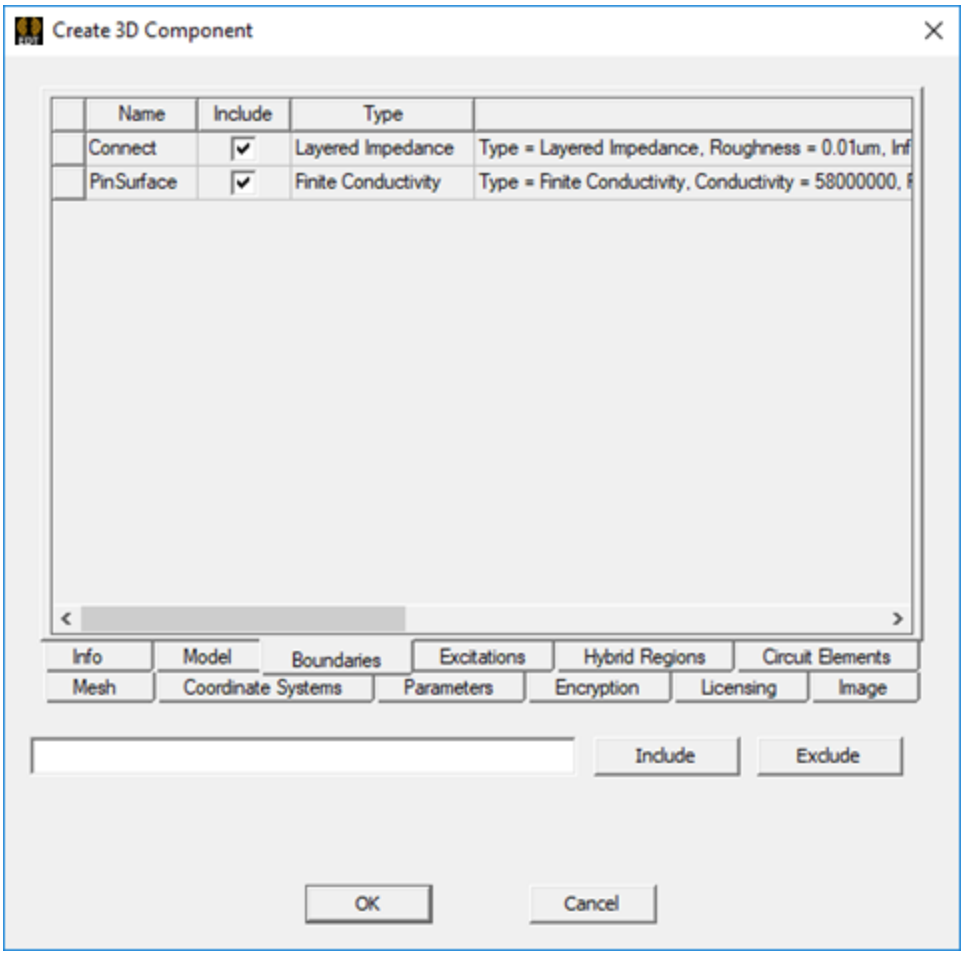
The list is populated with all the boundaries on selected objects.

- Use the **Include** check boxes to include (selected) or exclude (cleared) individual boundaries for the 3D component.

  Alternatively, you can type a boundary *Name* into the text box and click the **Include** or **Exclude** button. Wildcards (? and *) are supported when you use this method, so you can change the inclusion state of multiple boundaries with similar names in a single operation.

- The *Type* and *Description* columns are read only.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



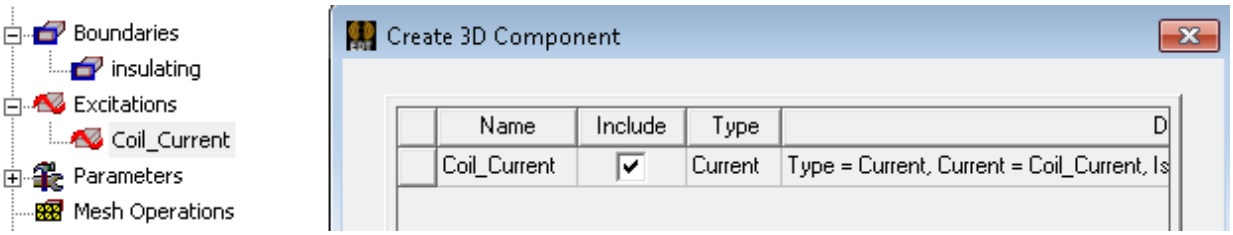
Create 3D Component Excitations Tab

The list is populated with all the excitations on selected objects.

- Use the **Include** check box to include excitations in the 3D component (selected), or to Exclude them from the component (cleared).

  Alternatively, you can type an excitation *Name* into the text box and click the **Include** or **Exclude** button. Wildcards (? and *) are supported when you use this method, so you can change the inclusion state of multiple excitations with similar names in a single operation.

- *Type* and *Description* columns are read only.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

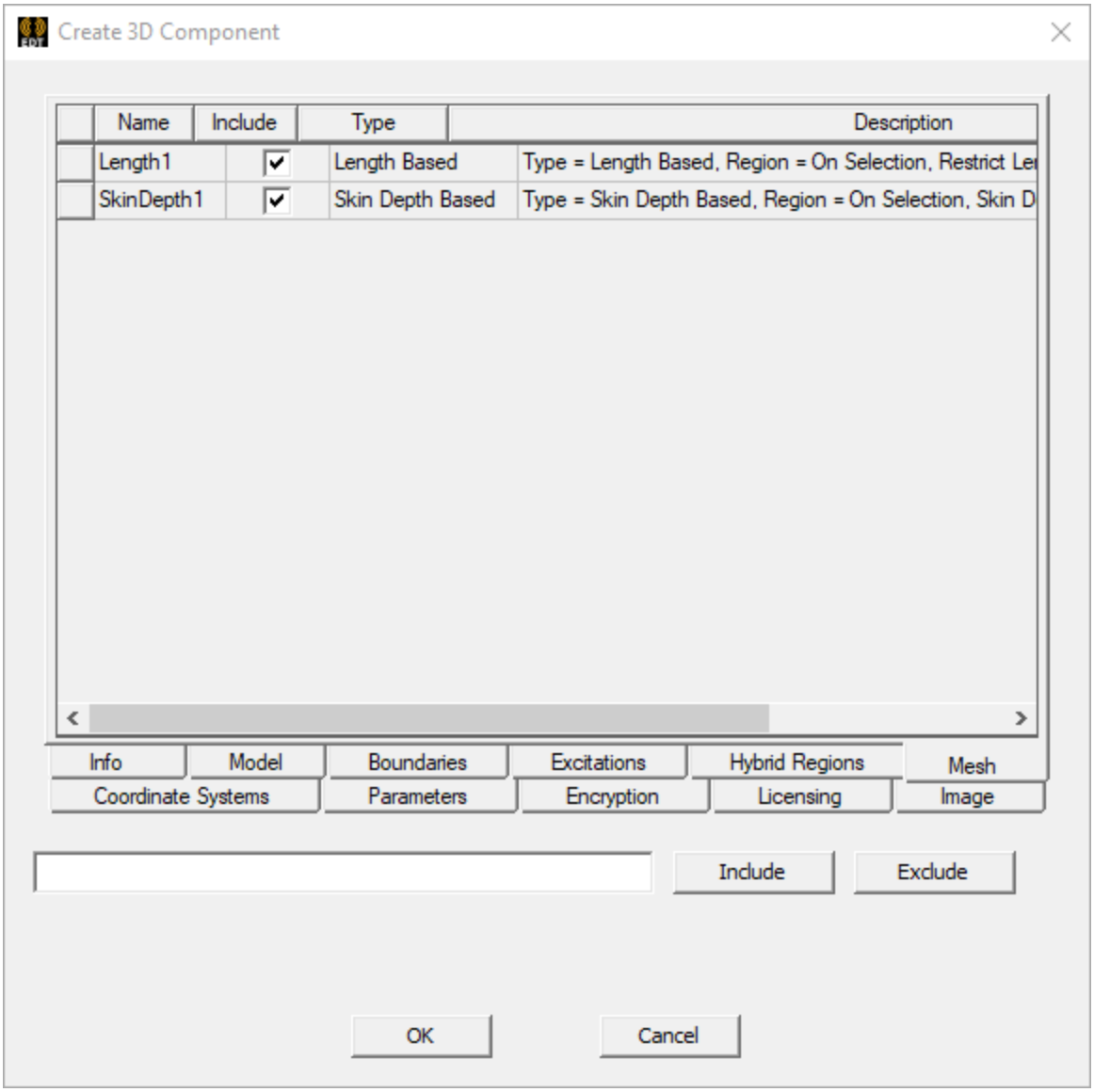
Create 3D Component Mesh Tab

The list is populated with all the mesh operations on selected objects.

- Use the **Include** check box to include mesh operations in the 3D component (selected), or to Exclude them from the component (cleared).

  Alternatively, you can type a mesh operation *Name* into the text box and click the **Include** or **Exclude** button. Wildcards (? and *) are supported when you use this method, so you can change the inclusion state of multiple mesh operations with similar names in a single operation.

- *Type* and *Description* columns are read only.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

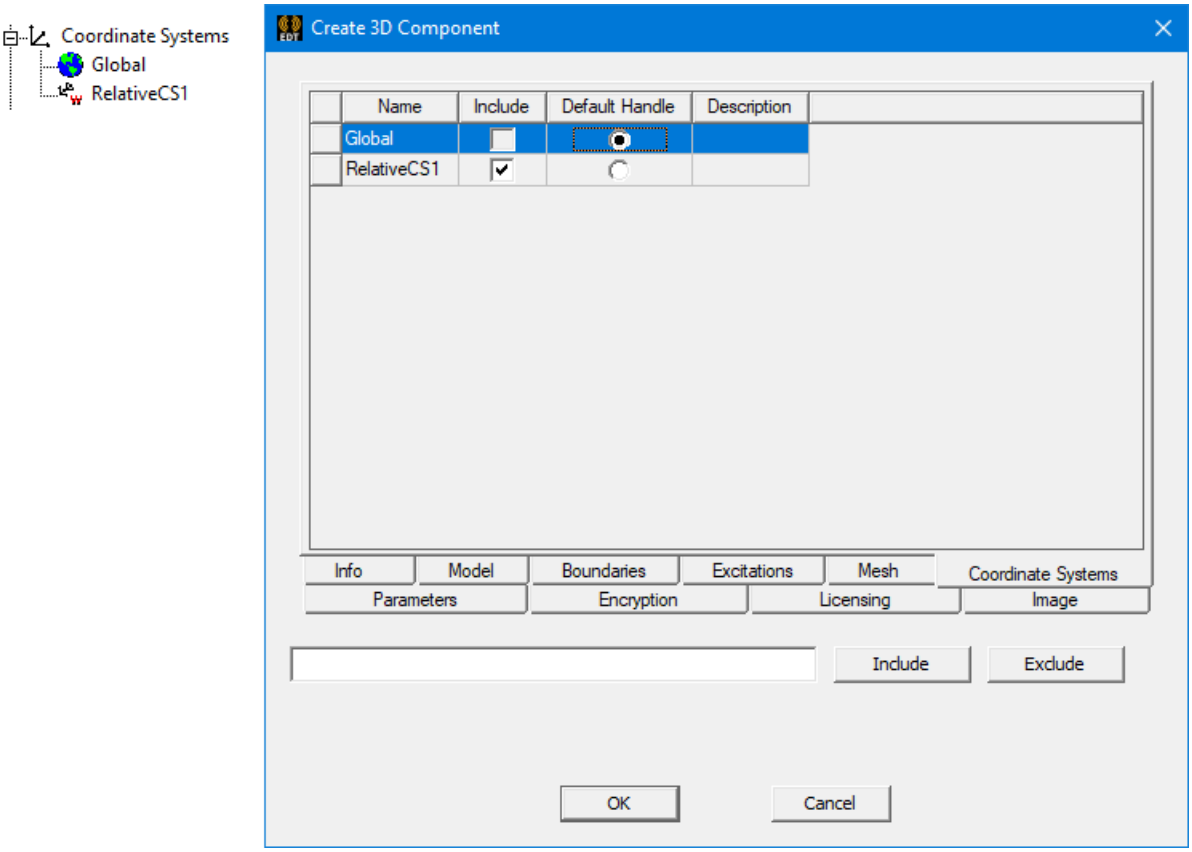
Create 3D Component Coordinate Systems Tab

The list is populated with all the Coordinate Systems on selected objects. By default, only the Coordinate Systems used to define the object orientations are included. You can include Coordinate Systems created on specific model parts.

- Use the **Include** check box to include coordinate systems in the 3D component (selected), or to Exclude them from the component (cleared).

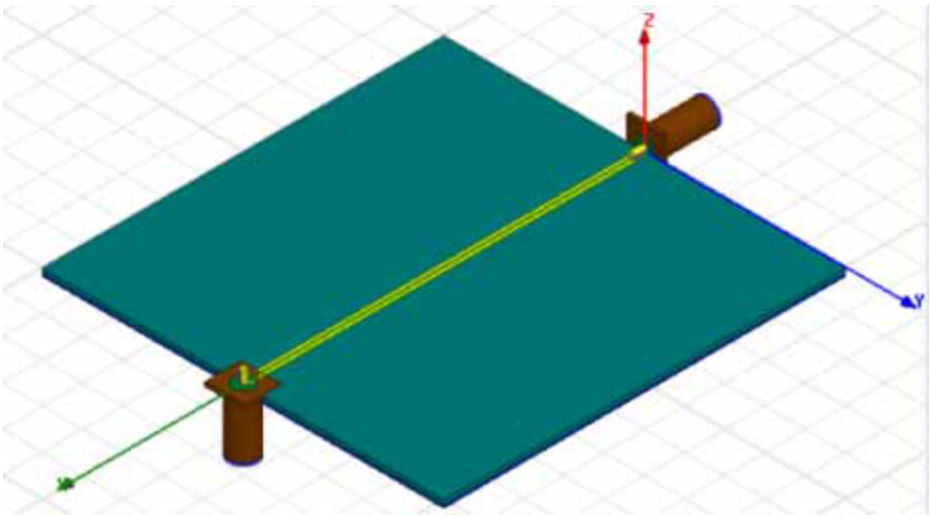
  Alternatively, you can type a coordinate system *Name* into the text box and click the **Include** or **Exclude** button. Wildcards (? and *) are supported when you use this method, so you can change the inclusion state of multiple coordinate systems with similar names in a single operation.

- You can select any of the CS as the default handle. When this component is placed in assembly design, the handle CS is aligned to target CS in assembly design. Note that the handle CS can be changed after import; see Editing 3D Component Properties for more details.
- By default, the current working CS is designated as the default handle CS.
- The default handle CS must be included in the model (except for global).



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The specified default handle affects the orientation of the component upon insertion:



Create 3D Component Parameters Tab

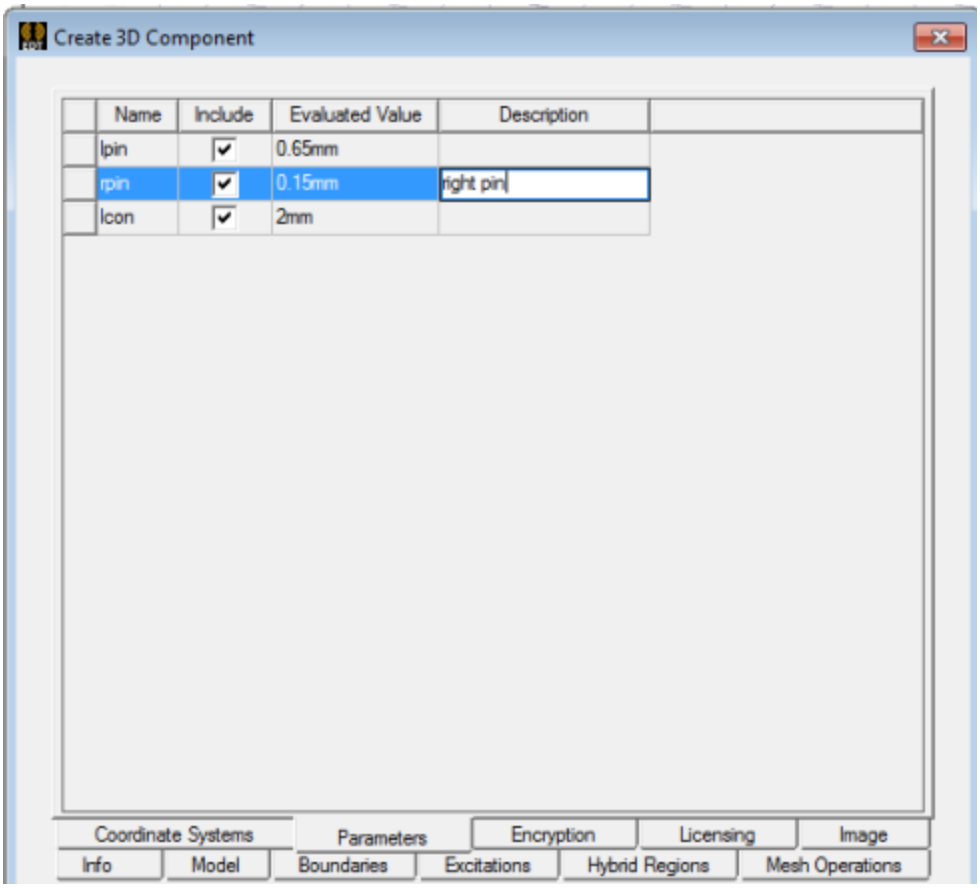
Component properties can be parameterized by assigning variables. The parameters list is populated with all the variables used by objects and coordinate systems, design data, and material properties included in the component.

- Use the **Include** check box to include parameters in the 3D component (selected), or to exclude them from the component (cleared).

  Alternatively, you can type a parameter **Name** into the text box and click the **Include** or **Exclude** button. Wildcards (? and *) are supported when you use this method, so you can change the inclusion state of multiple parameters with similar names in a single operation.

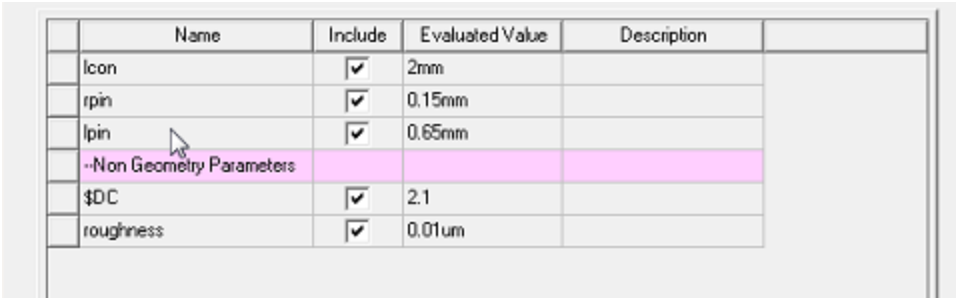
- You can fill in the **Description** field, if desired.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Non-Geometry Variables in 3D Components

If you create non-geometry variables materials or boundaries, these are listed separately:

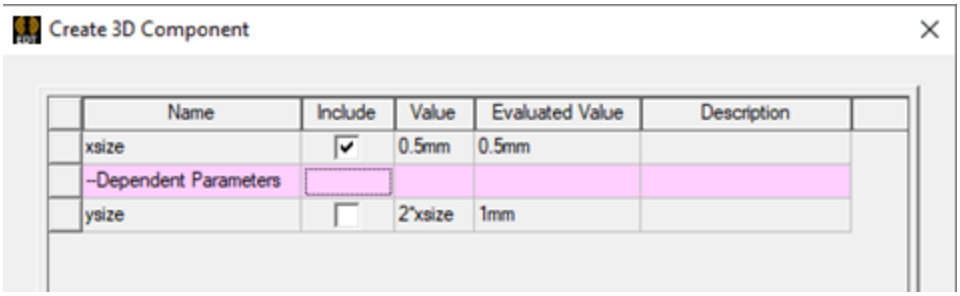


If you exclude any geometry, boundary, or material associated with a variable, the associated variable is automatically excluded on the **Parameters** tab.

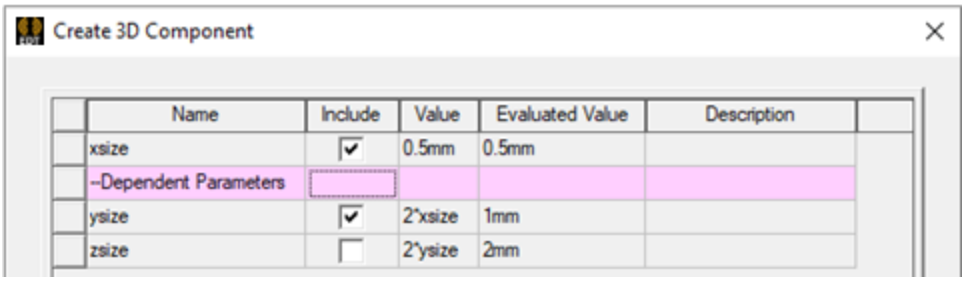
Dependent Variables in 3D Components

If you create dependent variables, you can choose to display the values.

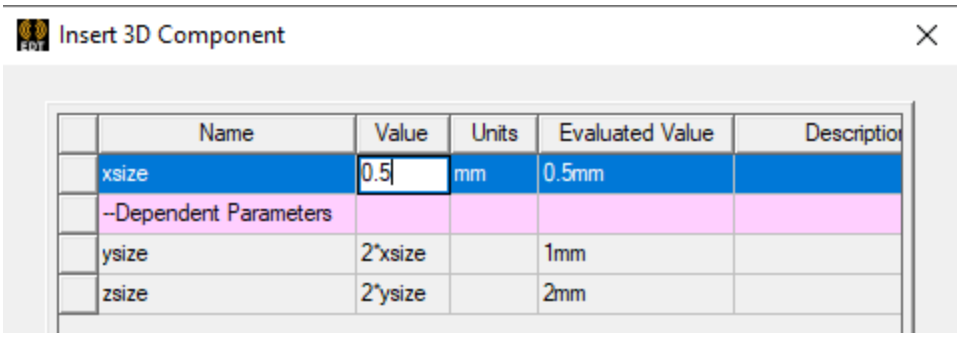
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If more variable dependency occurs for an Included variable, such variables are also displayed with the option to include.



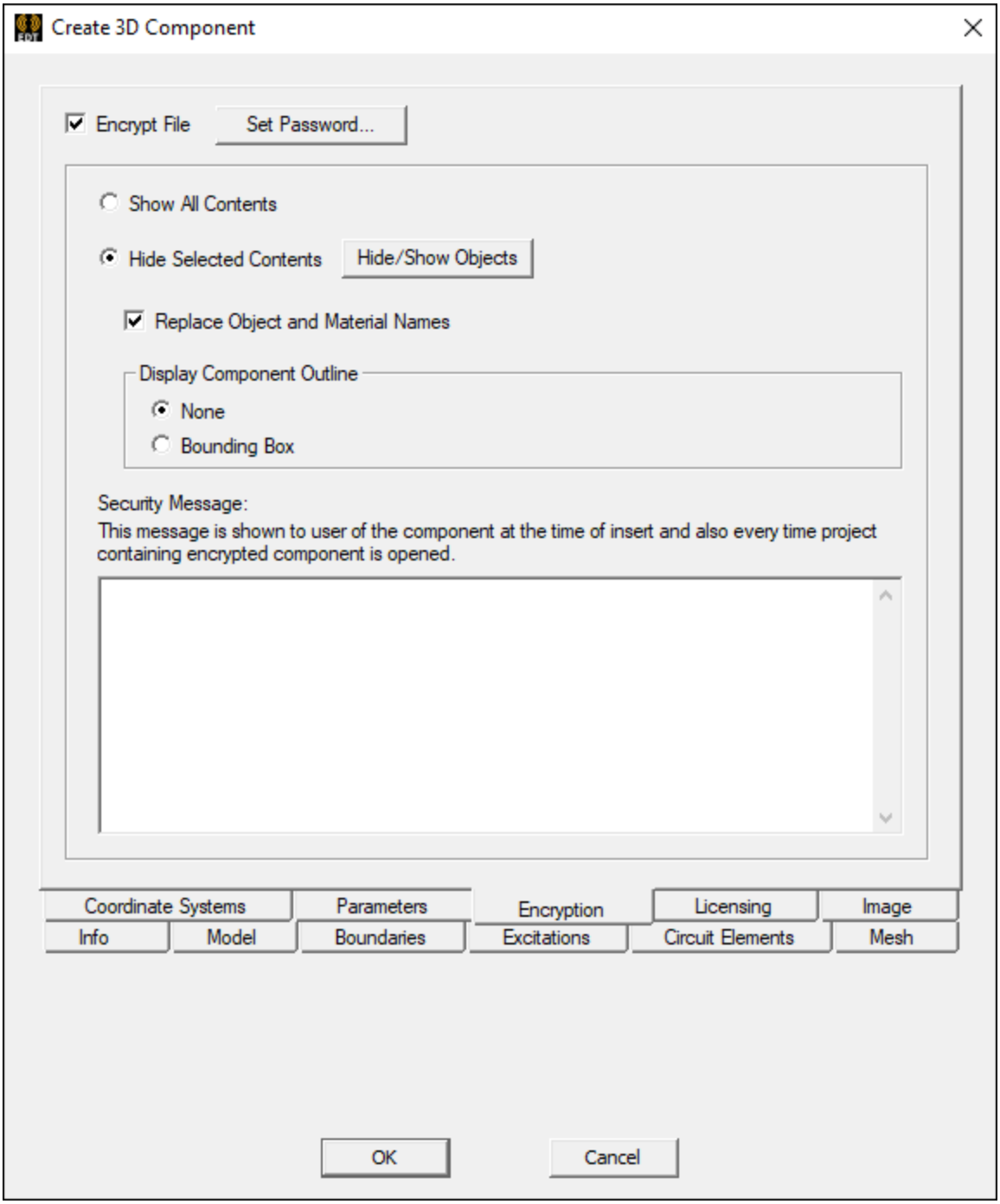
When you Insert a 3D Component with dependent Parameters included, you can edit the Independent variables, but can only view the evaluated values of the Dependent Parameters.



Create 3D Component Encryption Tab

The **Encryption** tab contains options for 3D component encryption.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Selecting the **Encrypt File** check box enables all options and automatically launches the **Set Passwords** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Encryption is a common technique for encoding information in a manner that is very difficult for unauthorized users to decode. When this option is enabled for 3D components, the contents of the component file are encrypted to protect them from external viewing or editing.

- 3D components use the Advanced Encryption Standard (AES) with a 256-bit key and password-based key derivation.
- If the encrypted component is created with an **Internal Key**, it can be used without additional authorization.
- If the encrypted component is created with a **User Password**, that password must be entered when the component is used in a design.
- It is not possible to edit the definition of an encrypted component within Electronics Desktop unless **Allow Component Edit with User Password** was selected when defining the component.
- If the **Show All Contents** option is selected, the geometry and other contents will be shown when the component is used. Geometry export and links will operate as for general geometry.
- The **Hide Selected Contents** option can be used to further protect access to an encrypted component's details. When this option is selected, hidden contents will not appear in the user interface.
  - All geometry will be hidden, except for objects that the user specifies as visible during component creation.
  - Hidden geometry will not be visible in the model window or included in the history tree.
  - Geometry export and links will not be allowed.
  - Materials, boundaries, and other setup data will also be hidden, with limited exceptions such as excitation settings that must be accessed in order to control the simulation and post-processing of the design.
  - For components with hidden contents, all geometry will be excluded from field calculations and from plots of fields and meshes. Plots in surrounding volumes will extend to, and can include, the outer surfaces of the component, but not the containing volume.

    > **Important:**
    >
    > Such plots may indirectly reveal component content by providing a view of the surfaces where the plots terminate. To prevent such exposure, include additional modeled objects that surround the hidden contents of the component and have material of the surrounding material environment (e.g., air, vacuum).

    See: Hiding Component Content.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

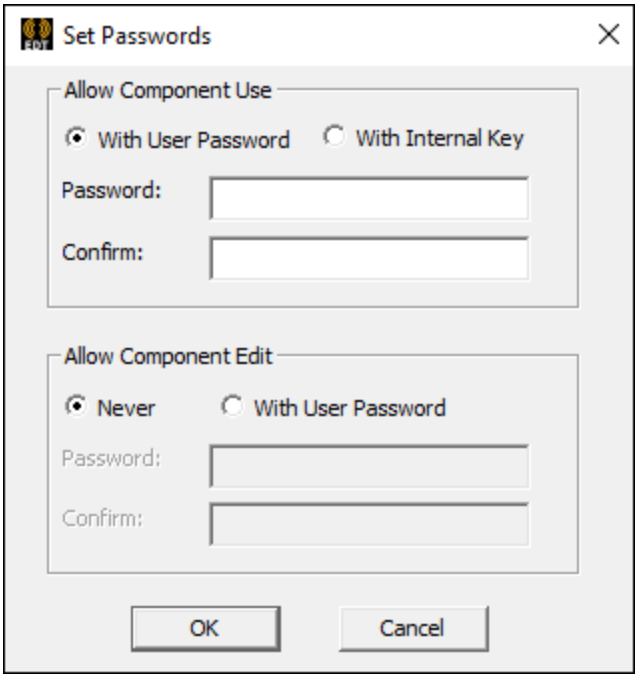
- When using **Hide Selected Contents**, the names of objects, materials, boundaries, and other setup data may appear in progress updates, messages, and other informational text. The **Replace Object and Material Names** option can be used to automatically convert object and material names to generic names in the exported component. Other sensitive names should be changed before exporting.
- The **Security Message** field specifies text that is displayed at component insertion and when the project containing the component is opened.
- When a project contains one or more encrypted 3D components, the project file is encrypted in order to protect its contents. This encryption uses an internal key, so that the file can be opened without entering a password.

> **Important:**
>
> There are some additional considerations with respect to projects using encrypted components:
>
> - The component itself will always be stored in encrypted form.
> - When an HFSS design contains an encrypted component with hidden contents, it will also protect solver inputs and other project files containing data that could be used to extract component information. However, note that solver inputs associated with SBR+ regions are not protected, even when encrypted components are in use.
> - For a hybrid design that contains a combination of SBR+, FEM, and IE regions: the FEM and IE regions will be fully protected, while the SBR+ regions will not.
> - Other design types, such as Maxwell, Q3D, or Icepak, do not include extended protection of solver inputs and other project files.

Set Passwords

Clicking **Set Password** launches a window containing fields for setting component passwords. This is the same window that automatically launches upon enabling **Encrypt File**.

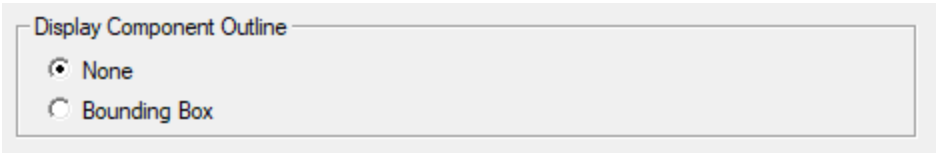
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Allow Component Use** – allows you to select between encrypting with a user password or an internal key (which requires no additional authorization). If setting a password, enter it in both the **Password** and **Confirm** fields and click **OK**.
- **Allow Component Edit** – allows you to choose between never allowing edits to components or allowing edit with a user password. If selecting the latter, enter a password in both the **Password** and **Confirm** fields and **click** OK.
- When enabling a password for both component use and component edit, the password does not have to be the same.

Hiding Component Content

The **Hide Selected Contents** option allows selected parts of the component to be displayed either as a **Bounding Box** or without an outline (**None**).

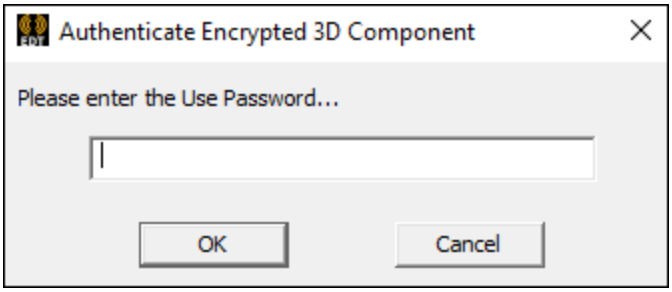


Once encryption is set, visibility can be set on a per-object basis using the **Model** tab of the **Create 3D Component** window. Preview the component using the**Image** tab.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

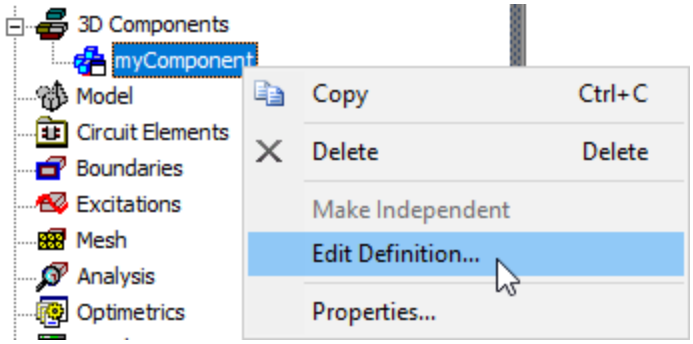
Inserting Encrypted Components

When an encrypted 3D component is added to a design, a prompt will appear asking for the password.

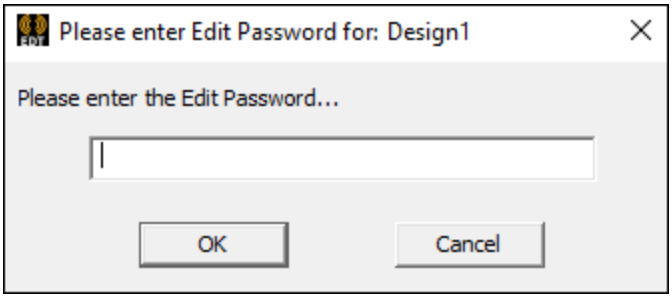


Editing Encrypted Components

3D component definitions can be edited from the **Project Manager**:

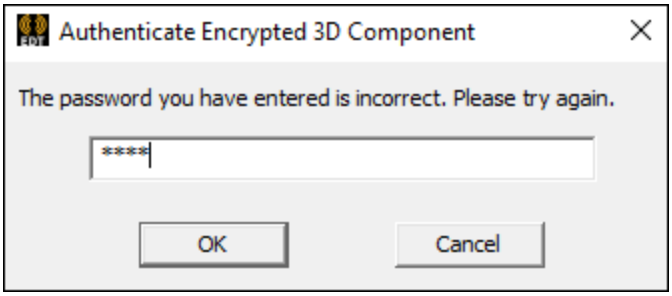


A prompt appears asking for the password:

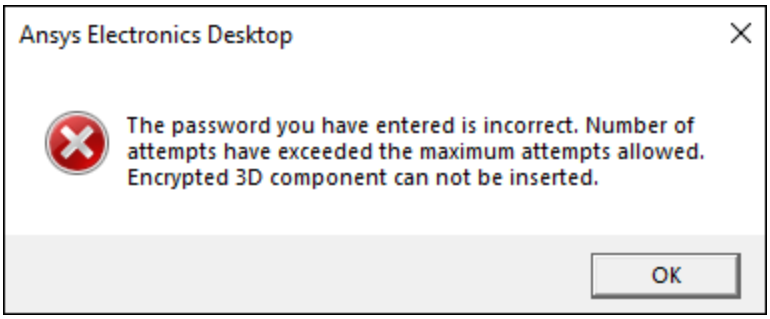


If the password is incorrect, a warning appears:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

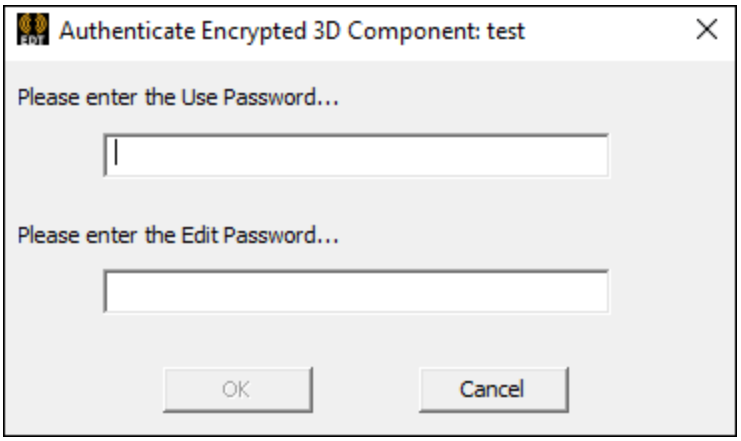


There is a three attempt maximum, after which a warning will appear and no further attempts may be made:



Opening a 3D Component File

When opening an encrypted 3D component file using **File** > **Open**, both the use password and edit password must be entered:



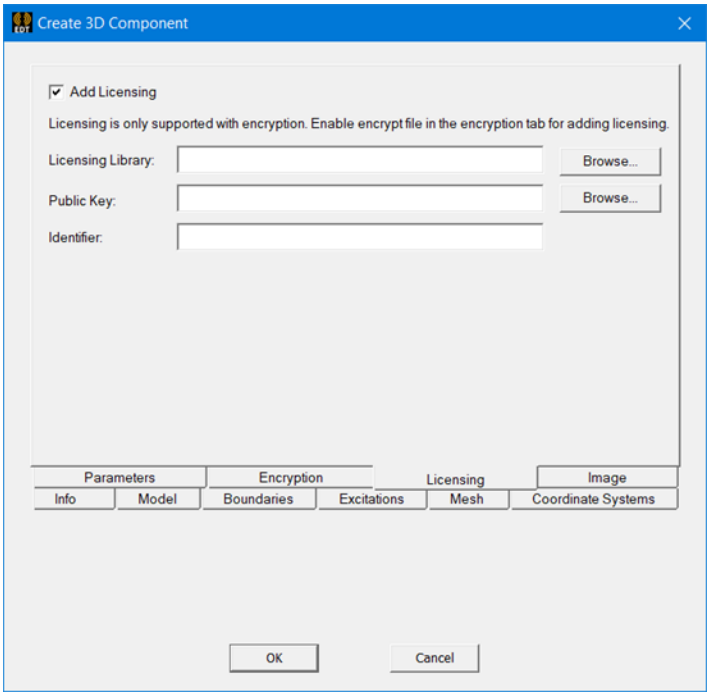
Copying and Pasting Encrypted Components

Encrypted components may be Copied and Pasted. They will be remain encrypted.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create 3D Component Licensing Tab

A 3D Component can be licensed using an external licensing library. This feature is for those who want to license and distribute their 3D Component models. On the **Licensing** tab, you specify a 3D Component Licensing dll, a public key and an optional component identifier. You can license a 3D Component only if it is also encrypted.

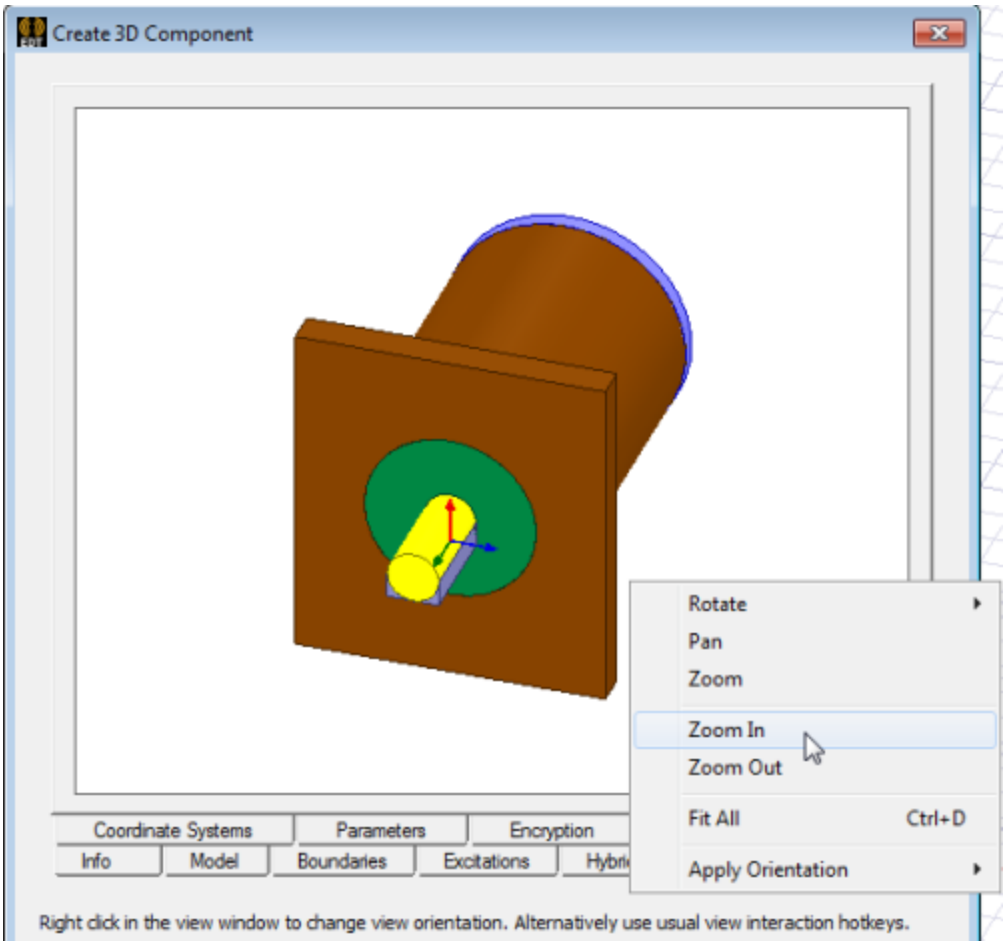


Create 3D Component Image Tab

The **Image** tab shows the preview image for the component, based on the reference Coordinate System. The image also responds when you **Include** or **Exclude** objects from the **Model** tab.

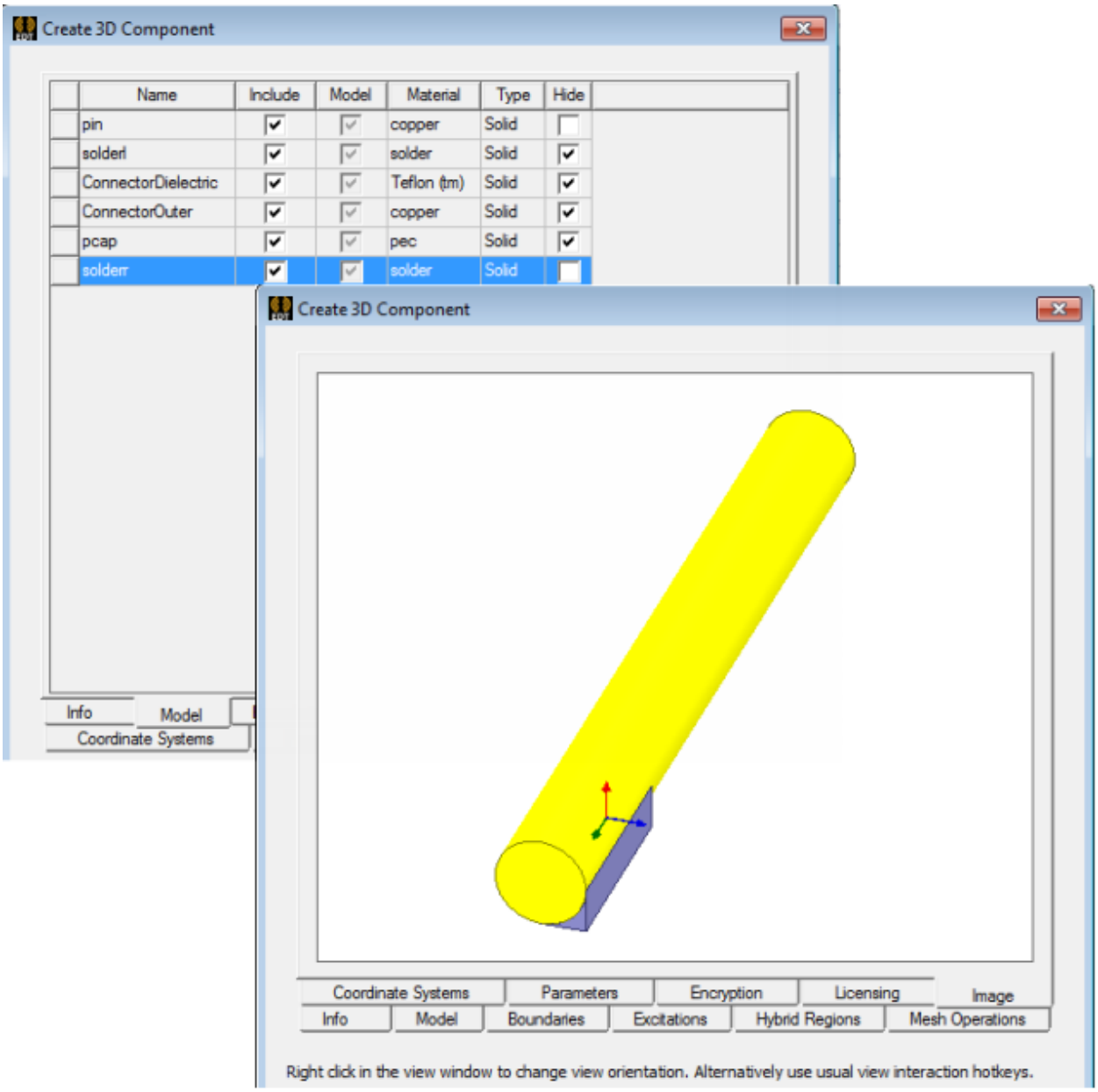
- You can right-click for a shortcut menu to change the view of the preview image.
- Changing the preview orientation does not affect the modeler window view.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



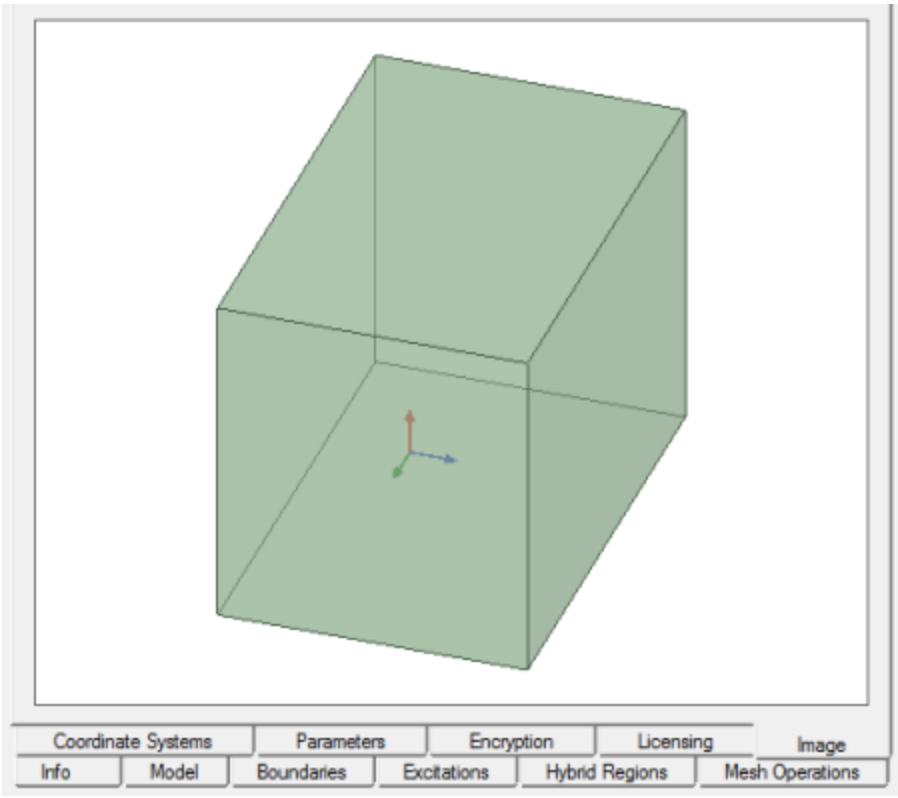
If you use the **Encryption** tab and **Model** tab settings to hide/show the model or specified model objects, the **Image** tab responds. You must use encryption to enable the Hide check boxes on the **Model** tab.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you use the **Encryption** tab to display only an object outline, the **Image** tab responds.

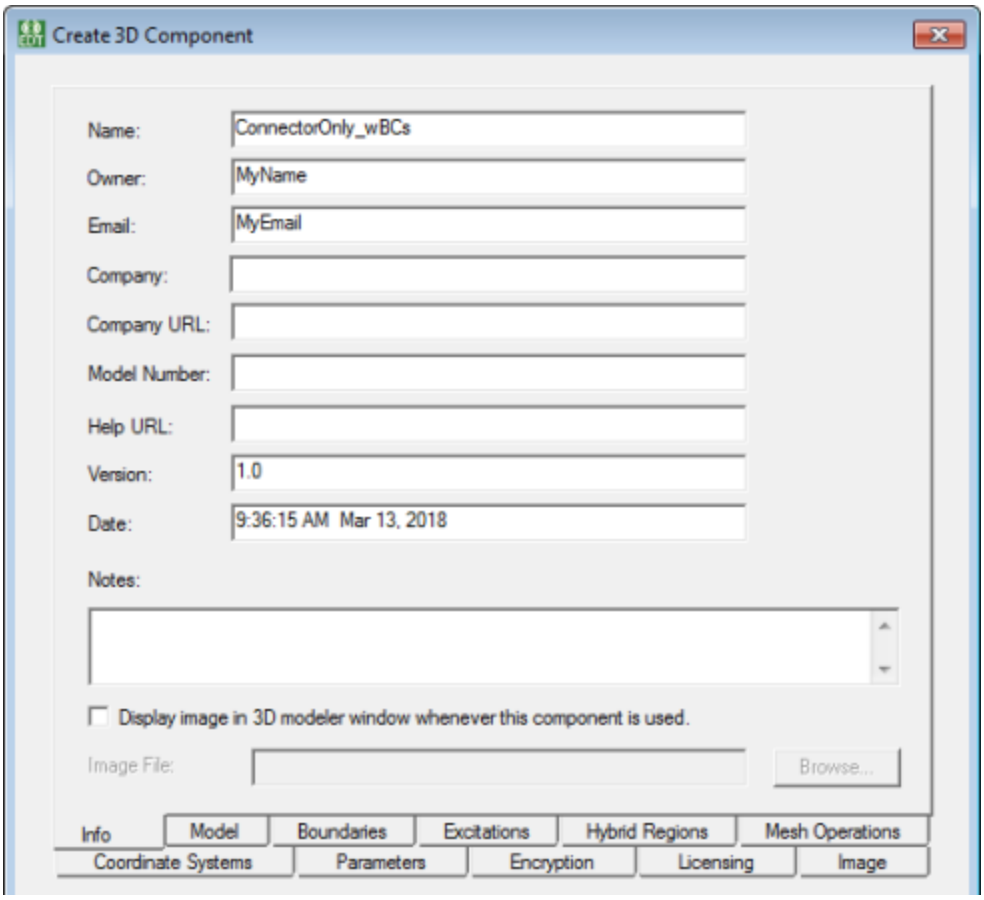
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



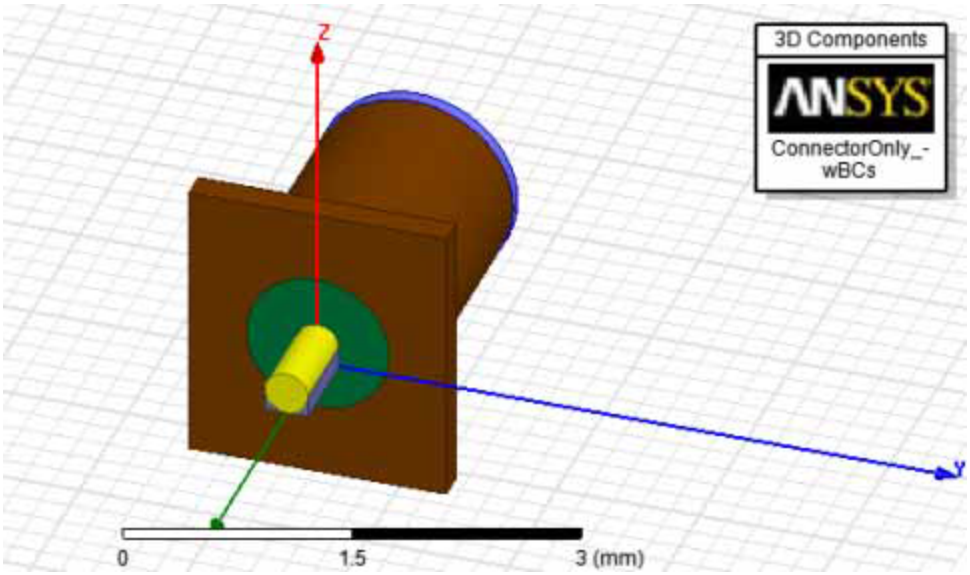
Create 3D Component Info Tab

On the **Info** tab you can view and edit component information, as well as add notes. The *Name* in the first field is used when the component is inserted into a design. Other fields include *Owner, email, Company, Company URL, Model Number, Help URL version, Date,* and *Notes*.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

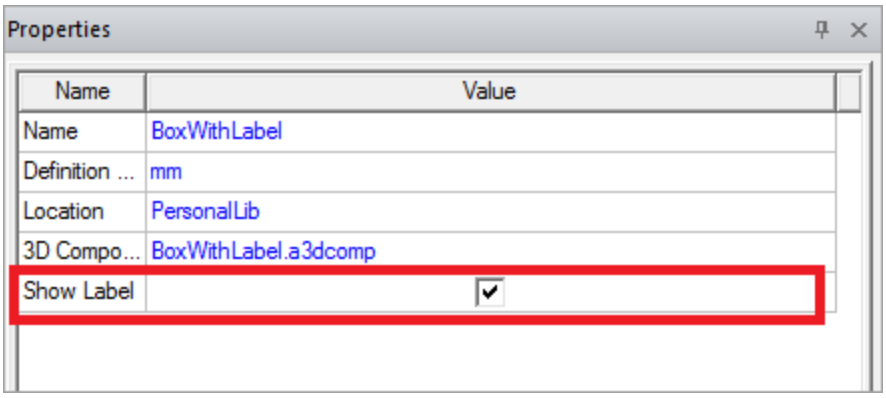


You can also included a .bmp format image file to display in the upper right area of the 3D modeler whenever the component is used. The image file typically contains a company logo. Check the **Display image** box to enable the **Image File** field, and click **Browse** to open a window to navigate your file system select the image.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

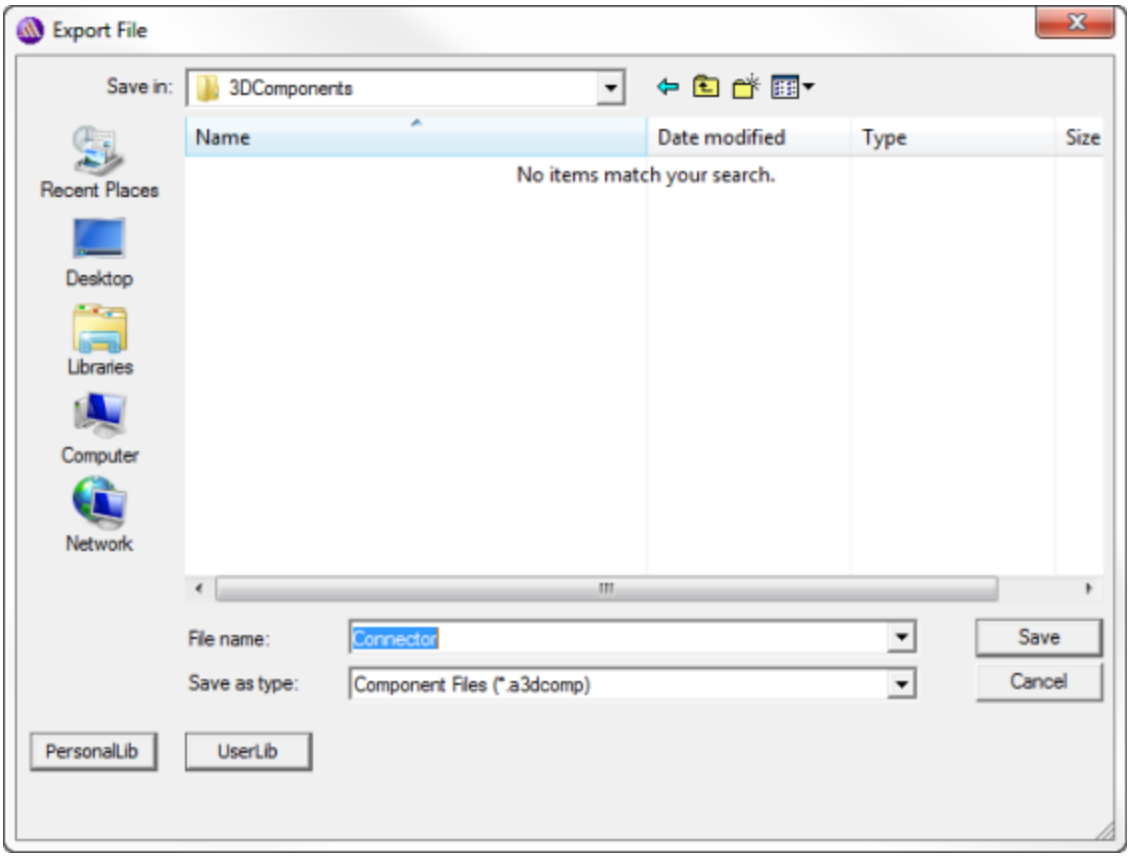
You can move the image in the modeling workspace by clicking the "3D Components" area of the box and dragging it. You can hide the image by editing the 3D component and clearing the **Display image...** check box, or by deselecting the Show Label check box in the Properties window:



Save 3D Component File

When you **OK** the *Create 3D Component* dialog box, an *Export File* dialog box displays. By default the **Save in** field shows the model source folder. Click **PersonalLib** or **UserLib** to display a 3D Components folder.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



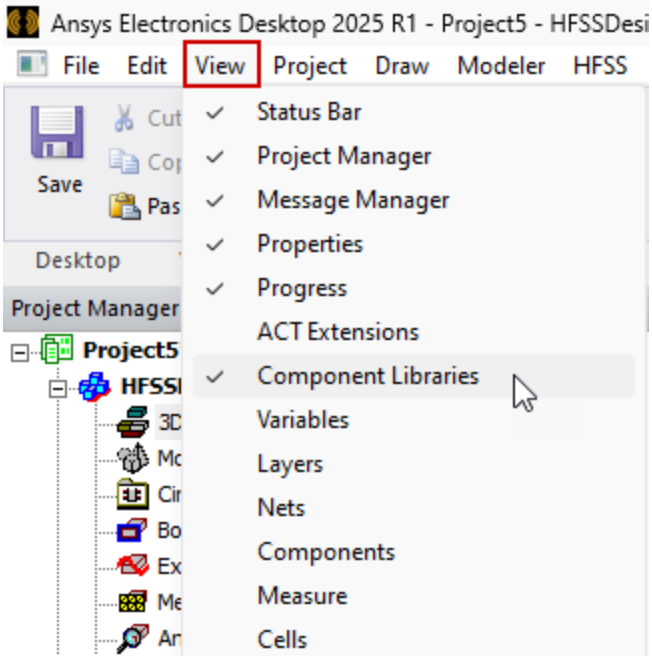
The default **File name** is the component *Name* specified in the *Info* tab.

Using the Component Libraries Window

Once you save one or more components to a library, and create target coordinate systems in the design where you intend to place the component, you can right-click the 3D component icon ( ) in the Project Manager or, using the menu bar, click one of the **Draw** > **3D Component Library** commands to browse your folders or libraries.

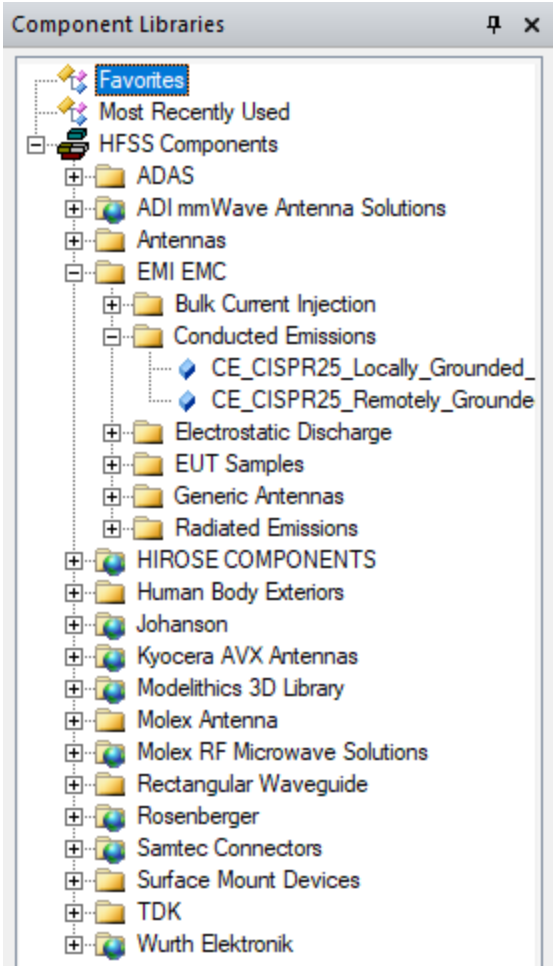
Toggle the visibility of the docked **Component Libraries** window from the menu bar by clicking **View** > **Component Libraries** and navigate the installed libraries from there. Visibility of the **Component Libraries** window is remembered for each design type so that, if context switches between different design types, the window is shown or hidden based on setting for new design type.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



From the **Component Libraries** window, you can navigate the directory of installed components, as well as any in your Personal Library. The **Component Libraries** window lists all the 3D components available in syslib, UserLib and PersonalLib. It shows a folder corresponding to a library only if there are components for the active design type in that library folder. There are also additional optional folders for managing Favorites and Most Recently Used components. **Component Libraries** window contents, including Most Recently Used and Favorites correspond to the active design. When context is switched between different designs, the *Component Libraries* window will be refreshed so that it is relevant to the active design.

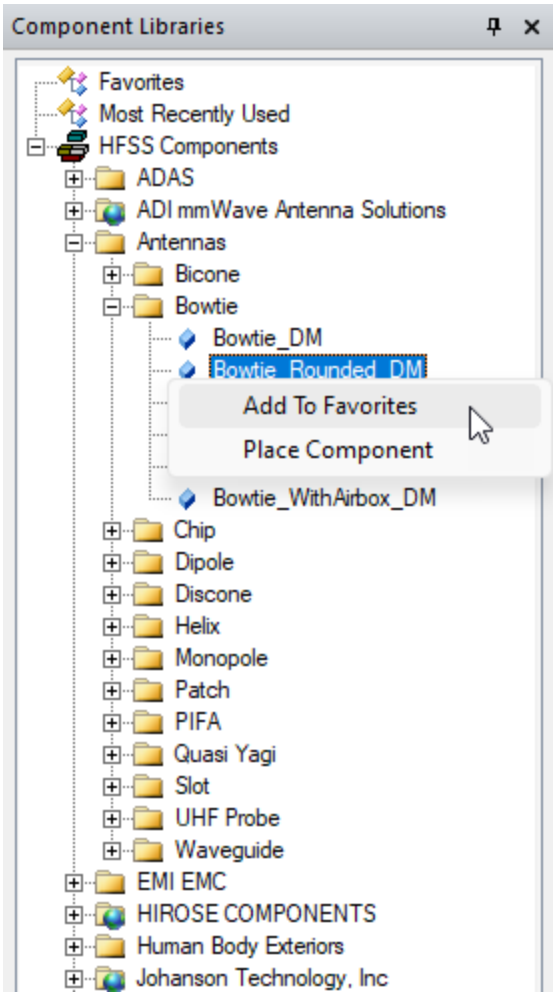
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can control whether to show the Favorites and Most Recently Used folders. Access the *Options* dialog box from the menu bar by clicking **Tools** > **Options** > **General Options**. The folder visibility options are on the General > Component Libraries Options page.

Right-click a component to view the **Add to Favorites** command for including frequently used components in the Favorites folder. Once a model is included in the Favorites folder, you can also right-click it to **Remove from Favorites**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When you have added a component to a design, it also appears in the **Most Recently Used** folder. You can also right-click model there to view the **Add to Favorites** and **Place Component** commands.

You can select any component from the library, and drag and drop to insert it in the design. If you insert a component by the drag and drop method, the component is immediately inserted, skipping the **Insert Component** dialog box. The component is inserted with default component parameter values and is placed in the currently active coordinate system.

You can also select **Place Component** from the shortcut menu or double-click any model in the component libraries to view the *Insert 3D Component Instance* dialog box opened to the **Parameters** tab. Using the **Place Component** menu command allows you to review component parameter values, change target coordinate system, and so forth, using the *Insert Component* dialog box. You can also view the **Image** and **Info** tabs.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component file organization on the disk

3D Component files are organized in three libraries (that is, *syslib, UserLib,* and *PersonalLib*). 3D component files must be in a **3DComponents** folder under the appropriate library location. Files must be further organized by design type in the **3DComponents** folder. For example, component files for Maxwell should be organized under 3DComponents\Maxwell, as done for syslib. The 3D Component Libraries window looks for components under a folder corresponding to the active design type, and it ignores component files under folders corresponding to another design type. For example, Maxwell will ignore component files placed directly under the HFSS subfolder.

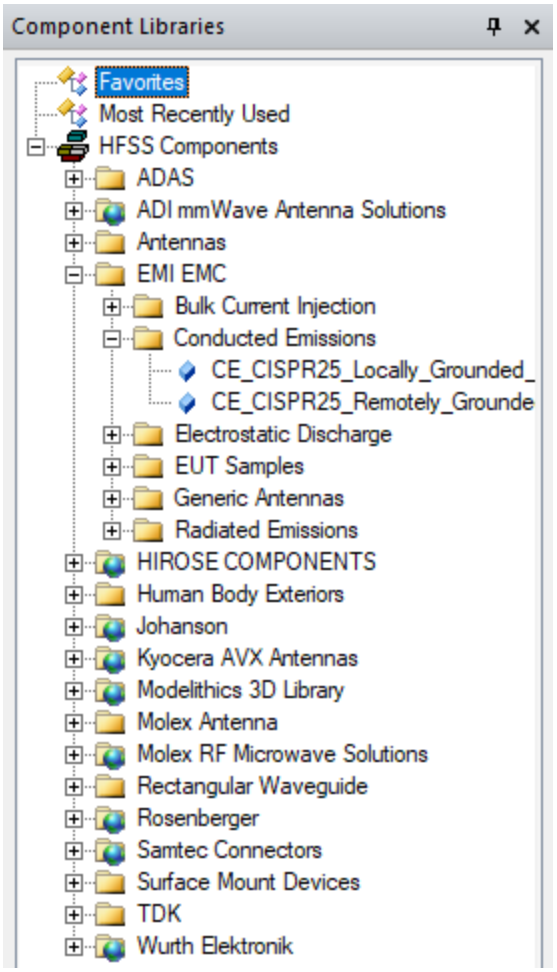
Inserting a 3D Component into a Design

Once you save one or more components to a library and create target coordinate systems in the design where you intend to place the component, you can right-click the 3D Component icon in the Project Manager or use the **Draw** > **3D Component Library** commands to browse your folders or libraries, or use the **View** > **Component Libraries** to display a Component Libraries window to navigate installed libraries. Alternatively, you can insert an instance of a component by right-clicking the 3D Component icon in the Project Manager and selecting **Insert Instance**.

From the Component Libraries window, you can navigate to the directory of installed components, as well as any in your Personal Library. The hierarchical tree display includes

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

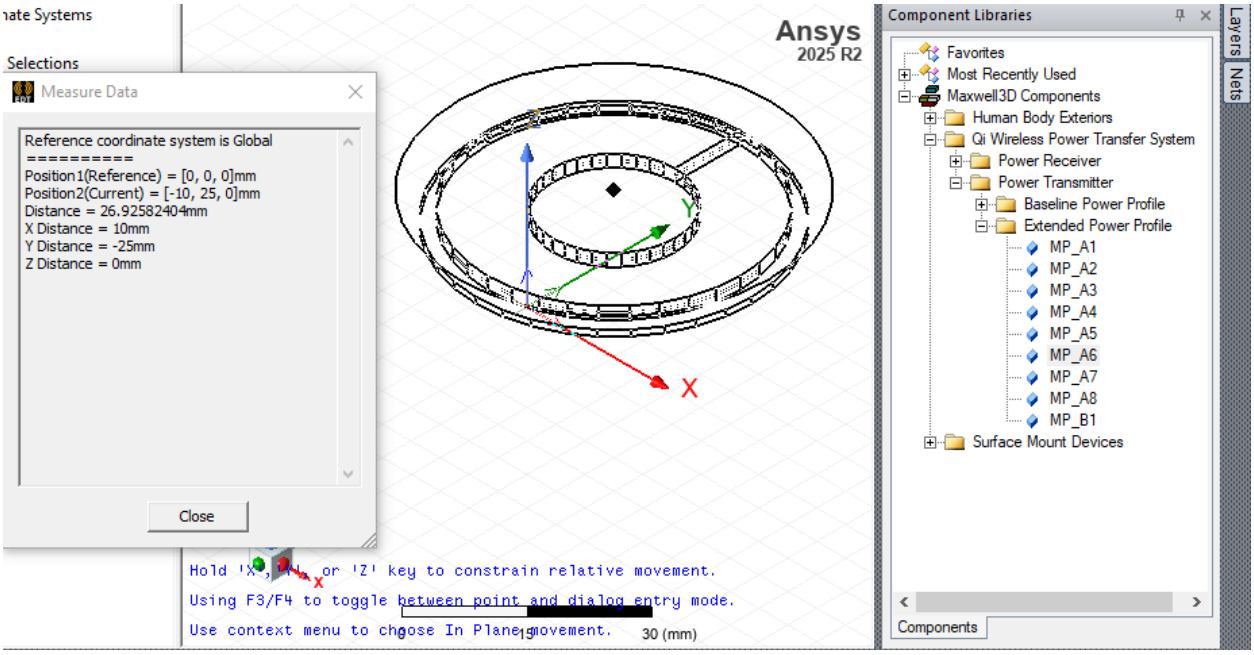
Favorites and **Most Recently Used** branches.



You can select any component from the library, and drag and drop to insert it in the design.

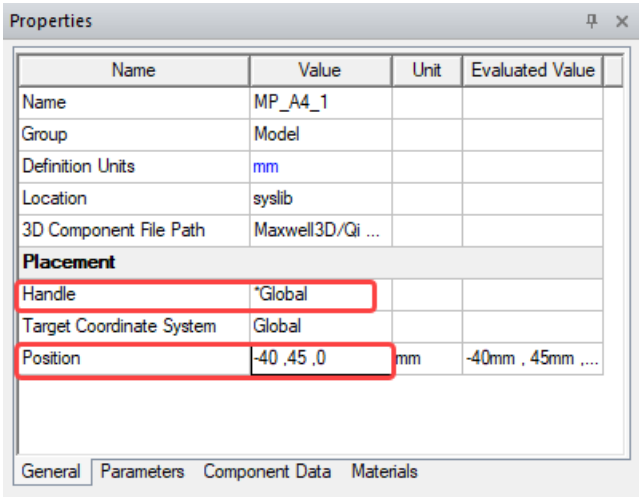
When you drag the component into the Modeler window or use the Browse function to select and place a 3D component, it appears as an outline to make it easier to place. The diamond-shaped marker designates the placement location.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Move the cursor in 3D modeler window to snap to a point where you want to insert the component and then click to place. While the component is being moved, the Measured Data window appears and reports the position in the Reference and Current coordinate systems.

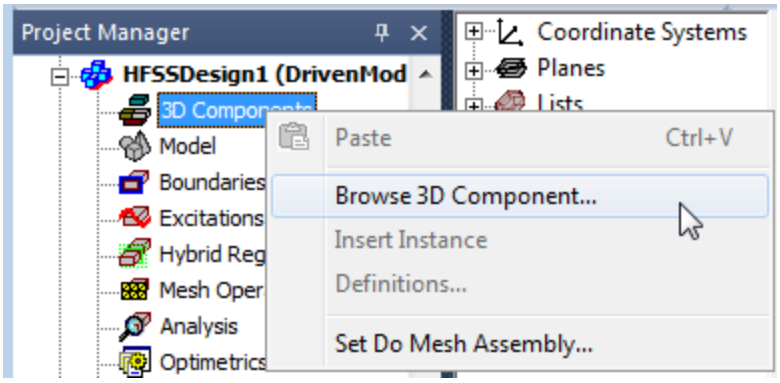
The position of the 3D component will be reported in the Properties window. It can also be edited from this window.



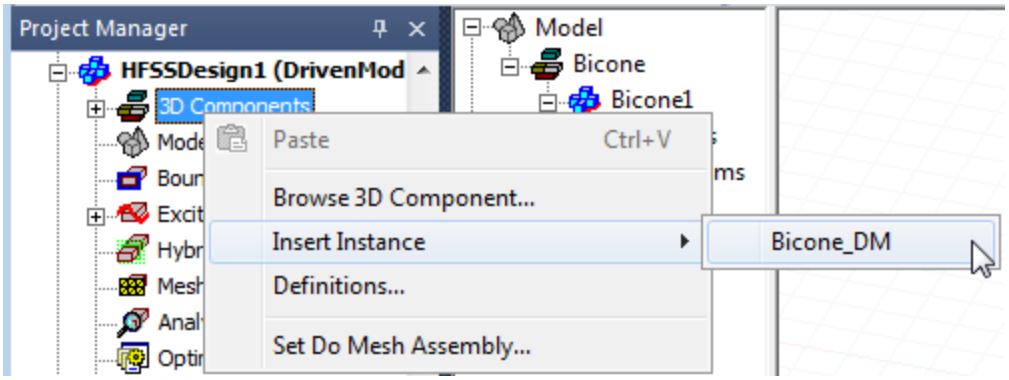
Double-click any model in the component libraries to view the **Insert 3D Component Instance** dialog box opened to the **Parameters** tab. You can also view the **Image** and **Info** tabs, and select the target coordinate system.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

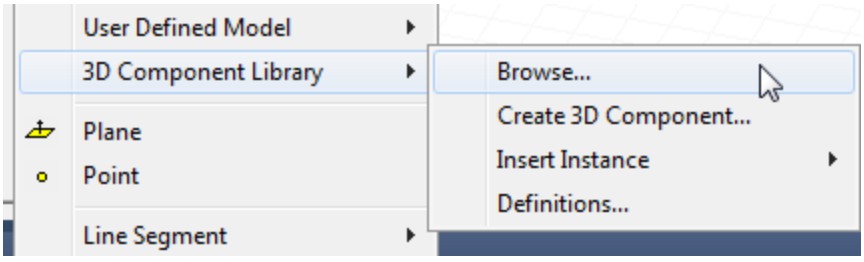
Selecting **Browse 3D Components** lets you navigate directories via a browser window.



If you have previously inserted a 3D Component, the **Insert Instance** command is enabled on the menu. You can use this command to select from 3D Models in the design to easily insert another instance.

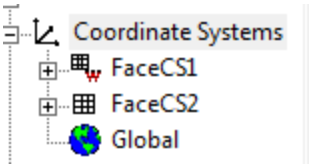


You can also click **Draw** > **3D Component Library** to access the **Browse** and **Insert Instance** menus **Antennas, Rectangular Waveguides**, and **PersonalLib** to select a component to insert into a design.

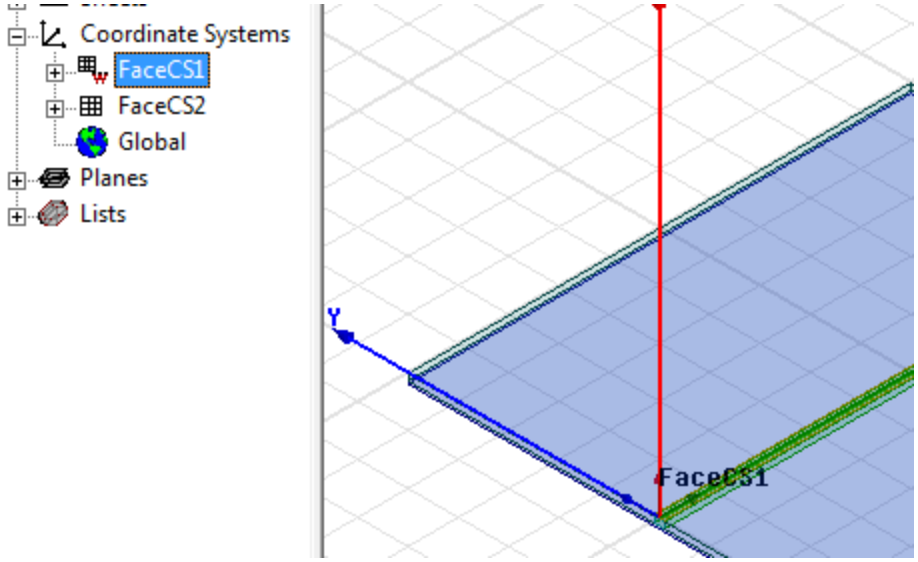


A target coordinate system provides a location for a component. For example, a design includes two additional coordinate systems:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

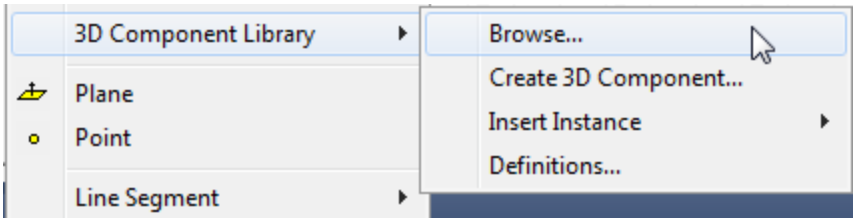


Selection of a coordinate system in the History tree displays a potential location for inserting a component.



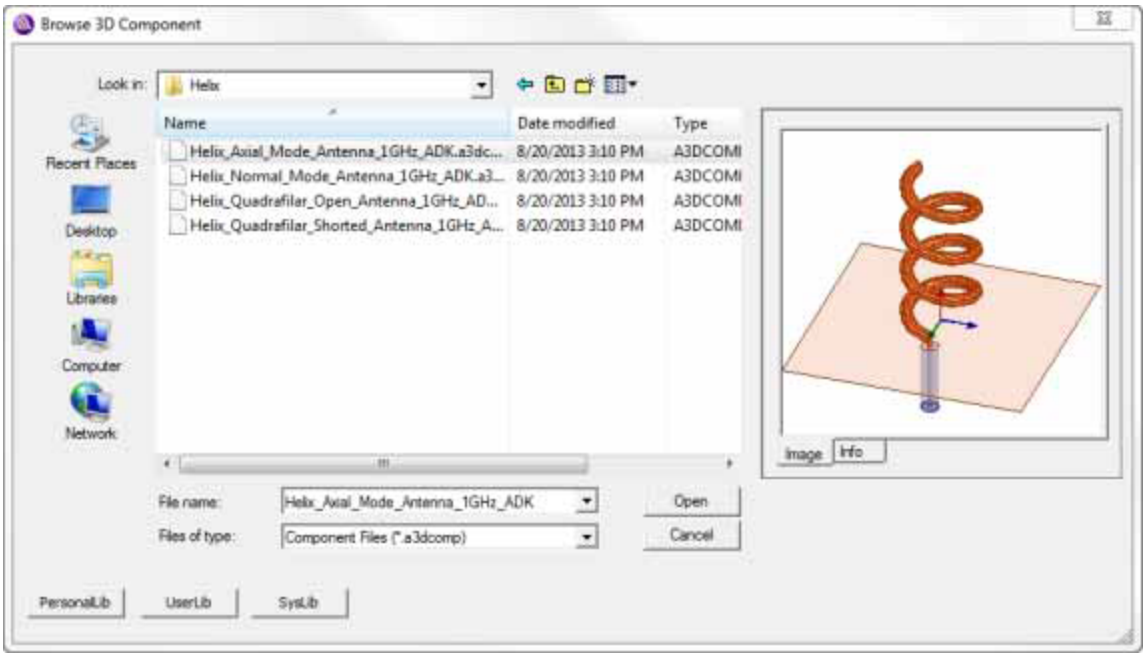
To Insert a 3D Component from the Menus

1. Right-click the **3D Component** icon in the **Project tree** to open shortcut menu. You can select **Browse** to use a browser window navigate the file system to the component.

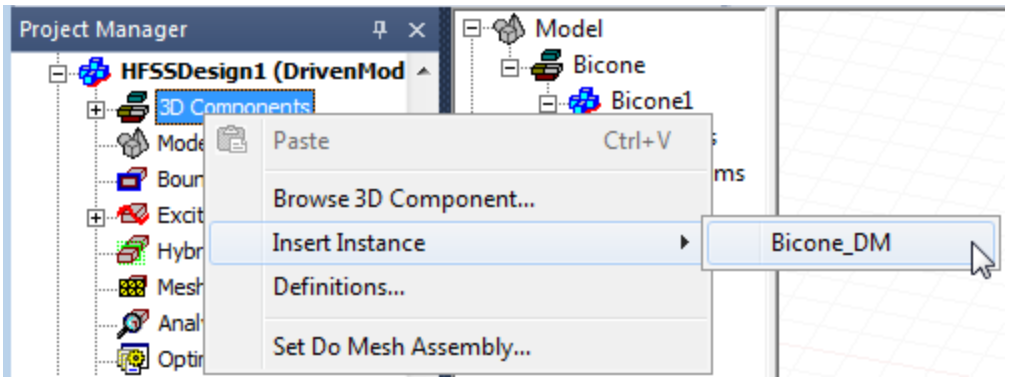


Selecting a 3D component file causes a display of the component image and the File name.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

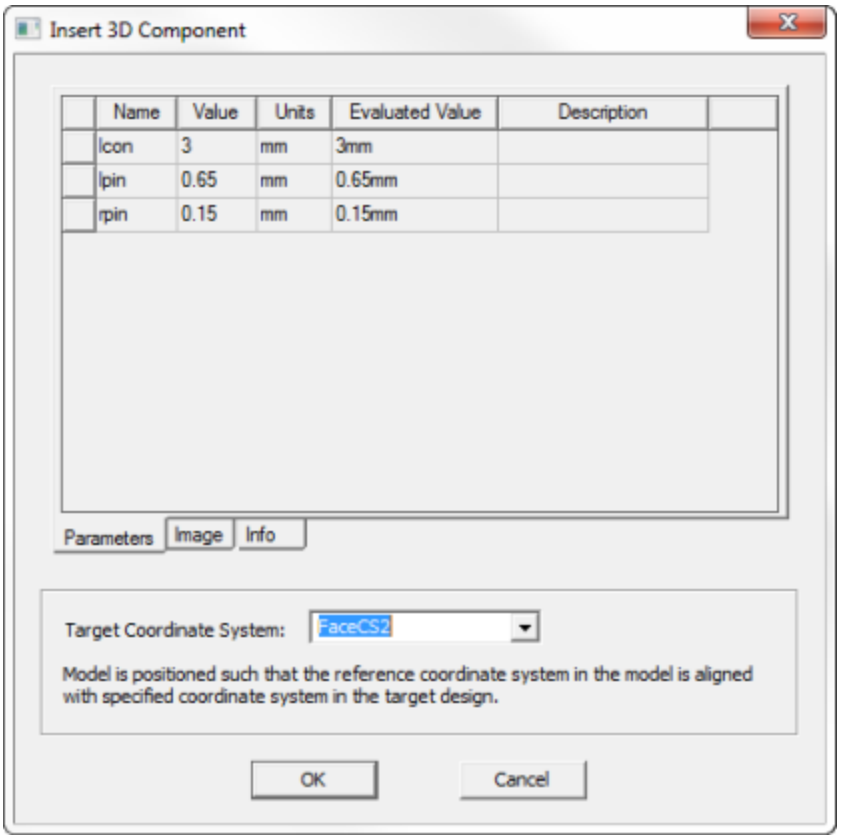


If you have previously inserted a component into a Project, the shortcut menu for the 3D Component displays that component so that you can easily insert another instance of the same component.



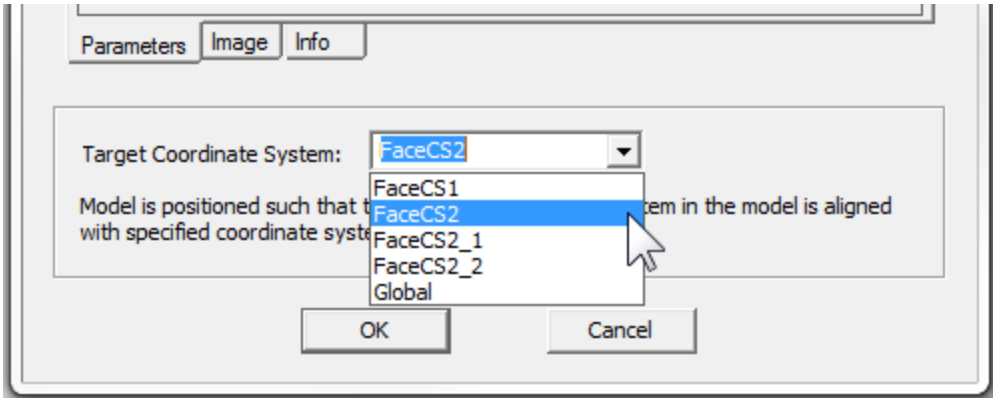
You see the **Insert 3D Component** dialog opened on the Parameters tab.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



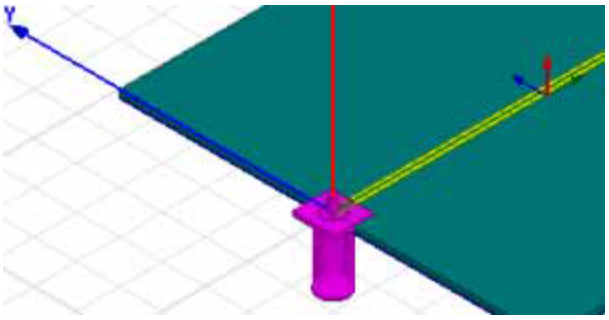
Tabs let you view the Parameters, Image, and Info. You can edit parameter values, and assign variables or expressions for parameters

2. Use the menu to select the Target Coordinate System if any have been defined in addition to the Global coordinate system. The target coordinate system that you select is highlighted in the modeler window.

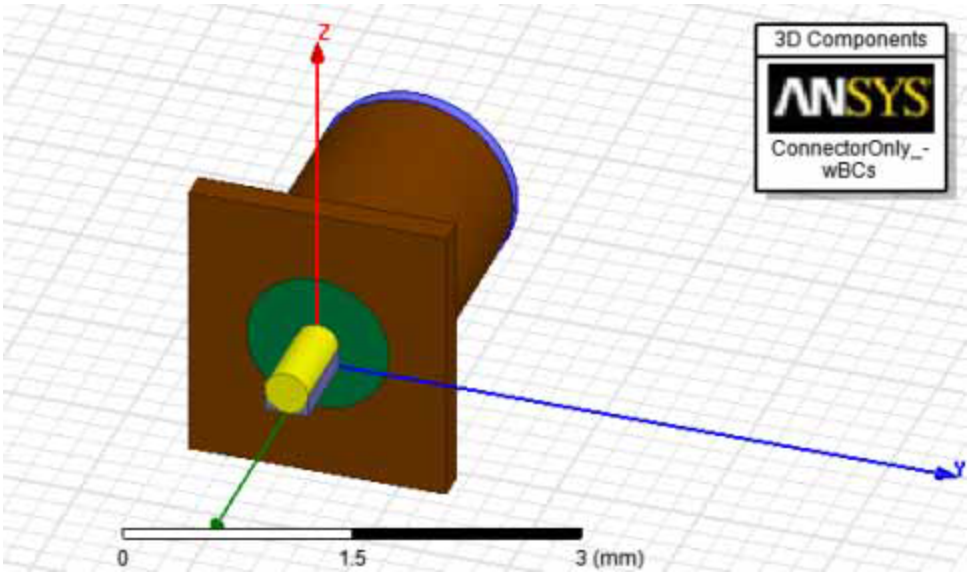


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. When you click **OK**, the outline of the component appears in the Modeler window, and you can then drag it to the desired location and click. The component is placed at the coordinate system you selected.

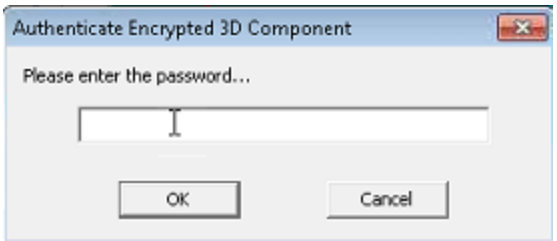


4. If the component has a logo defined, the image is always displayed in the upper right of the modeler window.



5. If you have used Browse, select a component and click **Open**. If you use the Insert Instance menu, select the component name.

If the component has been encrypted, you may be prompted to enter a password. If you insert another instance of the same component, you do not need to enter the password again for that component.



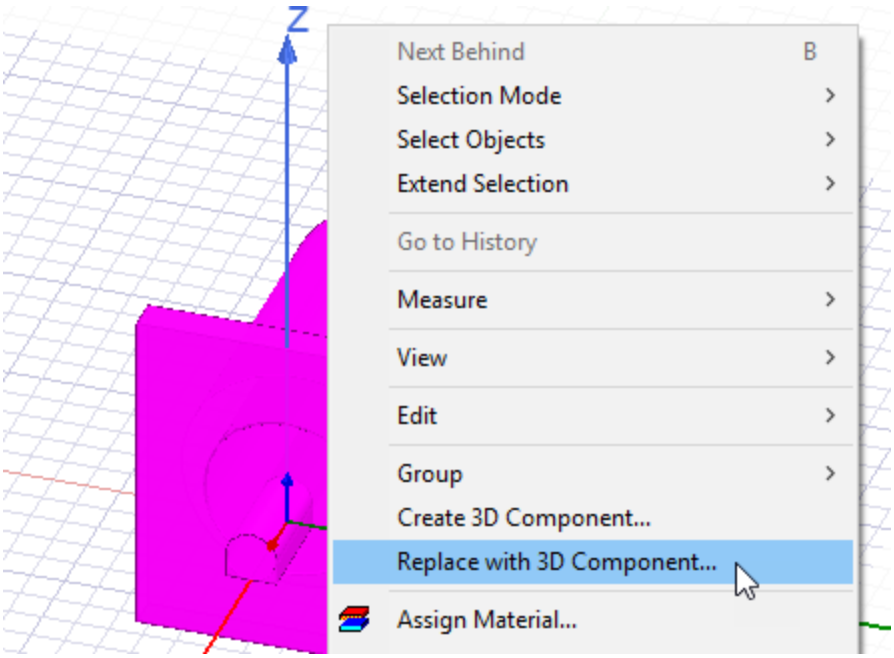
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You have three tries to enter the correct password. Upon successfully entering the password, the Insert 3D Component dialog opens with the Encryption tab displayed.

> **Note:** There are some additional considerations with respect to projects using encrypted components. When a project contains encrypted components, the project file and other related files are also encrypted. Encrypted component contents may also be hidden from view in the design where the component is used. For more details, see Create 3D Component Encryption Tab.

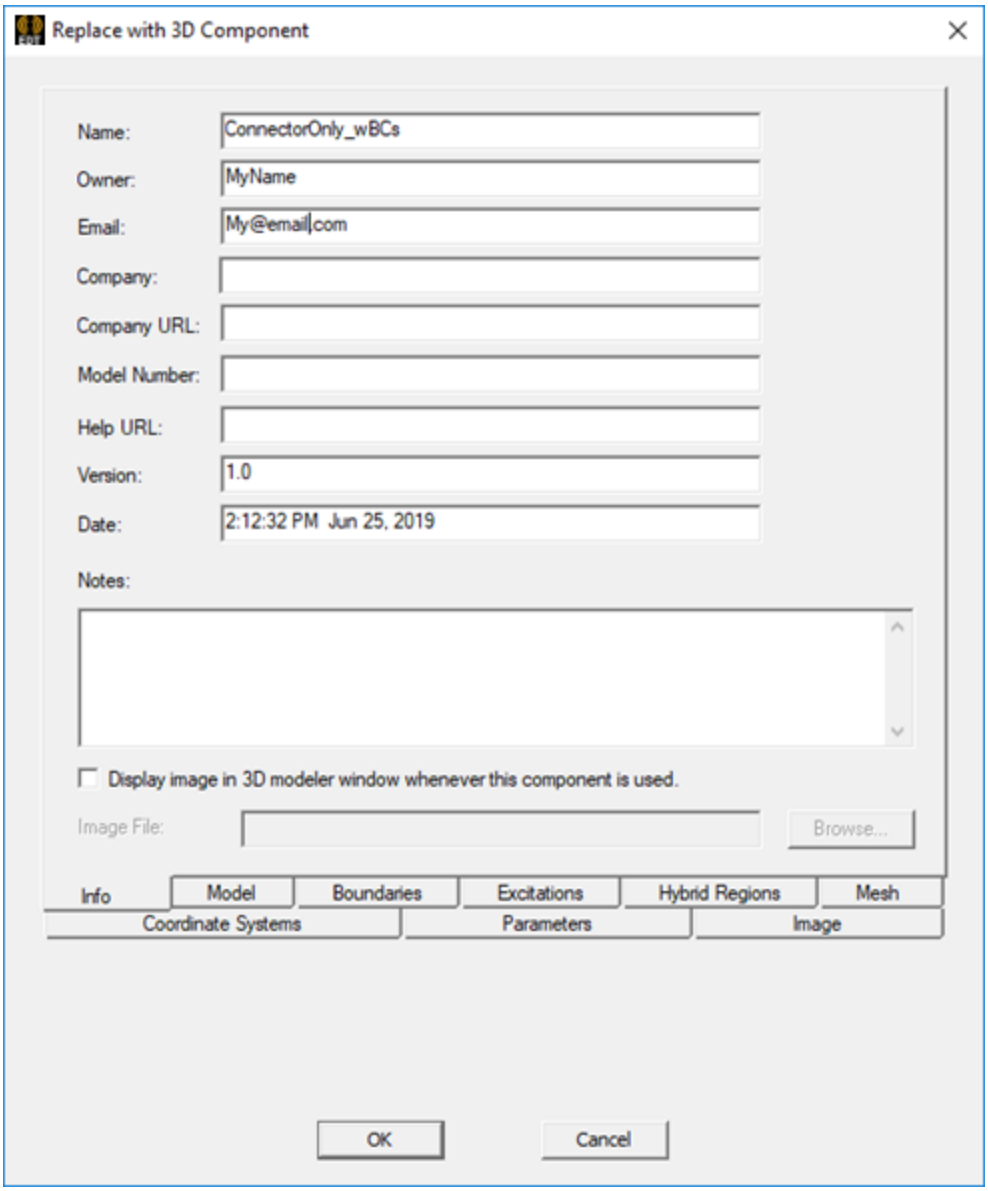
Replace Selection by 3D Component

To replace a selection in the 3D modeler window with a 3D component version of the selection, select an existing object or model, right-click, and select **Replace with 3D Component**. You can also click **Draw** > **3D Component Library** > **Replace with 3D Component**. You cannot replace components from a design with an array setup. However, you create a 3D component from a unit cell and use that to create a Multi-Unit Cell array. When you replace a selection, object names are retained, port names are retained, component parameters map to existing design variables and report, field plot, and edit source are all preserved.



This command opens the **Replace with 3D Component** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The tabs let you view the component features organized according to geometry, design data, and settings. The **Replace with 3D Component** dialog box does not offer tabs for encryption or licensing, compared to Creating a 3D Component. You can use these tabs to change inclusion of objects, boundaries, ports, coordinate systems, or parameters for the 3D component. When you click **OK**, no *Save* dialog box is shown.

- Info tab
- Model tab
- Boundaries tab
- Excitations tab
- Mesh tab

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

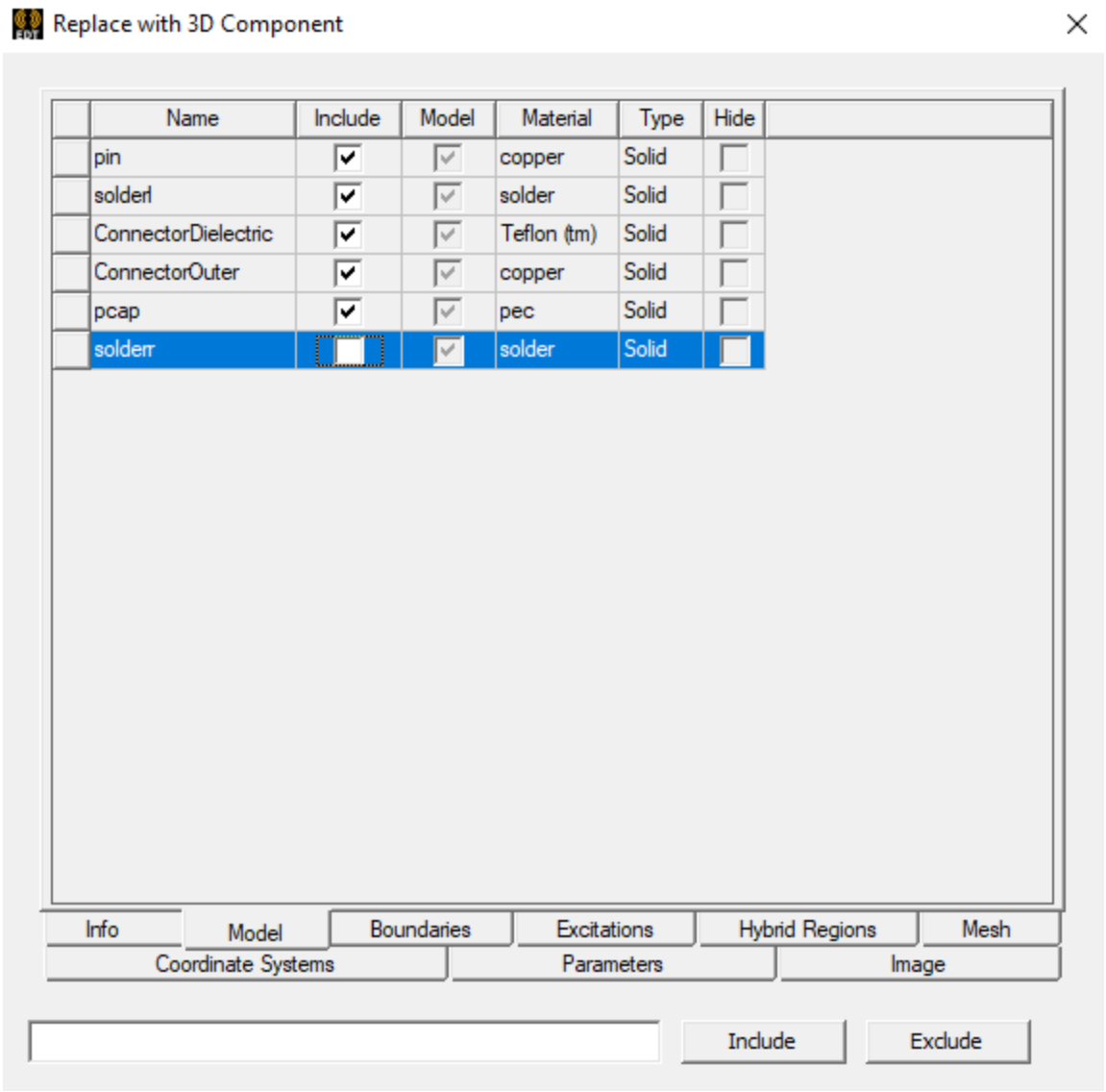
- Coordinate Systems tab
- Parameters tab
- Image tab

When you click OK, no save dialog is shown. The 3D component is saved to a temporary location. Insertion of instances uses the temporary file, but works just as regular 3D component insertion. The Project tree shows the Component under 3D Components, the boundaries are no longer listed under the Boundaries icon. Excitations, Reports and Analysis Setups remain the same. History Tree no longer shows detailed model edits. You can **Undo** and **Redo**.

Include or Exclude Feature from Component

You can view the various tabs and select which features to include or exclude while replacing the selection with a component. You can use the check boxes in the **Include** column for this purpose. You can also type a feature *Name* in the text box and use the **Include** or **Exclude** button. The text field supports wildcard characters (* and ?), so you can toggle the inclusion state for multiple objects with similar names in a single operation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



An object's **Include** option under the *Model* tab impacts the inclusion state in other tabs. For example, if you exclude an object from the component creation, the design data, parameters, and coordinate systems corresponding to that object are also excluded.

Sorting Columns Using the Headers

The features listed for each tab include columns for the Name, check boxes for whether to include, and properties. You can click on the column headers to sort by Name, Model, or other feature listed for each tab. Click the column header to selects the column to sort. Click again to invert the column.

You can sort lists by using all columns except Include.

Design Data Notes:

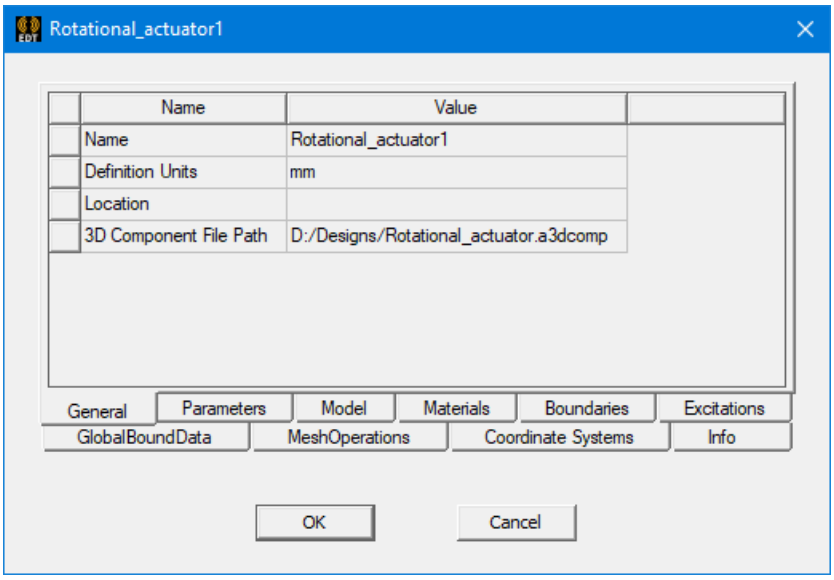
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- DC thickness on selected objects is always included in components. It is not listed in the **Boundaries** tab.
- The **Create 3D Component** dialog does not list design data without an assignment .
- The parent of any included boundary/excitation is included, as long as the parent does not require assignment.
- Design settings like material overrides are included.

Once you have made the Include and Exclude settings, and have specified the Image and Info you click **OK** to Replace with 3D Component. When you click OK, the component is validated for everything that is included in the component. For example, you cannot include boundaries if the object on which the boundaries have been created is not included.

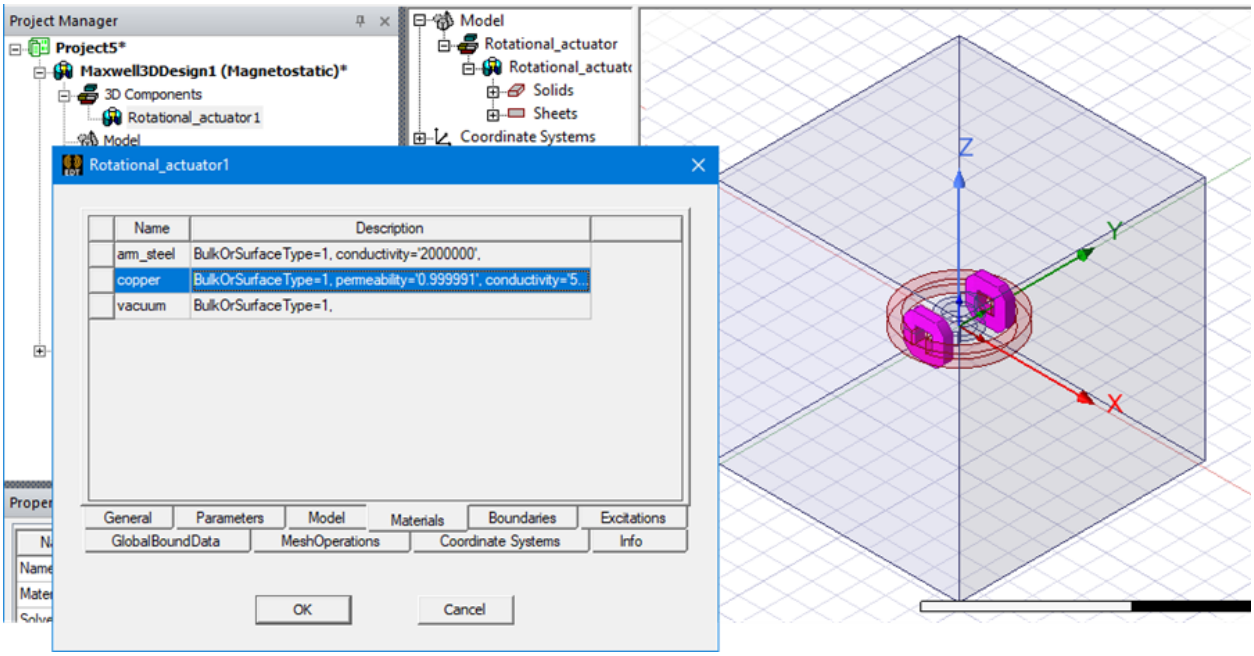
Viewing Properties of 3D Components

The Properties displayed for a selected 3D component depend on how you select that component. For a complete display of Properties, right-click the selected component (Project Manager, History Tree, or Modeler window) and click **Properties** on the shortcut menu. Alternatively, click **Edit** > **Properties** using the menu bar. This action displays a window with tabs for all component properties.

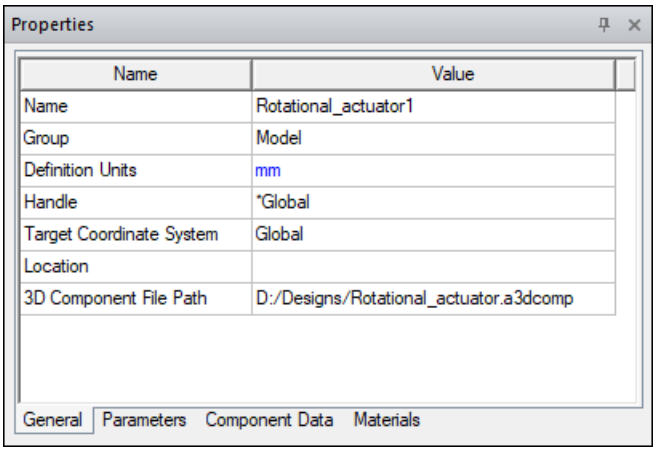


If you choose the *Materials* tab, you can then select any listed material, and the parts of the component made of that material are highlighted in the Modeler window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



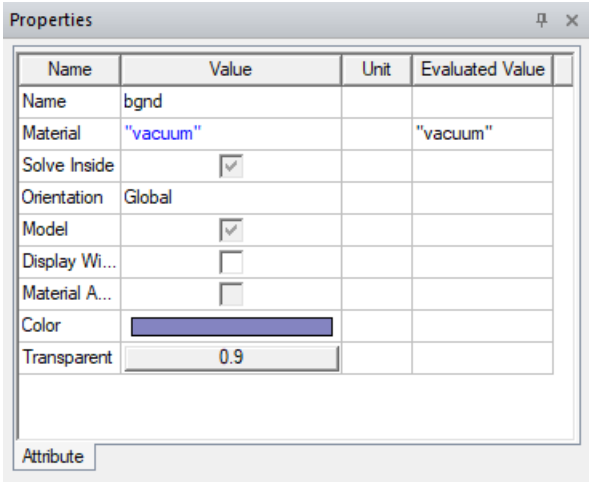
Selecting a 3D component in the Project Manager or the history tree displays tabs for *General, Materials,* and *Component Data* in the docked *Properties* window.



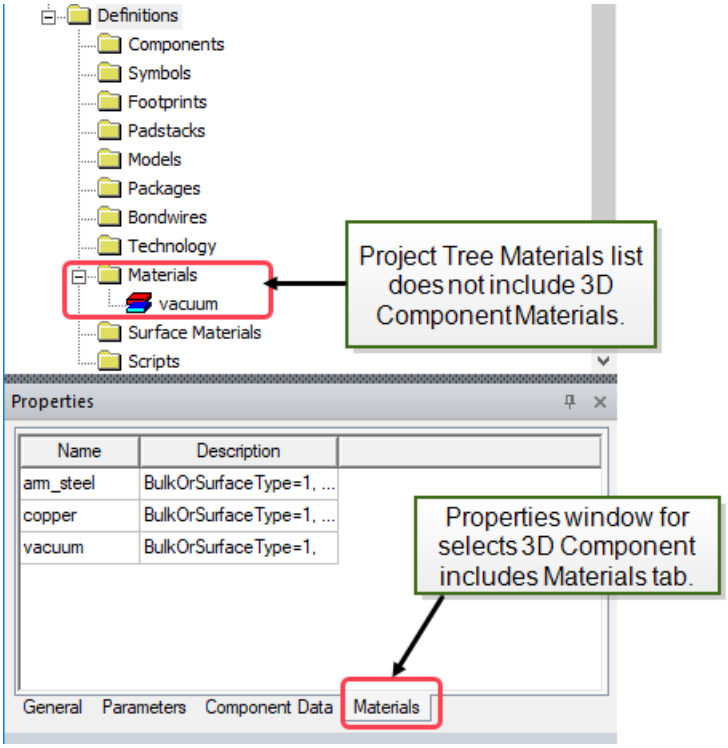
Note that the Handle property lists the handle coordinate system that was selected as reference when the 3D component was created; from this window, it can be changed to any coordinate system in the 3D component.

Selecting a 3D component object in the Modeler window displays the Attributes tab in the docked Properties window:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



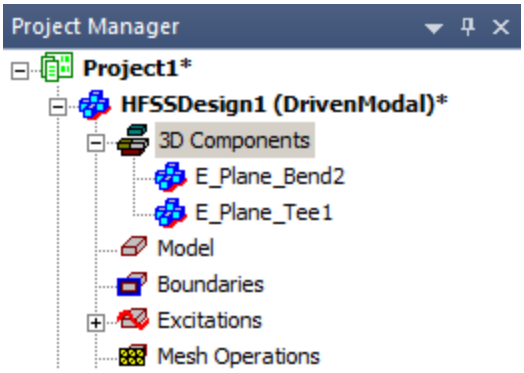
Component materials do not appear in the Project Manager's Definitions list. If you select the icon for an inserted component, the docked *Properties* window contains a *Materials* tab that shows the materials defined for the selected component.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

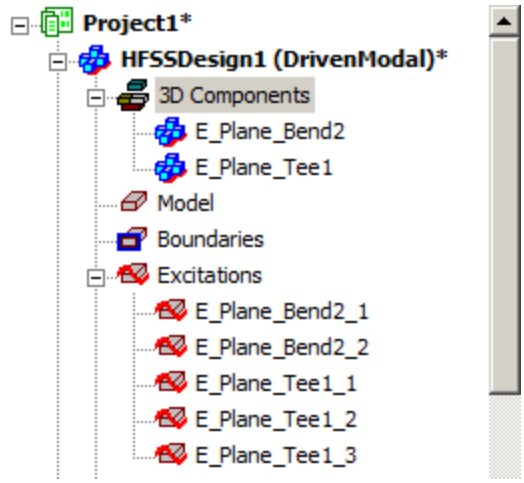
3D Components in the Project Manager Window

The **Project Manager** window organizes any 3D components that you add to a design under the 3D Components icon in the tree.



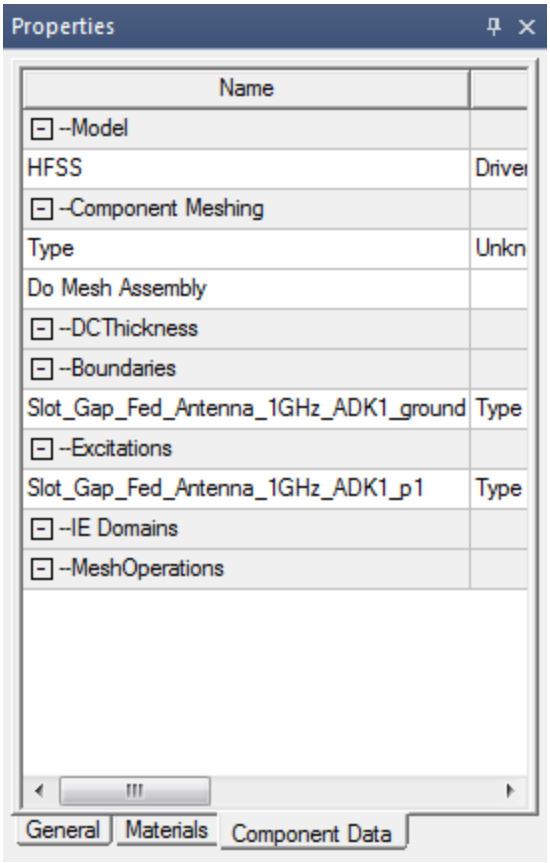
Selecting these inserted components populates the docked **Properties** window, which has tabs that display the instance attributes.

Excitations included in the 3D Components appear in the Project Manager. This gives you the ability to edit Post Processing settings, such as the de-embed distance, renormalization, and power settings.

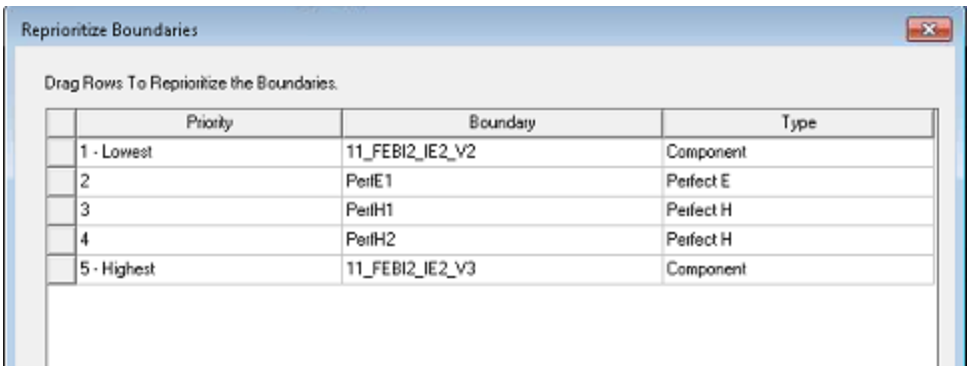


Boundaries defined inside the component do not appear in the Project Manager. However, boundary properties can be seen when you select the component in the Project Manager and view the component properties under the *Component Data* tab of the docked *Properties* window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



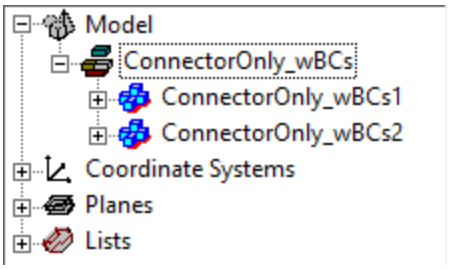
However, you can use **Maxwell** > **Boundaries** > **Reprioritize** to set the priorities of all boundaries relative to non-component boundaries.



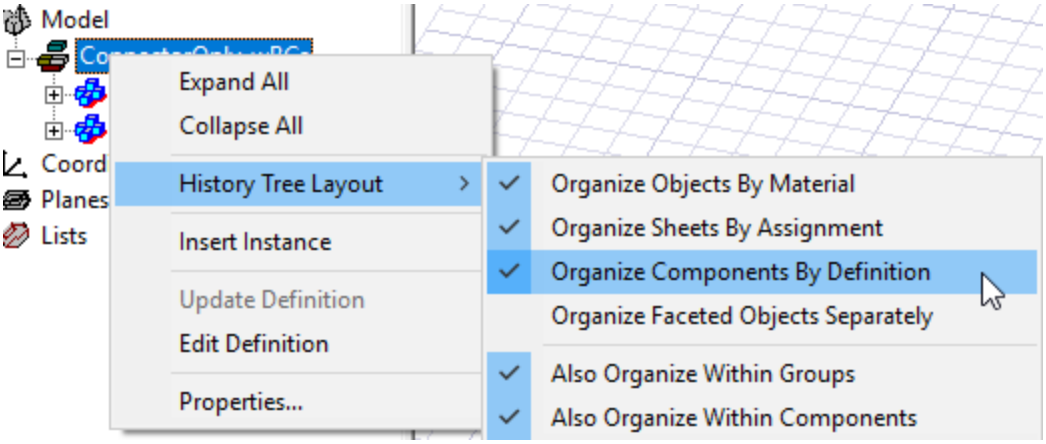
3D Components in the History Tree

Although Group Commands do not work with 3D components, you can group all instance of a components by the component definition in the History tree.

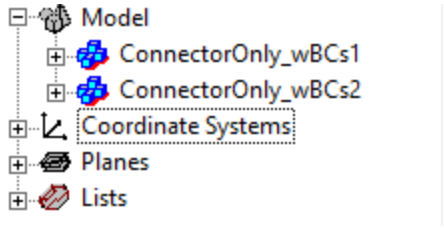
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To disable this grouping by instance, right-click an instance, and unselect **History Tree Layout>Organize Components by Definition**.

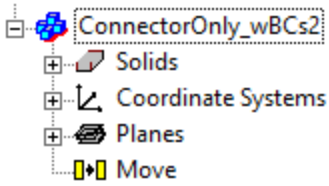


This causes the display to show numbered instances listed without the icon for the primary component.



Component Operations in the History Tree

- Component operations appear under that component folder toward the end.



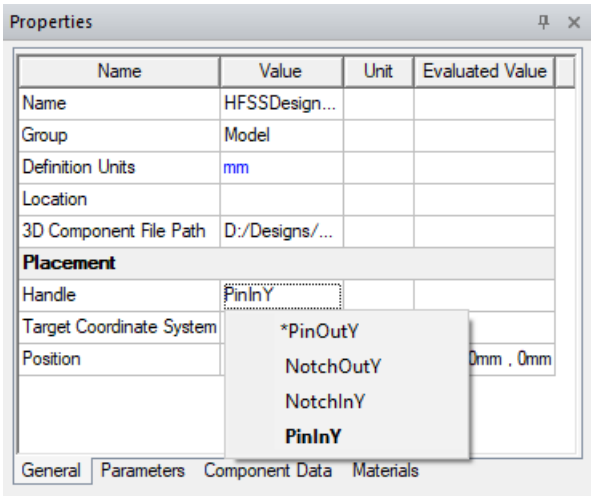
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Component operations acts on parts as well as CS.
- Boundaries, mesh operations are duplicated with duplicate and copy/paste operation.

Editing 3D Component Properties

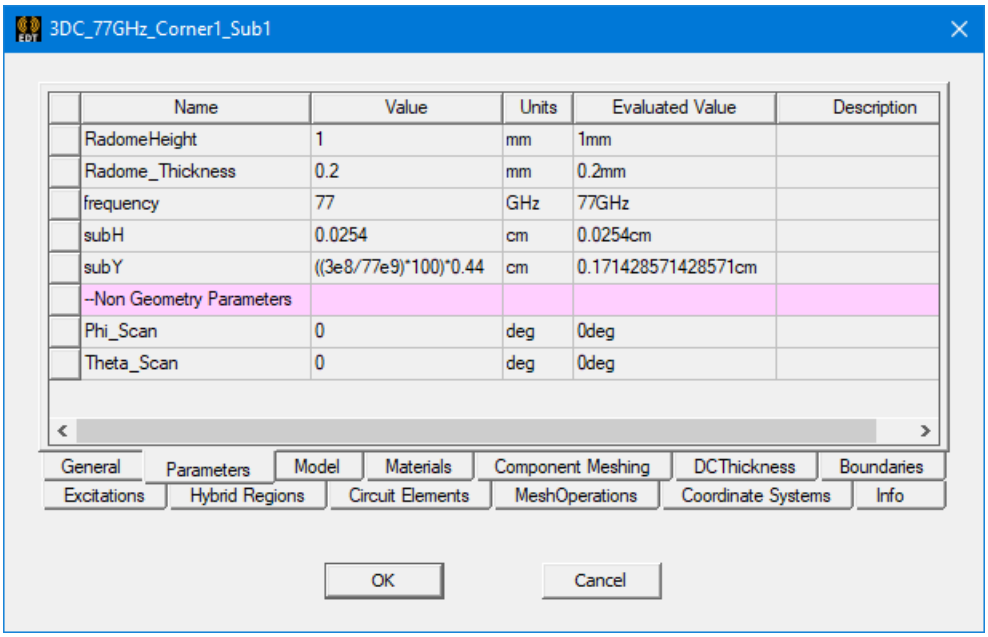
You can change and manipulate 3D components in several ways:

- Edit component handle and target CS in the General tab of component instance properties. All CS that are included in the component are available to be used as the handle. All coordinate systems in the assembly design that are eligible for component placement are available as target CS. Note that the default handle is shown with an asterisk.

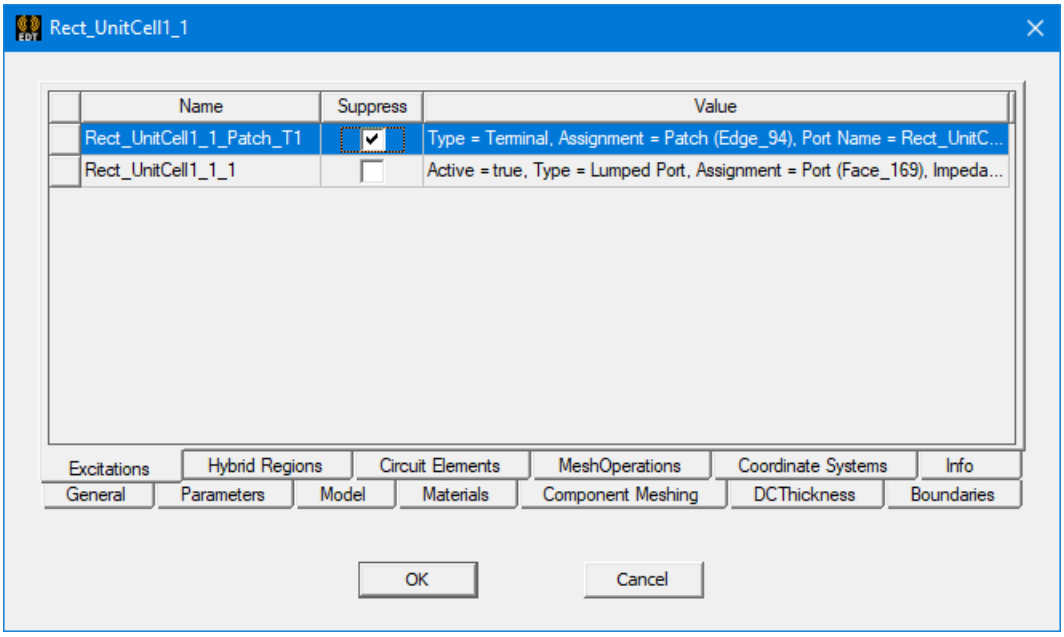


- Edit component parameters, if defined for that component. Editing the parameters of one component instance affects only that instance. If a component includes non-Geometry parameters, these are grouped separately on the Parameters tab.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



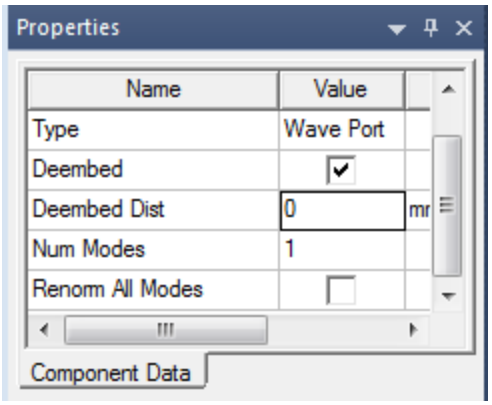
- Edit excitations by suppressing their use by means of the **Excitations** tab in the **Properties** window.



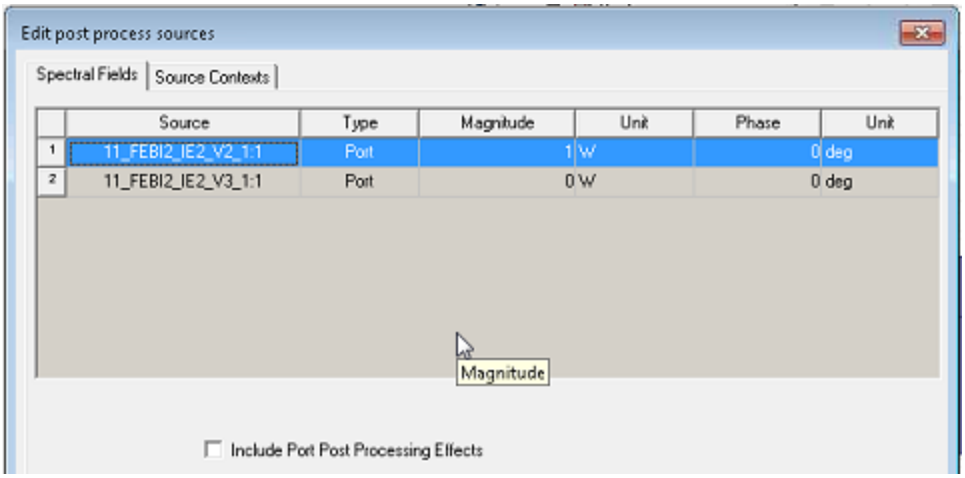
Checking Suppress in the Properties window removes the port on the selected instance from the Excitations list on the Project tree. Unchecking Suppress activates that port for the model and causes the port to appear in the Excitations list.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Edit excitation Post Processing such as de-embed distance, renormalization.



- Edit Sources for Post Processing



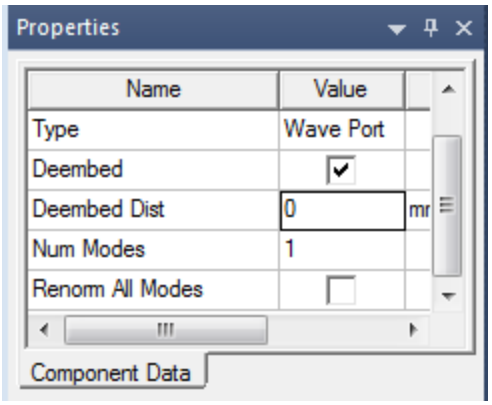
- Edit Port Impedance Multiplier value
- Edit Mesh operations coming from component

> **Note:**
>
> Operations on individual parts of a component are not allowed.

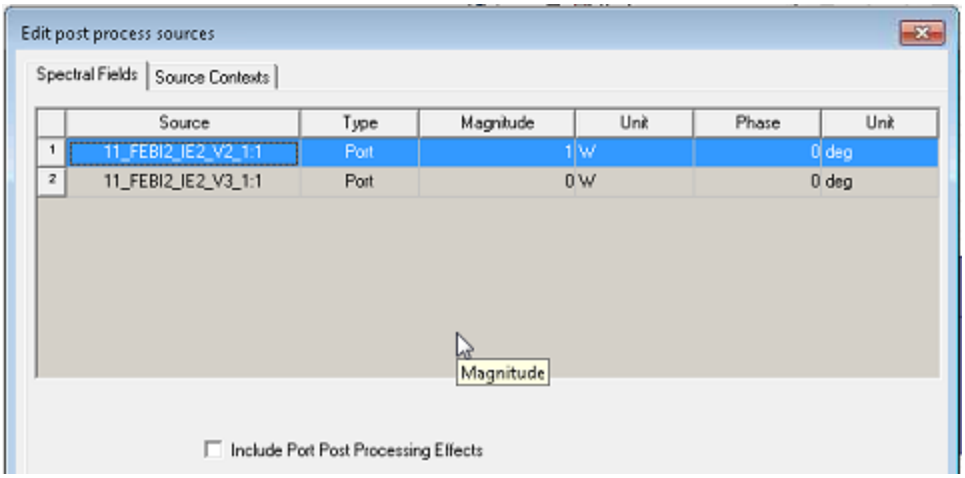
- Copy/Paste component
- Delete component
- Arrange (Move, Rotate, Mirror) component
- Duplicate component (Along Line, Around Axis, or Mirror)
- Edit Attributes for Display Wire frame, Color, and Transparency.
- Use **Edit** > **Select** > **Submodel** to quickly select component

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Edit excitation Post Processing such as de-embed distance, renormalization.



- Edit Sources for Post Processing



- Edit Port Impedance Multiplier value
- Edit Mesh operations coming from component

> **Note:**
>
> Operations on individual parts of component are not allowed.

- Copy/Paste component
- Delete component
- Arrange component: Move, Rotate, and Mirror
- Duplicate component: Along Line, Around Axis, and Mirror
- Edit Attributes for Display Wire frame, Color, and Transparency
- Use **Edit** > **Select** > **Submodel** to quickly select component

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

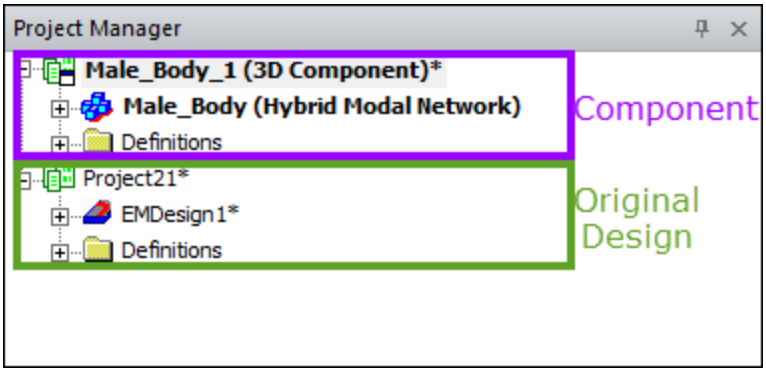
Editing a 3D Component Definition

> **Note:**
>
> If you reached this page looking for help on editing the definition of a **Layout Component**, as opposed to a *3D Component*, go here instead.

The **Edit Definition** feature allows the user to create new component definition using an existing, unencrypted 3D component file. This speeds up the task of creating new versions of components without having to start from the beginning. When **Edit Definition** is initiated, **Electronics Desktop** automatically creates a new project containing all 3D component data from the file in 'exploded' form. Component materials are added at the project level; variables are added at the project and design level; a detailed history is created for all objects in the component; and design data is added to the appropriate folders in the Project Manager.

> **Important:**
>
> Only 3D components that are **not encrypted** can be edited as described in this section.

The **Edit Definition** command creates a new project, named for the component, in the **Project Manager** window.

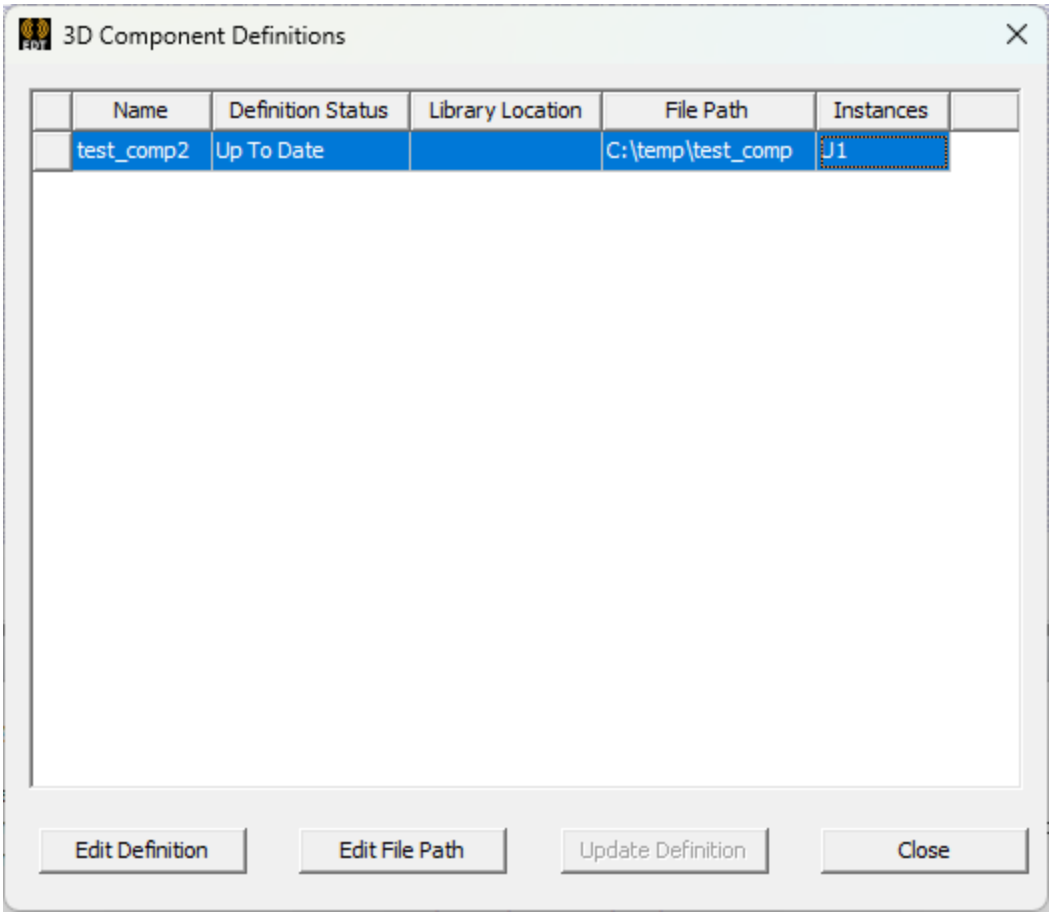


When the edited component is saved, an **Update 3D Component** window appears. From the **Update 3D Component** window, choose whether to save the edited component as a local instance or to the original library. To edit a 3D component definition:

1. Navigate to the **Edit Definition** window by following one of the following paths:
   - From the **History Tree**, right-click the component and select **Edit Definition**.
   - From the **History Tree**, right-click an instance under the component and select **Components** > **Edit Definition**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

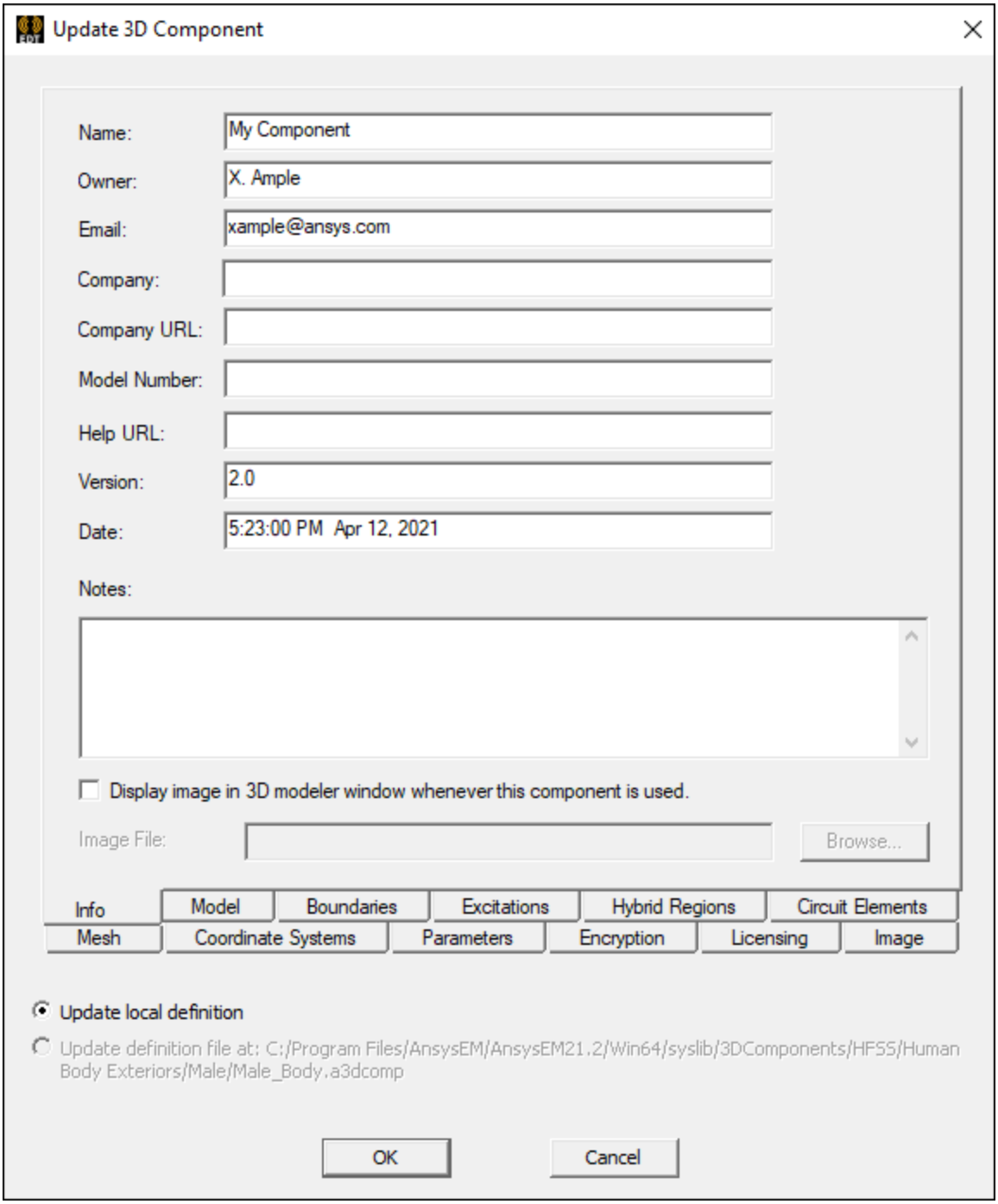
- From the **Project Manager**, right-click **3D Components** and select **Definitions** to open the **3D Component Definitions** window. From the **3D Component Definitions** window, select a component and click **Edit Definition**.



Regardless of which path is taken, a copy of the 3D component opens in the **Layout Editor**, and a new project folder appears in the **Project Manager** window, containing a copy of the original design.

2. Edit the component as desired.
3. To save the component as a new component (*.*a3dcomp*) or as its own **Electronics Desktop** (*.*aedt*) file, click **File** > **Save As**. Specify a file name and destination and click **Save**.
4. To save the component in the current project or for all projects, click **File** > **Save** to open the **Update 3D Component** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This window is populated with all entities (objects, boundaries, excitations, etc.) in the component design.

5. Use the radio buttons to select one of the following options:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Update Local Definition** – updates the component definition for use in the original design only.
- **Update Definition File At <Path>** – updates the component in the component library, affecting all other instances.

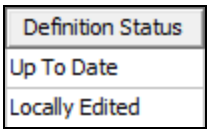
6. Click **OK**.

Verifying Updates to Components

To verify that a component has been edited:

1. Access the **3D Component Definitions** window:
   - From the **Project Manager**, right-click **3D Components** and select **Definitions**.

   The **Definition Status** column shows "Locally Edited" for any altered components.



2. Click **Close** to exit the window.

Updating Definitions

Once the updated component is saved, return to the original design and update all instances of that component. See: Updating 3D Component Definitions.

> **Note:**
>
> Definition update is not automatic even if **Edit Component** was initiated from that design. Instead, when an assembly design becomes active, a dialog box appears, prompting the user to update the 3D component definition. Then choose to update the definition from the prompt.

To update a component's definition:

1. Access the **3D Component Definitions** window:
   - From the **Project Manager**, right-click **3D Components** and select **Definitions**.

   Components with updates available display "Update Available" in the **Definition Status** column.

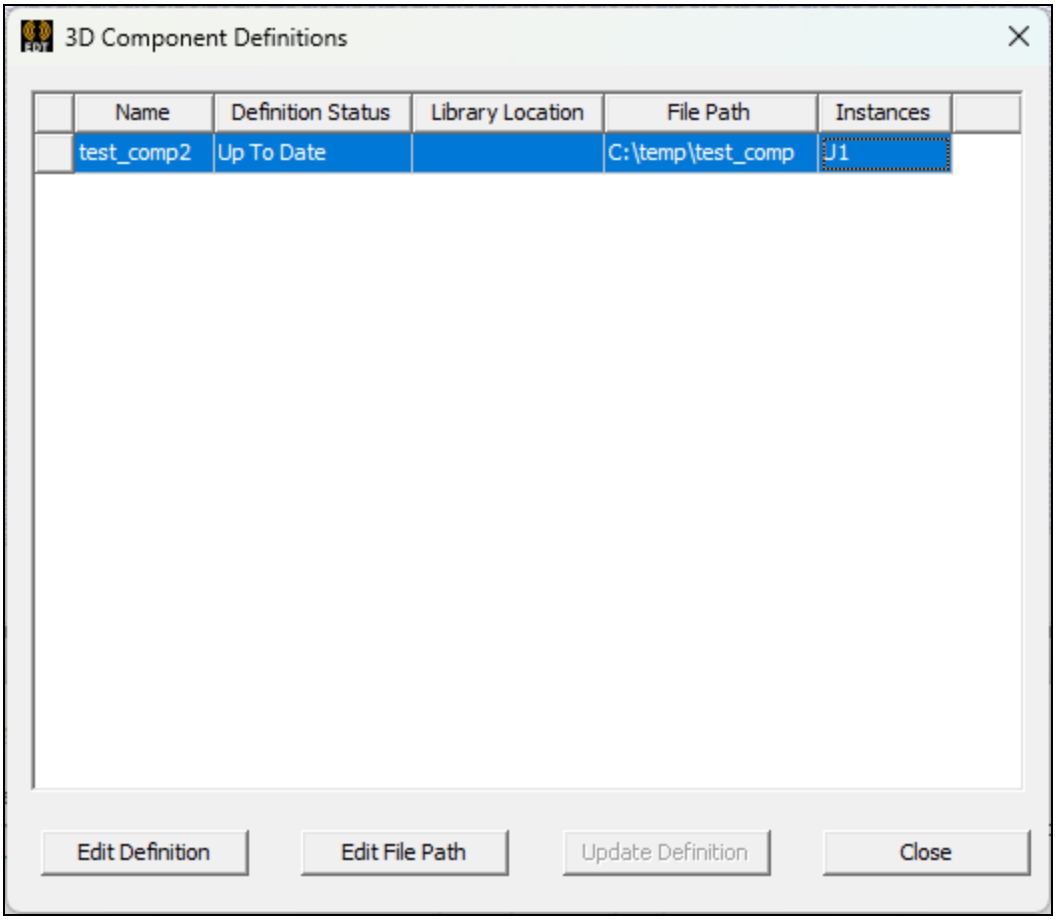
2. Select the desired component and click **Update Definition**.
3. Click **Close** to exit the window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Replacing 3D Components

3D components (and/or layoutcomponents) can be replaced everywhere they have been placed in a design simultaneously from the **3D Component Definitions** window. Complete these steps to make a replacement.

1. Navigate to **Layout** > **3D Component Definitions** to open the **3D Component Definitions** window.
2. Highlight a component and click **Edit File Path** to open an explorer window.



3. Navigate to the new component and either select it and click **Open** or double-click it to immediately replace the selected component. Keep in mind the following:

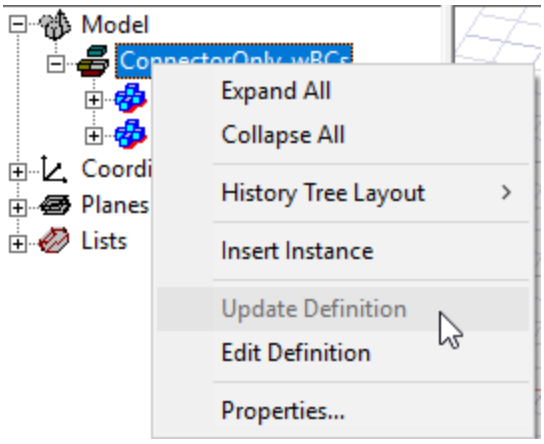
- All instances of the definition within the active design are replaced.
- All transformations (i.e., modifications) to the instance(s) are retained.
- All ports previously added to the instance(s) are also retained.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Names are not retained.
- The replacement is irreversible.

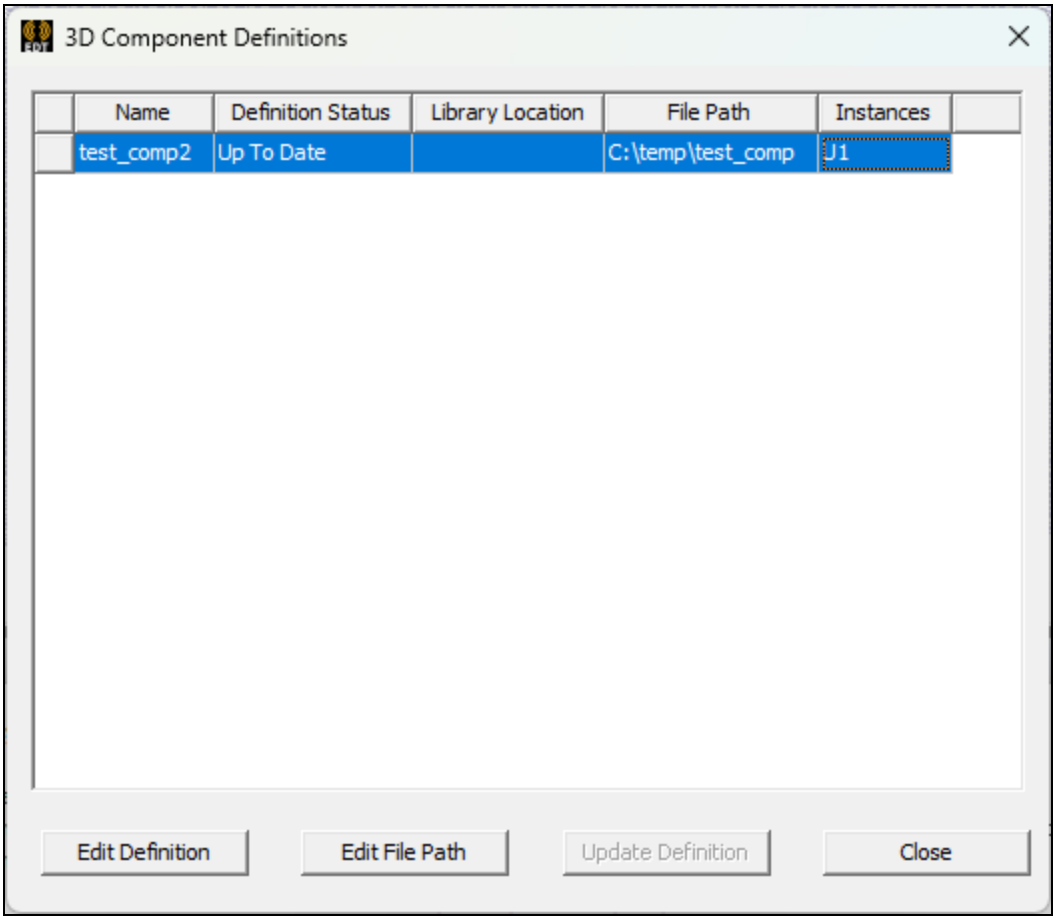
Updating a 3D Component Definition

If you create or obtain a new version of a component, and then open a project that contained an earlier version of that component, you can right-click the component in the History tree and select **Update Definition** to replace the older version of the component with the newer one This updates all the instances of the component using that definition. Update Definition is unavailable if a newer definition is unavailable.



You can also accomplish this by right-clicking on the 3D Component icon in the Project Tree and selecting **Definitions** from the shortcut menu. This displays a dialog listing available component definitions. The Definition status column lets you know if an undated definition is available.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



All instances and component operations in the design are updated:

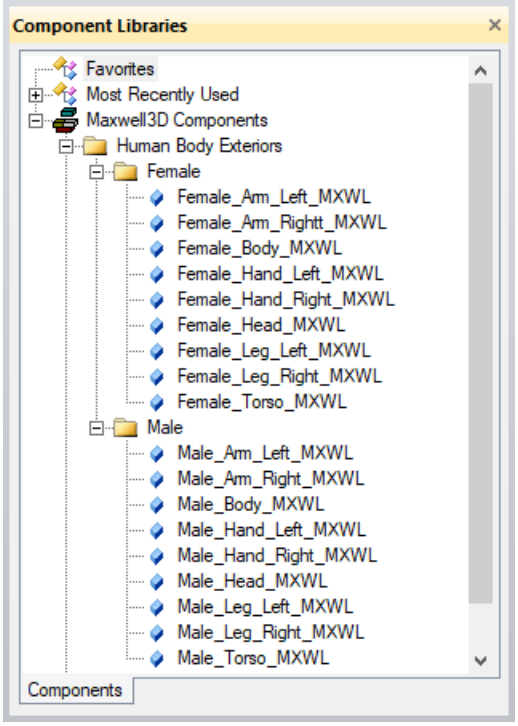
- Parameter values are preserved.
- New parts are generated in all instances.
- Deleted parts are deleted from all instances.
- Edited parts are changed in all instances.
- Design data, materials, coordinate systems can also be updated.
- Undo is supported; command is scriptable.
- The Definitions dialog box provides complete list of definitions in the design.
- Definitions can be updated from the dialog box.

Human Body Exteriors

There are two ways to view the Human Body Exteriors in the 3D Component Libraries. If you have enabled **View** > **Component Libraries**, you can navigate to the different models in the tree. From the **Component Libraries** tree, you can drag and drop models to the **Modeler**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

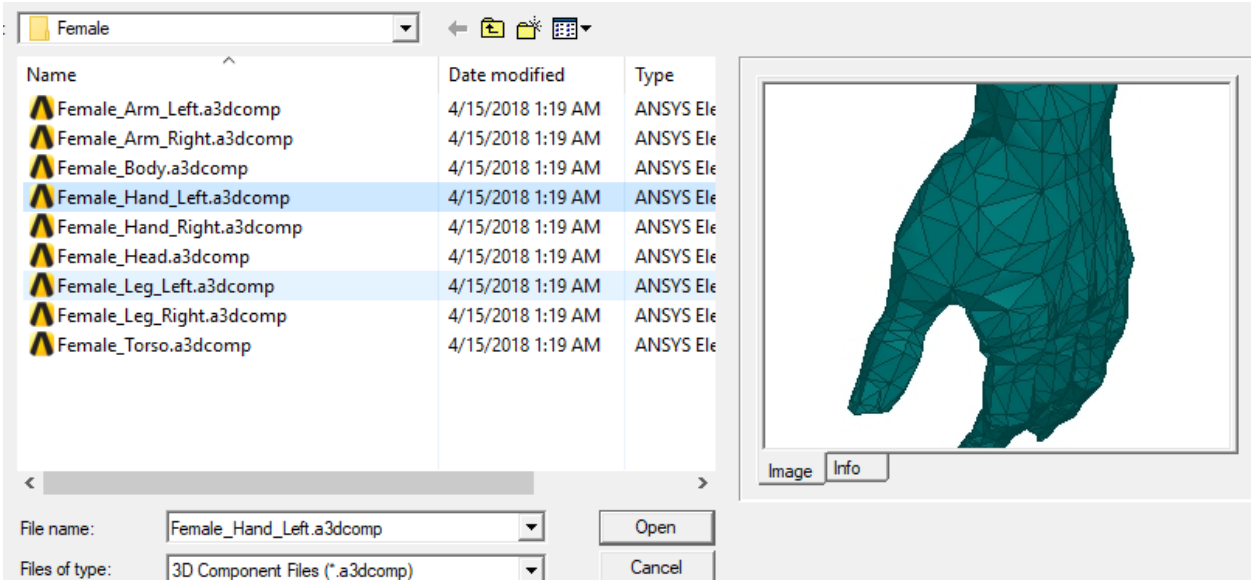
window.



You can also access the **Human Body Exteriors** library using **Draw** > **3D Component Library** > **Browse...** to open the **Browse 3D Component** dialog box. Click on the **SysLib** button at the bottom of the dialog box, then navigate to \\Maxwell3D\Human Body Exteriors.

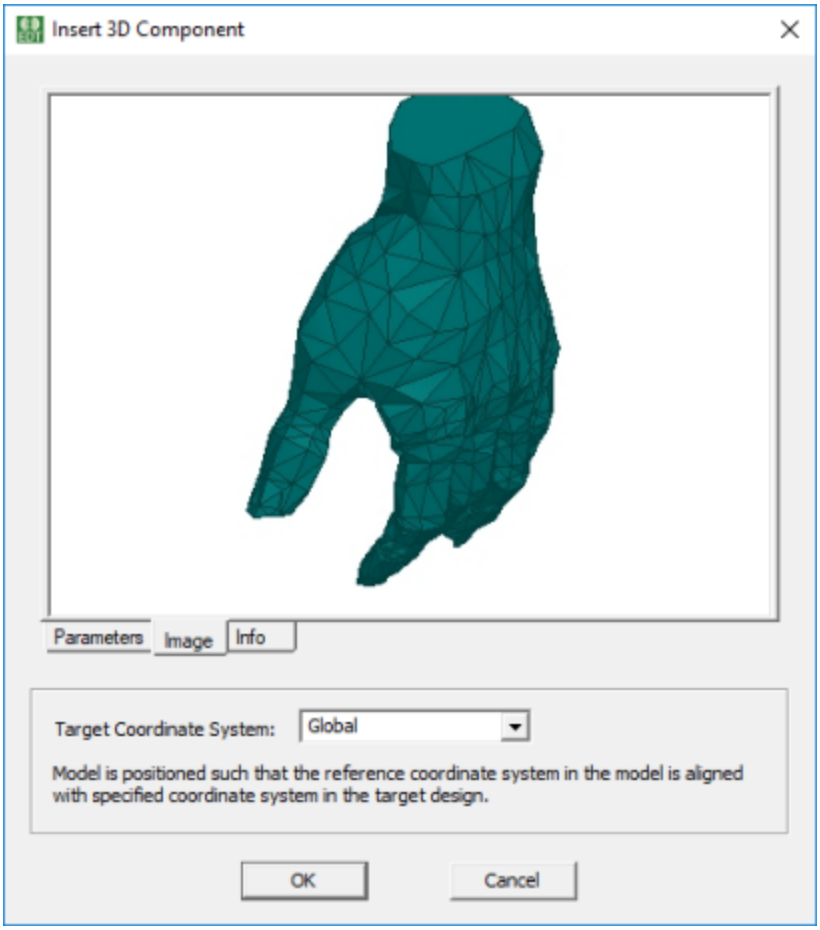
You can select from the Human Body Exterior Male and Female models. Once you have selected a component file, you can view the image and info for that component.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can click **Open** to display the **Insert 3D Component** window. You can view the **Image** and **Info** tabs for that component. Human Body Exteriors do not have parameters.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can also open the **Insert 3D Component** window by double-clicking on a model in the **Component Libraries** window. See Inserting a Component into a Design.

If you want to edit a 3D Component:

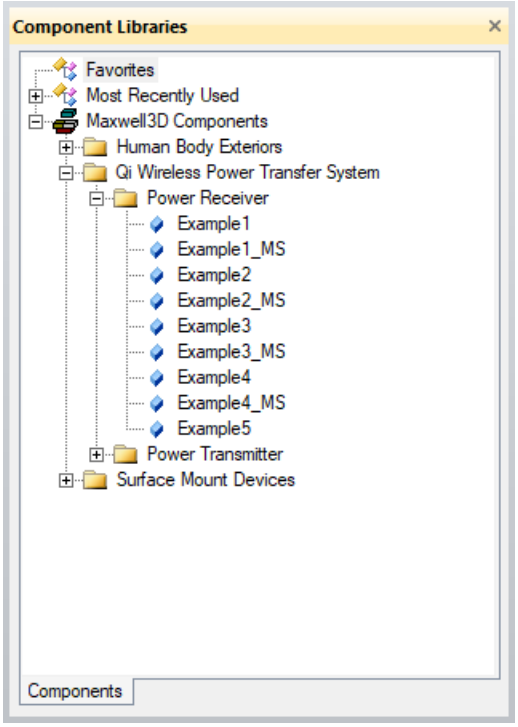
1. From Windows Explorer, copy and save the component design file to your **PersonalLib** or **UserLib** directory.
2. Do an update menu for 3D Components.
3. Open the component from the **PersonalLib** or **UserLib** directory and use edit mode for making modifications.

Qi Wireless Power Transfer System

> **Note:** The Ansys models included in the Qi Wireless Power Transfer System 3D Component library are based on model specifications published by the *Wireless Power Consortium*.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

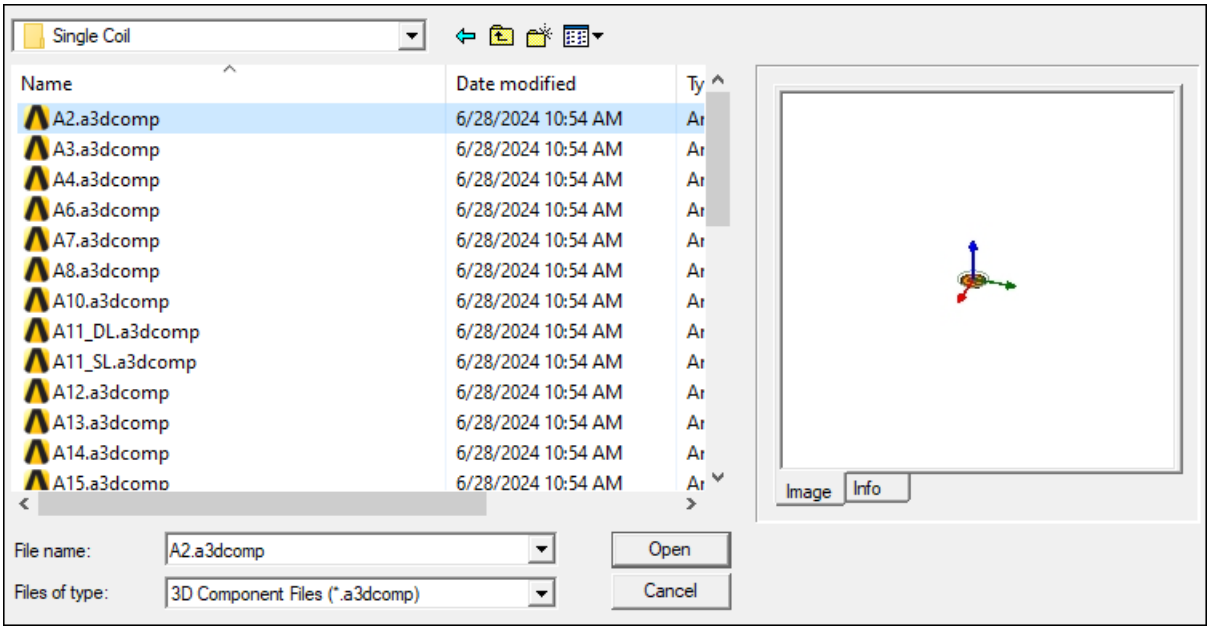
There are two ways to view the Qi Wireless Power Transfer System components in the 3D Component Libraries. If you have enabled **View** > **Component Libraries**, you can navigate to the different models in the tree. From the **Component Libraries** tree, you can drag and drop models to the **Modeler** window.



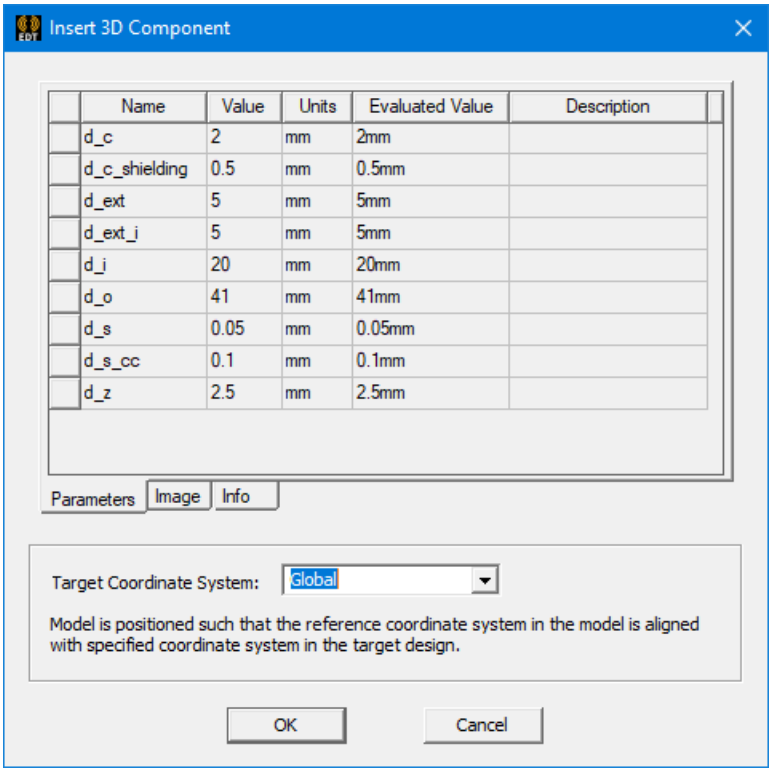
You can also access the **Qi Wireless Power Transfer System** library using **Draw** > **3D Component Library** > **Browse...** to open the **Browse 3D Component** dialog box. You then click **SysLib** to display the libraries included in your installation.

You can select from the Qi Wireless Power Transfer System models, and navigate the folders to select from available components. Once you have selected a Component file, you can view the Image and Info for that component.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can click **Open** to display the **Insert 3D Component** window. You can view the Parameters, Image, and Info tabs for that component.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also open the **Insert 3D Component** window by double-clicking on a model in the **Component Libraries** window. (Refer to Inserting a Component into a Design.)

If you want to edit a 3D Component:

1. From Windows Explorer, copy and save the component design file to your **PersonalLib** or **UserLib** directory.
2. Open the component from the **PersonalLib** or **UserLib** directory and edit the component. (Refer to Editing a 3D Component Definition.)

Legacy Projects with 3D Components

Components created in earlier projects do not have the same degree of encapsulization and are not compatible with the current version and Ansys Electronics Desktop.

Legacy Projects: Do not encapsulate components

Modifying Objects

You can quickly modify the position, dimensions, and other characteristics of objects created in the 3D Modeler window.

What do you want to do?

Object and History Editing	**Modify Object Appearance**
Modify Object Dimensions	**Modify Object Location or Orientation**
Surface or Edge Operations	**Boolean Operations on Objects**

Modify Object Appearance

- Assign color to an object.
- Assign transparency to an object.
- Also see Modifying the Model View

Modify Object Dimensions

- Assigning a Cross Section and Dimension to a Polyline
- Scale the size of objects
- Connect objects
- Move faces or edges
- Convert polyline segments

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Rounding the edge of an object (Fillet)
- Flattening the edge of an object (Chamfer)

Modify Object Location or Orientation

- Move objects
- Rotate objects
- Change the Orientation of an object
- Mirror objects about a plane.
- Offset an object (move every face of an object).
- Also see Modifying the Model View

Surface or Edge Operations for Objects

- Sweep objects.
- Cover lines.
- Cover faces.
- Uncover faces.
- Detach faces.
- Detach edges.
- Create a new object by taking a cross-section of a 3D object.
- Wrap Command
- Project Sheet
- Thicken Sheet

Boolean Operations on Objects

- Unite objects.
- Subtract objects.
- Create objects from intersections.
- Create an object from a face.
- Create an object from an edge.
- Split objects.
- Separate objects.
- Imprint Projection Commands
- Imprinting an Object

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning Color to Objects

1. Select the object to which you want to assign a color.

   > **Note:**
   >
   > If the *Properties* window is not visible on the desktop, click **View> Properties Window** or use **Edit> Properties**.

2. In the *Properties* dialog box or docked *Properties* window, click the **Attribute** tab.
3. Click the color sample in the **Value** column of the **Color** row.

   The *Color* palette appears.

4. Select a *Basic* or *Custom* color from the **Color** palette.

   Alternatively, you can specify a color by typing numbers between 0 and 255 to specify the following values:

   - **Hue, Sat** (saturation), and **Lum** (luminance), or
   - **Red, Green,** and **Blue**

   You can also specify a color by clicking within the color spectrum box and using the luminance slider to the right of this box.

5. Optionally, click **Add to Custom Colors** to save the settings as a custom color, which makes in easier to specify the same color for other objects.

   > **Note:**
   >
   > If you wish to overwrite a previously defined or default custom color, choose the custom color swatch to overwrite *before* specifying the new color parameters. To clear your customizations and restore default choices, click **Reset Custom Colors to Defaults**.

6. Click **OK**.

   The color is assigned to the selected object.

Setting the Default Color of Objects

The default color you specify for objects is applicable to when the objects are rendered as solids (shaded) or rendered as wireframes (that is, outlines only). The outline color is the same as the object color you specify according to the following procedure:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Using the menu bar, click **Tools**> **Options**> **General Options**, or, from the **Desktop** ribbon tab, click **General Options**.
2. In the tree at the left side of the *Options* dialog box, select **3D Modeler**> **Display**> **Rendering**.
3. In the *Object Appearance* section, click the color sample button to the right of **Default color**.

   The **Color** palette appears.
4. Select a color from the **Color** palette, as described in the parent topic.

   Any objects you draw after this point will be assigned the default color you selected. However, if the *Material Appearance* option is selected within the *Attributes* tab of the object properties, material color and transparency assignments will override the default assignments.

   Alternatively, select **Use material appearance if available** within the *Options* dialog box to use the material color and transparency values by default (when they exist).
5. Click **OK**.

Assigning Transparency to an Object

1. Select the object to which you want to assign a transparency.

   > **Note:**
   >
   > If the *Properties* window not visible on the desktop, click **View** > **Properties Window** or use **Edit** > **Properties**.

2. In the *Properties* dialog box or docked *Properties* window, click the **Attribute** tab.
3. Click the *Value* in the **Transparent** row.

   The *Set Transparency* dialog box appears.
4. Move the slider to the right to increase the transparency of the object. Move the slider to the left to decrease the transparency of the object. Or, type a decimal number between 0 and 1 (inclusive) in the box to the right of the slider.
5. Click **OK**.

Setting the Default Transparency of Objects

1. Using the menu bar, click **Tools** > **Options** > **General Options**, or, from the **Desktop** ribbon tab, click **General Options**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the tree at the left side of the *Options* dialog box, select **3D Modeler** > **Display** > **Rendering**.
3. In the *Object Appearance* section, click and drag the **Default Transparency** slider. Move the slider to the right to increase the default object transparency. Move the slider to the left to decrease the default object transparency. Or, type a decimal number between 0 and 1 (inclusive) in the box to the right of the slider.

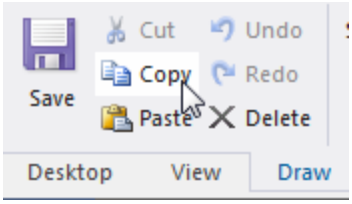
   Any objects you draw after this point will be assigned the default transparency you specified. However, if the *Material Appearance* option is selected within the *Attributes* tab of the object properties, material color and transparency assignments will override the default assignments.

   Alternatively, select **Use material appearance if available** within the *Options* dialog box to use the material color and transparency values by default (when they exist).

Copying and Pasting Objects

To copy objects and paste them in the same design or another design, use the **Edit** > **Copy** and **Edit** > **Paste** commands. For data link purposes, where you want to include the material assignments as well as the geometry, you can use the Import from Clipboard command.

1. Select the objects you want to copy.
2. Click **Edit** > **Copy** or in the **View/Draw/Model** tabs of the ribbon, click the **Copy** icon:



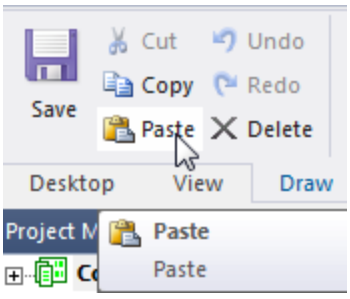
   The objects are copied to the Clipboard, a temporary storage area. The selected items are not deleted.

   To cut an item to the clipboard and deleting the original, use the scissors icon on the toolbar.

3. Select the design into which you want to paste the objects. It can be the same design from which you copied the items.
4. Click in the **3D Modeler** window.
5. Select the working coordinate system. Objects are pasted relative to the current working coordinate system.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Click **Edit** > **Paste** or in the **View/Draw/Model** tabs of the ribbon, click the **Paste** icon:



   The objects appear in the targeted Modeler window and in the new History tree. The pasted objects become the current active selection. If you undo the paste, the active selection goes back to the previous selection. If you redo, the pasted objects become the current active selection.

Items on the Clipboard can be pasted repeatedly. The items currently stored on the Clipboard are replaced by the next items that are cut or copied. Designs and projects containing encrypted components can be copy pasted for products using 3D Modeler

Import a Model from the Clipboard

You can import a model from the Clipboard in order to use the geometry from a different design. To use geometry with datalink, the geometry ID must be preserved. The **Import from Clipboard** command preserves geometry IDs.

To import a model from the Clipboard, the model for the current design must be empty. If the design is not empty, existing geometry is deleted.

The geometry model is imported from the Clipboard with the ID preserved.

1. Select the objects you want to copy from the source design. For selecting all objects, you can use **Edit** > **Select All** or **Ctrl+A**.
2. Click **Edit** > **Copy**.

   The objects are copied to the Clipboard, a temporary storage area. The selected items are not deleted from the source location.

3. Select the design into which you want to paste the objects. It can be in the same project as the design from which you copied the items or in a different project.
4. Click in the **3D Modeler** window.
5. Click **Modeler** > **Import From Clipboard**.

   The geometry is pasted from the Clipboard with the IDs preserved. Additionally, the source design's coordinate systems, materials, appearances, and other attributes are preserved and duplicated in the target design.

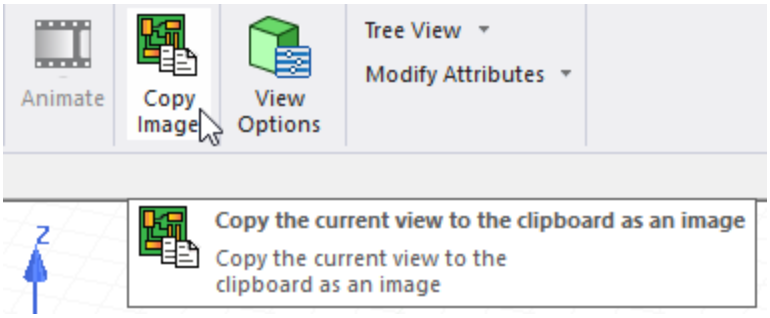
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copy Image

You can import images of the **3D Modeler** window or of Reports into any other application. The image has to be copied to the clipboard, so that it can be imported into the other application.

To copy an image of the Modeler window and paste it into another application:

1. Make the Modeler window from which you wish to capture an image active.
2. Copy an image to the clipboard using one of the following three methods:
   - From the menu bar, click **Edit** > **Copy Image**.
   - Right-click the Modeler window and select **Copy Image** from the shortcut menu.
   - On the **View** ribbon tab, click **Copy Image**:



   A raster image of the contents of the *Modeler* window is copied to the Clipboard.

3. Open the application into which you want to paste the image of your model and follow that application's image pasting procedure.

To copy an image of a Report to paste into another application:

1. Make the report the active window and use the **Copy Image** command from the **Edit** menu, short-cut menu, or **View** ribbon tab.

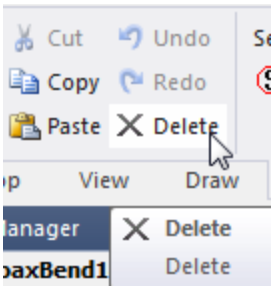
   An image of the report is copied to the Clipboard.

2. Open the application into which you want to paste an image of the report and follow that application's image pasting procedure.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Deleting Objects

1. Select the objects to delete.
2. Delete the selected objects using *one* of the following three methods:
   - From the menu bar, click **Edit** > **Delete**.
   - From the **View, Draw,** or **Model** ribbon tab, click the **Delete** icon:

   - Press the **Delete** key.

   The objects are deleted.

> **Note:**
>
> Boundaries, excitations, hybrid regions, and mesh operations that were associated only with deleted objects are also deleted along with the objects. However, associated design variables remain, and they can be assigned to other geometry.

Deleting Start Points and Endpoints

For individual segments of polylines, you can delete the start point and/or the end point, with the following conditions:

- For the *first* segment, you can only delete the *Start Point*, which effectively removes the entire segment.
- For the *last* segment, you can only delete the *End Point*, which effectively removes the entire segment.
- For all remaining (internal) segments, you can delete either the *Start Point* or the End Point. The new point becomes the opposite point of the adjacent segment, and the total number of segments is reduced by one.

To delete *Start Points* or *End Points* of polyline segments:

1. In the History Tree, locate the polyline that contains the segment you want to delete and expand the branch to expose the *CreateLine, CreateArc, CreateAngularArc,* or

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CreateSpline entries. These entries represent the individual polyline segments.

2. In the History Tree, right-click the polyline segment you want to edit and choose either:
   - **Delete Start Point**, which removes the preceding segment, or
   - **Delete End Point**, which removes the subsequent segment.

   Alternatively, you can select the segment and choose **Edit** > **Delete Start Point** or **Edit** > **Delete End Point** from the menu bar.

Delete Last Operation

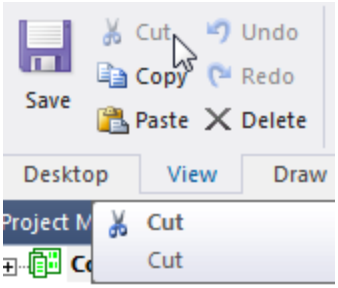
You can delete the last geometry operation performed on an object. For certain situations, this functionality can offer an advantage when compared with the *Undo* command. For example, if you performed several valid operations (such as assigning boundaries, excitations, or mesh operations) after operating on an object's geometry, you could revert the last geometry change without undoing the subsequent assignments.

1. Select the object.
2. From the menu bar, click **Modeler** > **Delete Last Operation**.

   This command undoes the last geometry operation performed on the selected object, including removing that operation from the History Tree and updating the context for the *Undo* and *Redo* commands.

Cutting Objects

1. Select the objects to cut.
2. From the menu bar, click **Edit** > **Cut** or, on any ribbon tab, click the **Cut** icon:



   The objects are copied to the Clipboard and removed from the design. They can subsequently be pasted elsewhere, such as into a different design.

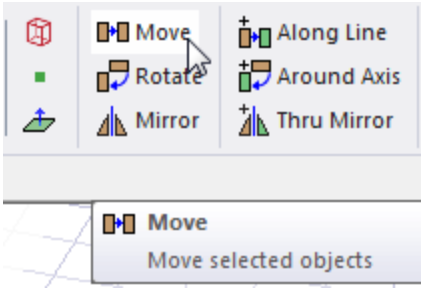
Moving Objects

The process for moving objects differs depending on the Drawing Data Entry Mode for the Drawing Options, whether Point mode (F3) or Dialog mode (F4).

For Point mode (click F3 to change to Point Mode):

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the objects to move.
2. Click **Edit** > **Arrange** > **Move** or, on the **Draw** ribbon tab, click the **Move** icon:



3. Select an arbitrary anchor point in one of the following ways:
   - Click the point. (The *Measure Data* window shows the coordinates of the points you click.
   - Enter the point's coordinates in the **X, Y,** and **Z** boxes.
4. Select a target point in one of the following ways:
   - Click the point.
   - Type the coordinates of a point relative to the anchor point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

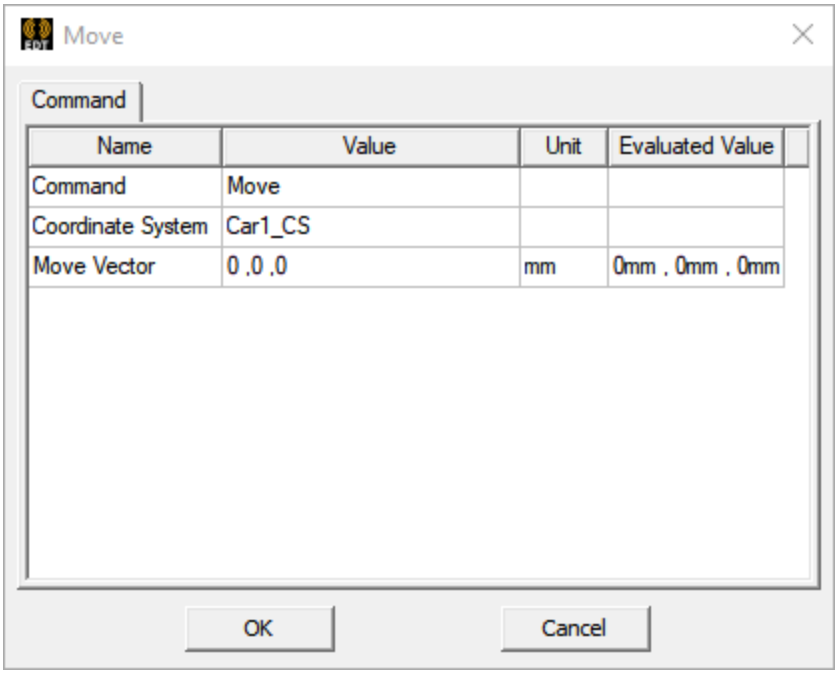
   All selected objects move the distance determined by the offset between the anchor point and the target point.

For Dialog mode (click F4 to change to Dialog mode):

1. Select the objects to move.
2. From the menu bar, click **Edit** > **Arrange** > **Move** or, on the **Drawing** ribbon tab, click the **Move** icon.

   A *Move* command dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



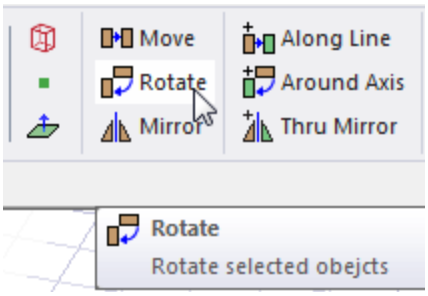
3. Enter the coordinates of the location to which the object should move and click **OK**.

Rotating Objects

Rotate objects about the x-, y-, or z-axis using the **Edit** > **Arrange** > **Rotate** command.

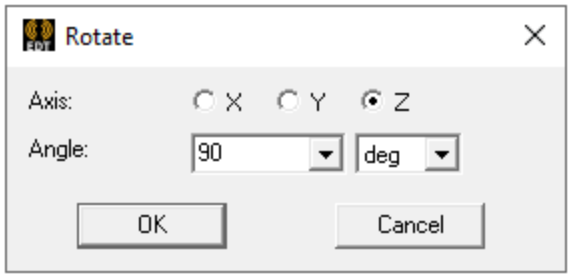
To rotate objects about an axis:

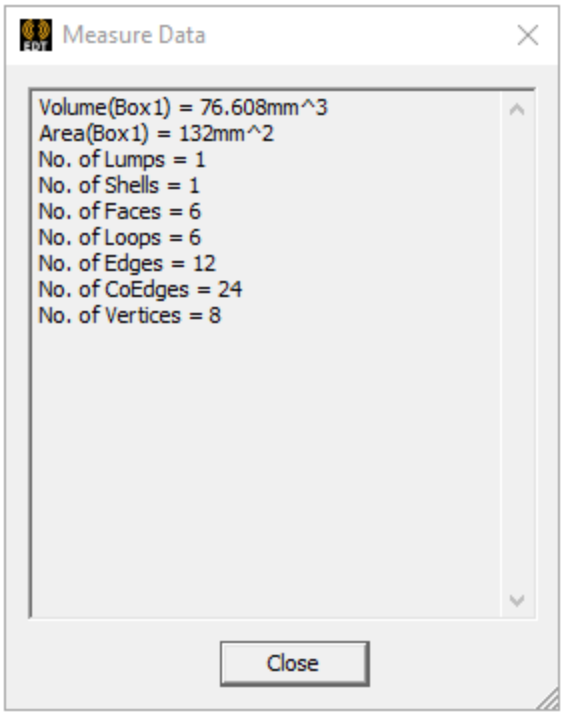
1. Select the objects to rotate.
2. From the menu bar, click **Edit** > **Arrange** > **Rotate** or, on the **Draw** ribbon tab, click the Rotate icon:

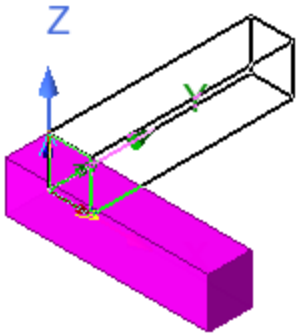


The **Rotate** dialog box appears along with the **Measure Data** window. A visualization of the proposed change appears in the Modeler window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

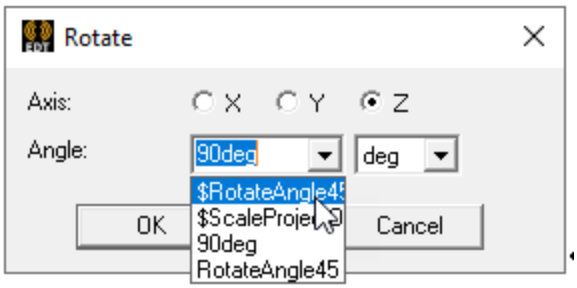






3. Select the axis about which to rotate the objects: **X, Y,** or **Z**.
4. Type the angle to rotate the objects in the **Angle** box. If you have previously specified values, or have created Project or Design Variables, this are listed in the drop-down menus.

   A positive angle causes the object to be rotated in the counter-clockwise direction. A negative angle causes the object to be rotated in the clockwise direction.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click **OK**.

   The selected objects are rotated about the axis.

To *rotate* and *copy* objects, use the **Edit** > **Duplicate** > **Around Axis** command.

Changing the Orientation of an Object

Each object has an **Orientation** property that specifies whether the coordinate system it uses is Global or a user-defined orientation relative to the Global coordinate system.

This property is useful in dealing with anisotropic materials. The properties of anisotropic materials are specified relative to each object's orientation. Changing the orientation of an object provides a way for objects made of the same material to have the material axes orientated differently.

To change an object's orientation:

1. Define the coordinate systems you want to have available.
2. Open the **Properties** dialog box for the object, or select the object and use the docked **Properties** window.
3. Click on the **Orientation** property and select the coordinate system from the drop-down menu. If no orientations other than **Global** have been defined, none appear on the list.
4. Click **OK** to close the dialog box and apply the changes.

> **Note:**
>
> The object will translate and rotate to make the geometry conform to the new coordinate system. If your intention was to change the material orientation, but the object had already been drawn in the correct physical orientation, you will have to redefine it's properties. The coordinates and dimensions used to define the object geometry must be based on the same coordinate system that is needed for proper material orientation. To avoid this problem, it is best to choose the correct coordinate system for material orientation before drawing the object.

Mirroring Objects

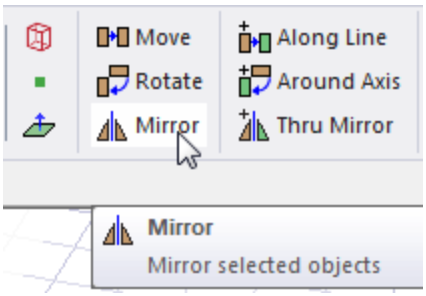
Mirror an object about a plane using the **Edit** > **Arrange** > **Mirror** command. The plane is selected by defining a point on the plane and a normal point. This command allows you to move an object and change its orientation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> The distance between the point on the mirror plane and the point along the normal does not matter — only the vector direction matters.

To mirror an object about a plane:

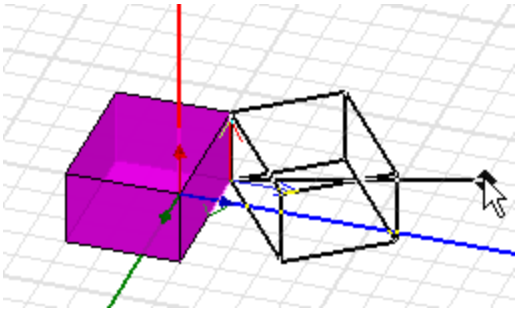
1. Select the object or objects that you want to mirror. You can select multiple objects.
2. From the menu bar, click **Edit** > **Arrange** > **Mirror** or, on the **Draw** ribbon tab, click the **Mirror** icon:



3. Select a point on the plane around which you want to mirror the object.

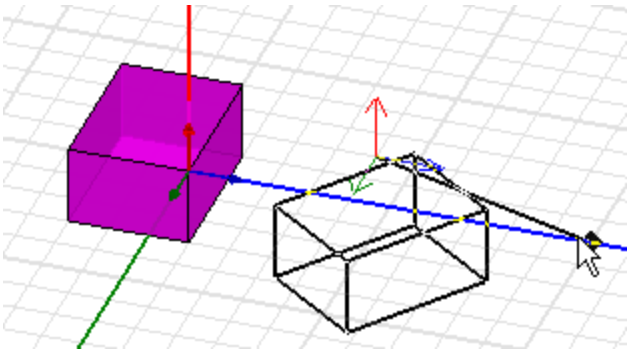
   You can do this by clicking a point, or typing coordinates in the X, Y, and Z boxes in the status bar.

   If you select a point on the object, the mirroring is relative to that point. In the following example, the first point clicked after selecting **Edit** > **Arrange** > ***Mirror*** was on the right-rear bottom corner of the selected object. So the axis of rotation as you move the cursor is that corner. As you move the cursor, it drags a diamond-shape on a vector extending from the initial point. The distance along the vector does not matter. Moving the mouse rotates an outline of the object to new orientations. Clicking the mouse moves the object to location indicated by the outline.



   In this second example, the initial point is at a distance from the original object, designated by the triad from which the handle for rotation extends to the dragging cursor.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Select a second point in one of the following ways:
   - Click the point.
   - Type the coordinates of a point relative to the first point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point. As you type in the values, the outline moves to the coordinates. Press **Enter** to complete the command.

   The selected object is moved to the plane you specified and oriented relative to the normal point you specify.

To *mirror* and *copy* objects about a plane, use the **Edit** > **Duplicate** > **Mirror** command.

Offsetting Objects

Move every face of a 3D object in a direction normal to its surface using the **Edit** > **Arrange** > **Offset** command. The faces are moved a specified distance normal to their original planes. This command enables you to move every face of a solid object without having to individually select and move each face. Use the **Surfaces** > **Move Faces** > **Along Normal** command if you want to move just one or more faces of an object.

To offset every face of an object:

1. Select the object you want to offset.
2. From the menu bar, click **Edit** > **Arrange** > **Offset**.

   The *Offset* dialog box appears.

3. Type the distance you want to move the object faces from their origins, and then select a unit from the pull-down menu.
4. Click **OK**.

   The selected object's faces are moved the distance you specified.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

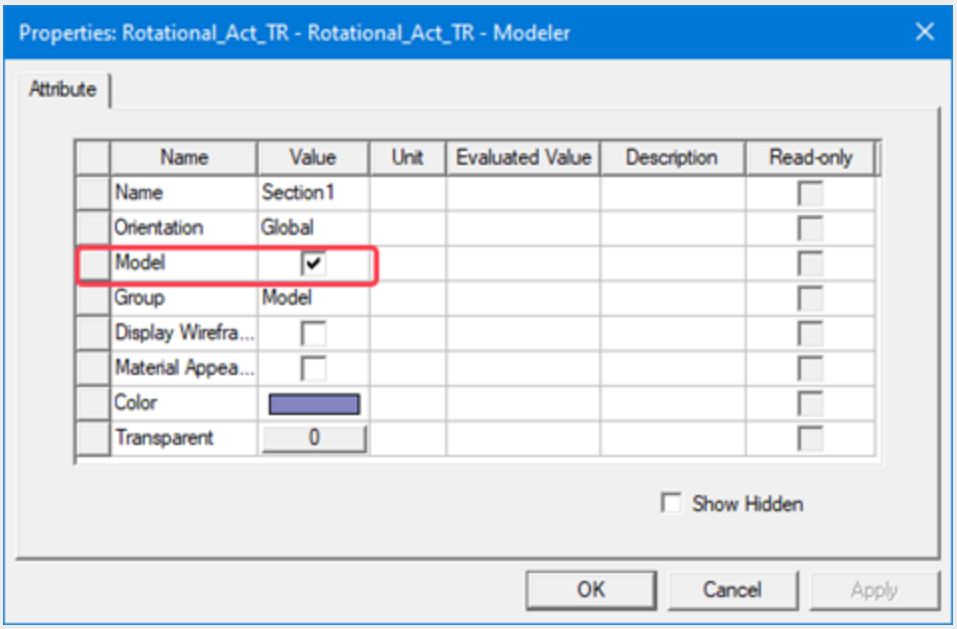
Duplicating Objects

You can duplicate objects within a design using the **Edit** > **Duplicate** commands. Duplicates are dependent upon the parameters of their *parent* object at the time they were created, that is, they share the parent object's history at the time of creation. The command hierarchy in the history tree will show the duplication command, illustrating which commands affect all duplicates (those performed before the duplication) and which commands would not affect the duplicates (those performed after the duplication). For example, if you modify the radius of a parent object's hole, the change is applied to the holes of the object's duplicates because they share the radius specification history, but if you move the faces of the parent object, its duplicates are not affected because this operation took place after the duplicates were created.

Operations performed on duplicates are independent. For example, if you duplicate a cylinder twice, creating a row of three, and then split the second cylinder, the first and third cylinders are not affected by the split.

When creating duplicates, the parent object is duplicated along a line or around an axis the number of times you specify. You can also create a single duplicate that mirrors the parent object about a plane.

Note: When you duplicate an object, the Model\Non-Model status of the new object is not copied from the parent, but is determined by the **Modeler** > **New Object Type** menu selection. You can change the Model\Non-Model status of an existing object from its Properties window:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Choose from the following commands:

Edit > Duplicate > Along Line	Duplicates the parent object along a straight line. The child object can be designated as attached to the parent object, but if so, no ports or boundary conditions are duplicated.
Edit > Duplicate > Around Axis	Duplicates the parent object around an axis. The child object can be designated as attached to the parent object, but if so, no ports or boundary conditions are duplicated.
Edit > Duplicate > Mirror	Duplicates a mirror image of the parent object about a plane.

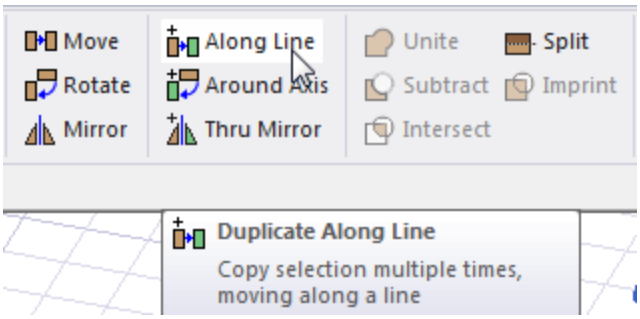
To copy objects to another design, use the **Edit** > **Copy** and **Edit** > **Paste** commands.

> **Note:**
>
> There is currently no method for dissolving the parent/duplicate relationship once a duplicate has been created.

Duplicating Objects Along a Line

To duplicate an object along a straight line, use the **Edit** > **Duplicate** > **Along Line** command. The line along which the object is duplicated can be vertical, horizontal, or lie at an angle.

1. Select the object you want to duplicate.
2. From the menu bar, click **Edit** > **Duplicate** > **Along Line** or, on the **Draw** ribbon tab, click the **Duplicate Along Line** icon:



3. Specify the vector along which the object will be duplicated. The *Measure Data* window appears to help you choose points:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

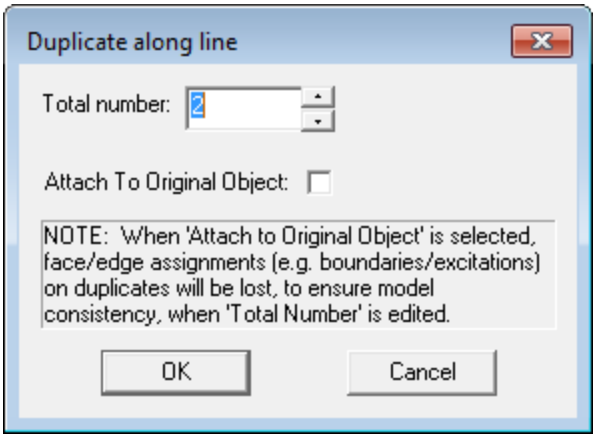
a. Select an arbitrary anchor point in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the in the **X, Y,** and **Z** boxes.

   Any point in the drawing region can be selected; however, selecting an anchor point on the object's edge or within the object makes it easier to select the duplication line.

b. Select a second point in one of the following ways:
   - Click the point.
   - Type the coordinates of a point relative to the anchor point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

   This point defines the direction and distance from the anchor point to duplicate the object.

   The *Duplicate Along Line* dialog box appears:



4. Type the total number of objects, including the original, in the **Total Number** box.
5. Optionally, check the **Attach to Original Object** check box. If this is checked, original object and its duplicates will be combined into one part. No ports or boundary conditions are duplicated for duplicates. If this option is used, it is recommended to run validation to ensure that the original and duplicates are not intersecting or touching each other.
6. Click **OK**.

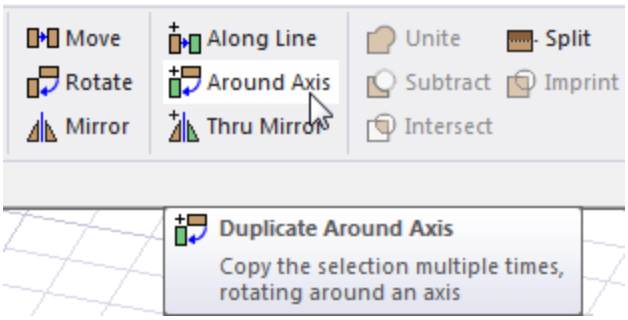
   The duplicates are placed along the vector you specified.

Duplicating Objects Around an Axis

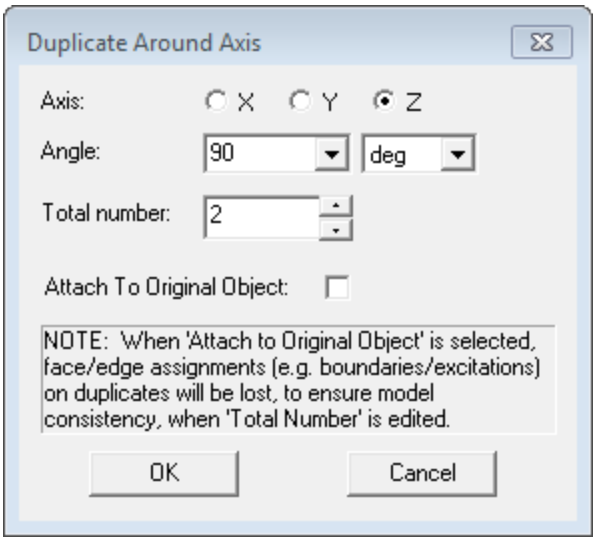
To duplicate an object around the x-, y-, or z-axis, use the **Edit** > **Duplicate** > **Around Axis** command.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the object you want to duplicate.
2. From the menu bar, click **Edit** > **Duplicate** > **Around Axis** or, on the **Draw** ribbon tab, click the **Duplicate Around Axis** icon:



The *Duplicate Around Axis* dialog box appears:



3. Select the axis around which you want to duplicate the object: **X, Y,** or **Z**.
4. Type the angle between duplicates in the **Angle** box.

   A positive angle causes the object to be pasted in the counter-clockwise direction.

   A negative angle causes the object to be pasted in the clockwise direction.

5. Type the total number of objects, including the original, in the **Total Number** box.
6. Optionally, check the **Attach to Original Object** check box. If this is checked, original object and its duplicates will be combined into one part. No ports or boundary conditions are duplicated for duplicates. If this option is used, it is recommended to run validation to ensure that the original and duplicates are not intersecting or touching each other.
7. Click **OK**.

   The object is duplicated around the axis at the angle you specified.

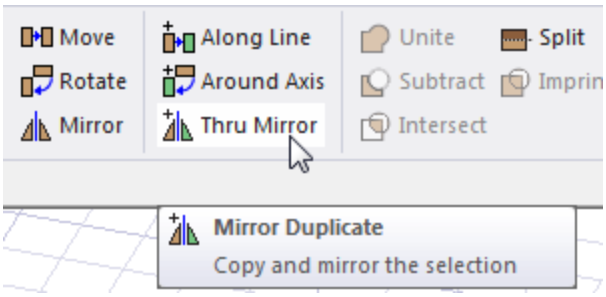
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Duplicating and Mirroring Objects

To duplicate and mirror an object about a plane, use the **Edit** > **Duplicate** > **Mirror** command. The plane is selected by defining a point on the plane and a normal point. This command allows you to duplicate an object and specify the duplicate's position.

This command is similar to **Edit** > **Arrange** > **Mirror**, except that this command duplicates an object, rather than moves it.

1. Select the object you want to mirror.
2. From the menu bar, click **Edit** > **Duplicate** > **Mirror** or, on the **Draw** ribbon tab, click the **Thru Mirror** (Mirror Duplicate) icon:



3. Select a point on the plane on which you want to mirror the object.

   A line drawn from this point to the mirror plane will be perpendicular to the plane. The distance between the point on mirror plane and point along the normal does not matter; only the vector direction matters

4. Select a normal point on the plane in one of the following ways:
   - Click the point.
   - Type the coordinates of a point relative to the first point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

   A duplicate of the object appears on the plane you specified, oriented according to the normal point you specified.

Scaling Objects

Scale an object's dimensions in one or more directions using the **Edit** > **Scale** command.

The scale of an object is determined by the distance of each of its vertices from the origin of the model coordinate system. When an object is scaled, the distance of each vertex from the origin is multiplied by the scaling factor, causing the object to be resized and/or moved.

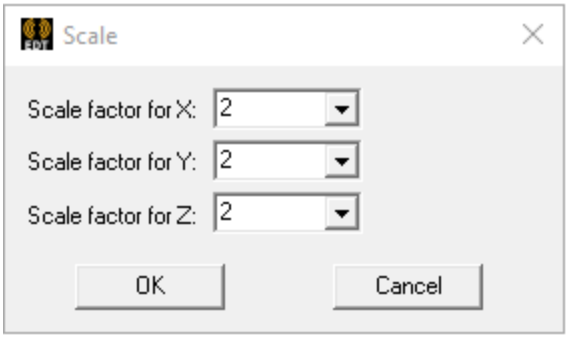
For example, if you specify a scaling factor of 2 in the X direction, each vertex in the model will be moved so that the distance to its origin is doubled. Note that a vertex located at the origin will not move. You can alter an object's proportions by scaling it in one direction.

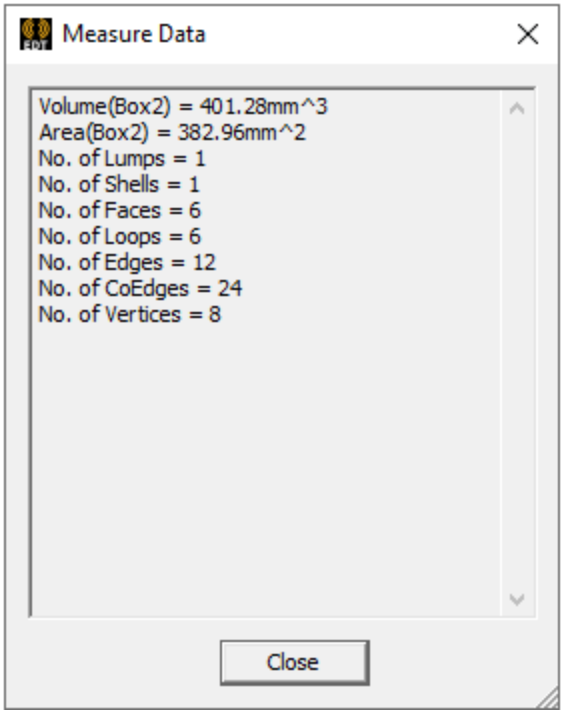
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To scale an object's dimensions in one or more directions:

1. If necessary, set a different working coordinate system to achieve the desired scaling.
2. Select the object to scale.
3. From the menu bar, click **Edit** > **Scale**.

   The **Scale** dialog box appears with the **Measure Data** window.

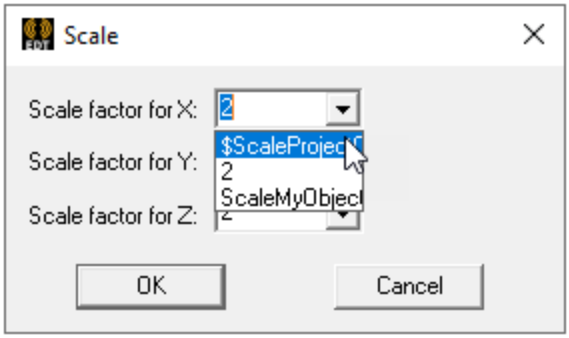




4. Type the **Scale factor** for each axis. If you have created Project or Design Variables, or have previously used a relevant value, these may be listed for selection in the drop-down

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

menu.



5. Click **OK**.

   The object is scaled about the working coordinate system's origin.

Sweeping Objects

You can sweep a 2D object around an axis, along a vector, or along a path to create a 3D solid object. Objects that can be swept include circles, arcs, rectangles, trapezoids, polylines, or any 2D object created in the **3D Modeler** window. The 2D object need not be orthogonal to the sweep path.

You can also:

- Thicken sheets to make a 3D object.
- Sweep open 1D objects, such as polylines, which results in open 2D sheet objects.
- Sweep one or more faces of a 3D object to create a new 3D object. (See Sweep Faces Along Normal).

Sweeping Around an Axis

Sweep a 1D or 2D object around the X, Y, or Z axis using the **Draw** > **Sweep** > **Around Axis** command. Sweeping circles around an axis is a convenient way to create an open coil loop.

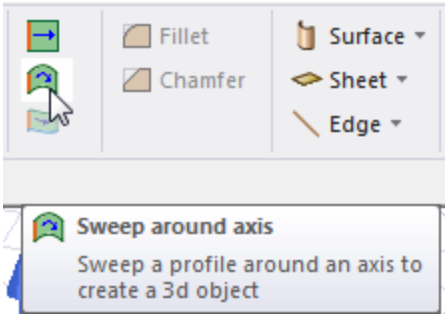
Before using this command, keep the following guidelines in mind:

- The object and the axis you are sweeping around must lie in the same plane. For example, if you are sweeping an object around the Z-axis, the object must lie in a plane that includes the Z-axis, such as XZ or YZ.
- The normal of the object's plane faces must be perpendicular to the axis around which you are sweeping.
- The object may not cross the axis around which it is being swept.

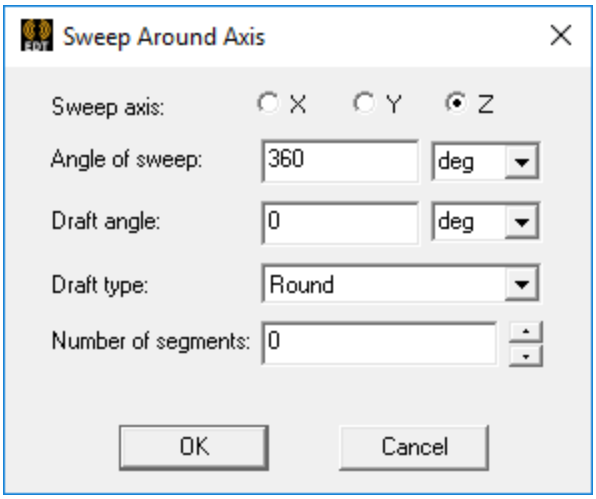
To sweep an object around an axis:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the object you want to sweep.
2. From the menu bar, click **Draw** > **Sweep** > **Around Axis** or, in the **Draw** ribbon tab, click **Sweep around axis**:



The *Sweep Around Axis* dialog box appears.



3. Select the **Sweep axis** (that is, the axis you want to sweep the object around: **X, Y,** or **Z**).
4. Type an angle into the **Angle of sweep** text box and select the desired unit of measure.

   This is the angle about the specified sweep axis through which you want the object to be twisted or rotated as it is swept. The value must be between **-360** and **360** degrees.

5. Type an angle into the **Draft angle** text box and select the desired unit of measure.

   This is the angle to which you want a 2D object's profile, or shape, expanded (positive angle) or contracted (negative angle) as it is swept. Among other purposes, drafts make it easy to remove molded or cast parts from their molds.

   For 1D objects, the draft angle progressively alters the sweep radius but does not alter the length of the object as it is swept. For example, a 10 mm straight line swept around a parallel axis with a nonzero draft angle would produce a spiral surface with a fixed width of 10 mm. When the sweep angle is *positive*, a positive draft angle produces a spiral surface with a *decreasing* radius (that is, the distance from the generated surface to the sweep

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

axis decrease along the sweep angle. A negative angle produces a spiral surface with an *increasing* radius along the sweep angle. When the sweep angle is *negative*, the spiral behavior is the opposite of that just described.

6. Select one of the following **Draft type** options from the drop-down menu. The draft type instructs the modeler whether to maintain the original object shape and how to transition between adjacent faces. This setting affects the results when sweeping 2D objects into 3D solids or when sweeping lines and curves into 3D sheets.
   - **Extended**
   - **Round**
   - **Natural**

   For a complete description of the behavior of each of these three options, see the Draft Types page.

7. Type the number of segments in the **Number of segments** text box and click **OK**.

   > **Note:**
   >
   > The default number of segments is zero, which creates a true path. A positive value results in a segmented sweep, while a negative value results in an error.
   >
   > If the sweep angle is 360 degrees, the number of segments is equal to the value specified. If the sweep angle is less than 360 degrees, half segments appear at the ends.
   >
   > Projects and scripts from previous software versions are treated as if the number of segments were zero.

   The object is swept around the axis. The new object has the properties of the original object. The *Properties* dialog box appears, enabling you to modify the object's properties.

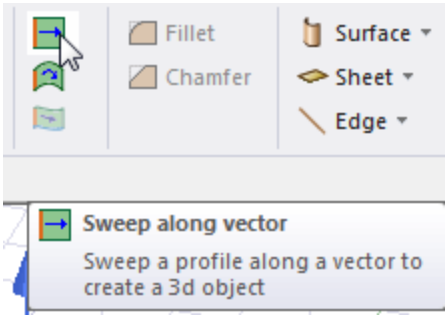
8. Click **OK**.

Sweeping Along a Vector

Sweep a 1D or 2D object along a vector using the **Draw** > **Sweep** > **Along Vector** command.

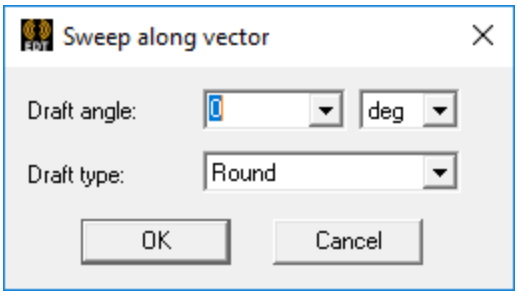
1. Select the object you want to sweep.
2. Click **Draw** > **Sweep** > **Along Vector** or in the **Draw** ribbon tab, click **Sweep along vector**:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Draw the vector you want to sweep the object along:
   a. Select the start point by clicking the point or typing its coordinates in the **X, Y,** and **Z** boxes.
   b. Select the endpoint in one of the following ways:
      - Click the point.
      - Type the coordinates of a point relative to the start point in the **dX, dY,** and **dZ** boxes, where **d** is the relative distance from the previously selected point.

   The *Sweep along vector* dialog box appears.



4. Type an angle into the **Draft angle** text box and select the desired unit of measure.

   This is the angle to which you want a 2D object's profile, or shape, expanded (positive angle) or contracted (negative angle) as it is swept. Among other purposes, drafts make it easy to remove molded or cast parts from their molds.

   For 1D objects, the draft angle alters the sweep direction but does not alter the length of the object as it is swept. For example, a 10 mm straight line swept with a 5-degree draft angle does not produce a trapezoidal 2D object. It produces a rectangle with a constant 10 mm width but tilted 5 degrees relative to the specified sweep vector.

5. Select one of the following **Draft type** options from the drop-down menu. The draft type instructs the modeler whether to maintain the original object shape and how to transition between adjacent faces. This setting affects the results when sweeping 2D objects into 3D solids or when sweeping lines and curves into 3D sheets.
   - **Extended**
   - **Round**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Natural**

For a complete description of the behavior of each of these three options, see the Draft Types page.

6. Click **OK**.

   The object is swept along the vector. The new object has the name and color of the original profile. The *Properties* dialog box appears, enabling you to modify the object's properties.

Sweeping Along a Path

Sweep a 1D or 2D object along a path that is defined by an open or closed polyline using the **Draw** > **Sweep** > **Along Path** command.

Best Practices

Adhere to the following best practices for good sweep outcomes using the Parasolid geometry kernel:

- When sweeping along a path that is not closed (i.e., the start and end point are not the same), ensure that the start point of the path is on the profile being swept.
- When sweeping a profile around a closed path, ensure that the path intersects the profile.
- Parasolid sweep operator preserves the angle between the profile and path. Ensure that the profile is perpendicular to the path if it is supposed to be perpendicular.
- If the sweep profile is rotationally symmetric (e.g., circle, rectangle, square), position the profile so that the path intersects the profile at the center of the profile. If the profile does not have a center, allow the path to go through or start at the vertex of a profile.

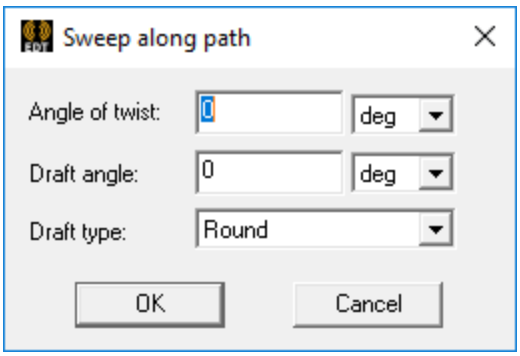
> **Note:**
>
> If the path curve is tangential to the sweep profile, then the sweep will fail.

To sweep an object along a path:

1. Create the polyline to be used as a path.
2. Select the object you want to sweep and then select the new polyline.
3. Click **Draw** > **Sweep** > **Along Path** or, from the **Draw** ribbon tab, click the **Sweep along path** icon:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Sweep along path** dialog box appears.



4. Type an angle in the **Angle of twist** text box and choose the desired unit of measure.

   This is the angle you want the profile to rotate about the sweep path vector as it is swept through the complete path.

5. Type an angle into the **Draft angle** text box and select the desired unit of measure.

   This is the angle to which you want the object's profile, or shape, expanded (positive angle) or contracted (negative angle) as it is swept. Among other purposes, drafts make it easy to remove molded or cast parts from their molds.

   For 1D objects, the draft angle alters the sweep direction but does not alter the length of the object as it is swept. For example, a 10 mm straight line swept with a 5-degree draft angle does not produce a trapezoidal 2D object. It produces a rectangle with a constant 10 mm width but tilted 5 degrees relative to the specified sweep path.

6. Select one of the following **Draft type** options from the drop-down menu. The draft type instructs the modeler whether to maintain the original object shape and how to transition between adjacent faces. This setting affects the results when sweeping 2D objects into 3D solids or when sweeping lines and curves into 3D sheets.
   - **Extended**
   - **Round**
   - **Natural**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For a complete description of the behavior of each of these three options, see: Draft Types.

7. Click **OK**.

   The object is swept along the path. The polyline object used as the path is deleted. The new object has the properties of the original object. The **Properties** dialog box appears, enabling you to modify the object's properties.

Sweeping Faces Along Normal

To create a new object by sweeping select 3D object's face a specified distance in a direction normal to its original plane, use the **Modeler** > **Surface** > **Sweep Faces Along Normal** command. Note that the adjoining faces will not be sheared or bent.

This command is useful for extruding faces, resizing holes, and removing rounded corners.

To sweep selected object faces in a normal direction:

1. Right-click in the Modeler window and choose **Select Faces** from the shortcut menu.

   Alternatively, with the Modeler window active, press **F** to switch to the *Face* selection mode.

2. Select the faces of the object you want to sweep.
3. From the menu bar, click **Modeler** > **Surface** > **Sweep Faces Along Normal**.

   The *Sweep Faces Along Normal* dialog box appears.

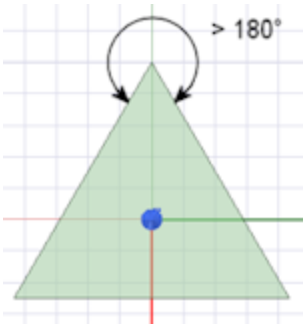
4. Type the distance you want to sweep the object face from its origin.
5. Click **OK**.

   The face is swept the distance you specified to create a new 3D object.

Draft Type Options

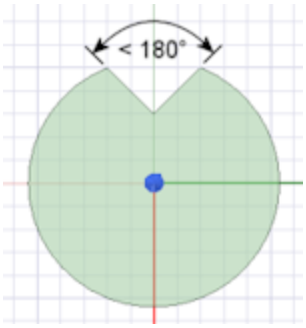
In many cases, the **Draft type** setting may have little to no effect on the resultant geometry of a swept object unless a *positive, non-zero draft angle* is specified. Consider a *sharp corner* feature. For the purpose of this discussion let's define a sharp corner as any protruding feature where the *external* angle between the adjacent faces is greater than 180-degrees:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When this type of feature is expanding, the corner can gradually become rounded. However, if the same corner is contracting, the radius is already zero at the start of the sweep and cannot contract or decrease further. Therefore, no rounding occurs in the latter case.

Now, consider a *groove* in the perimeter of a 2D object to be swept. Let's define a groove as any indented feature, where the *external* angle between the adjacent faces is less than 180-degrees:



A groove will grow in size as the object is swept with a *negative* draft angle specified. In this case, progressive rounding of the corners is possible.

You may need to experiment with the three draft type options to determine what the effect will be for nontrivial object shapes. The behavior of each option can differ according to the shape of the 2D object, the draft angle, and whether it is swept along a vector, a path, or about an axis.

The following three **Draft type** options are available:

- **Natural:** This draft type generally maintains the shape of the original object as it is swept along the specified direction. That is, all sharp corners remain sharp, producing sharp edges in the sweep direction. There are exceptions to this general behavior. Rounding of certain corners may occur if required to accommodate sides with complex shapes (for example, a complex side might be split into a combination of planar, cylindrical, helical, or conic-section faces).
- **Extended:** This option maintains the shape of the original object as it is swept along the specified direction. Sharp corners of the original object remain sharp corners along the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

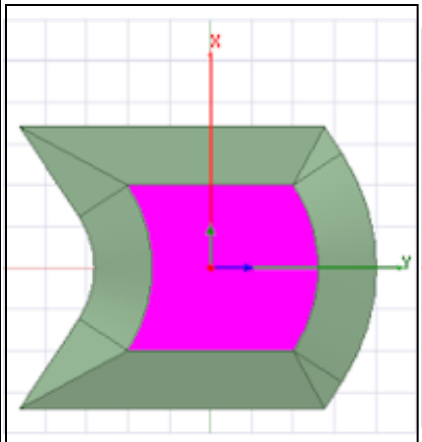
entire swept direction. However, the side faces may be split into three separate tangential faces, as needed and as described in the preceding bullet.

- **Round:** This option rounds all sharp corners of the original object, when possible, as it is swept along the specified direction.

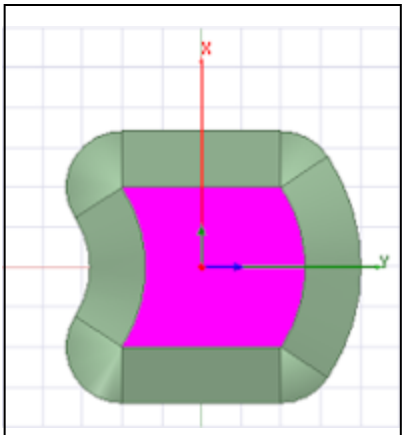
Examples

The images below compare the results of sweeping an object using each of the available **Draft type** options. The first six images are of an object swept with a positive draft angle defined. The final three images are of a different object swept with a negative draft angle defined. For all examples, the original 2D object is highlighted in magenta.

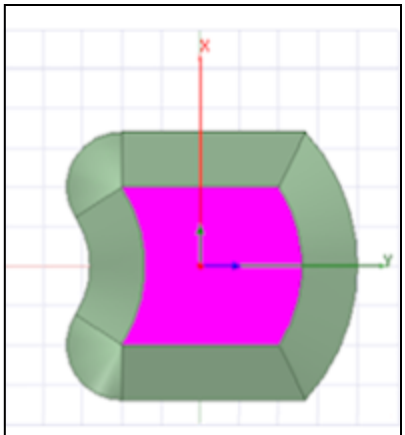
1. **Sweep Along Vector** (Positive Draft Angle):



1a: Extended

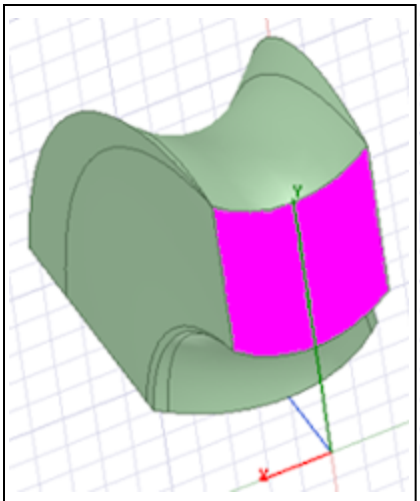


1b: Round

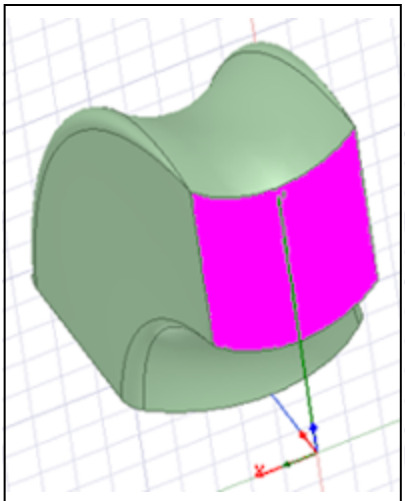


1c: Natural

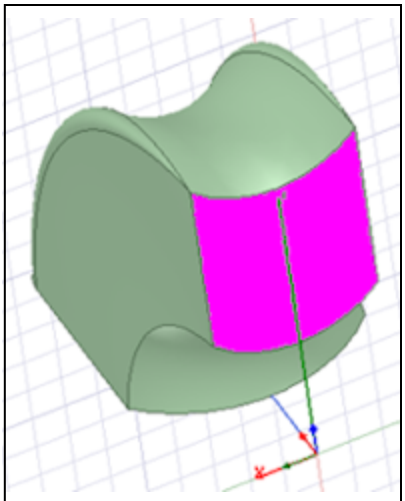
2. **Sweep About Axis** (Positive Draft Angle):



2a: Extended



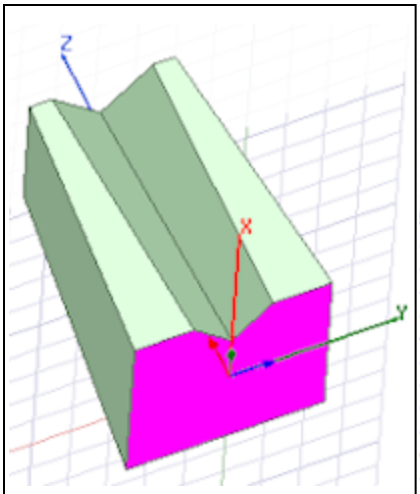
2b: Round



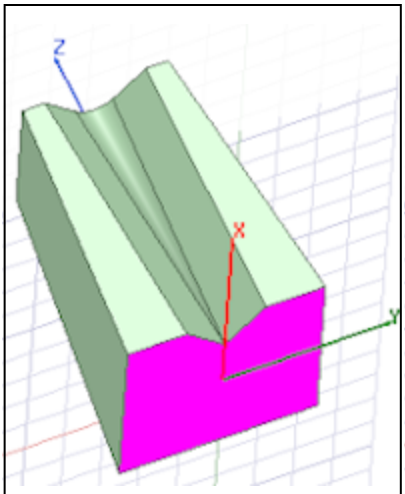
2c: Natural

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

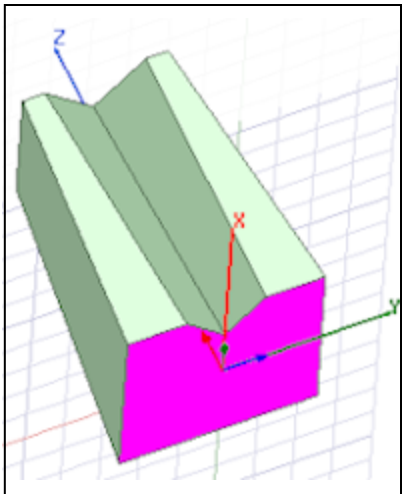
3. **Sweep Along Vector** (Negative Draft Angle):



3a: Extended



3b: Round



3c: Natural

Observations:

- **Extended Option:** Notice how two of the four sides of example **1a** have been divided into three faces each (the sides generated by sweeping curved edges). However, the other two sides (those generated from straight edges) have not been divided.

  In example **2a**, all four sides have been divided into three faces each. Even the straight sides generate a complex surface due to the combination of a positive draft angle and a circular sweep path.

  In example **3a** (negative draft angle), no face is split and no edge is rounded.

- **Round Option:** In both examples **1b** and **2b**, all corners of the original 2D object have been progressively rounded (with the radius increasing from zero at the plane of the original object to a maximum radius at the end of the sweep).

  In example **3b** (negative draft angle), only the corner of the groove has been rounded. The indented feature is the only one increasing is size along the sweep direction.

- **Natural Option:** In both examples **1c** and **2c**, the corners adjacent to the convex curved face remain sharp. However, the corners adjacent to the concave curved face are progressively rounded in both cases (with the radius increasing from zero at the plane of the original object to a maximum radius at the end of the sweep).

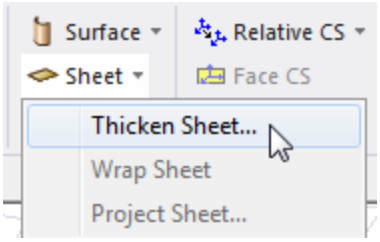
  In example **3c** (negative draft angle), no face is split, and no edge is rounded.

Thicken Sheet

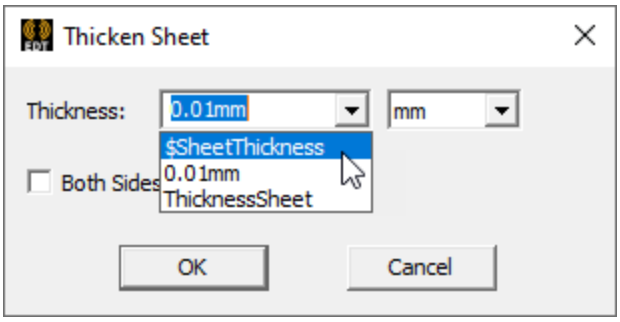
To thicken one or more sheet objects to make 3D objects:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the sheet or sheets you want to thicken into 3D objects.
2. From the menu bar, click **Modeler** > **Surface** > **Thicken Sheet** or, on the **Draw** ribbon tab, choose **Sheet** > **Thicken sheet**:



The *Thicken Sheet* dialog box appears:



3. Specify the **Thickness**. If you have set Project or Design variables with relevant values, or have previously assigned values, this are listed in the dropdown.
4. If necessary, specify the units by selecting from the drop-down menu.
5. If you want to thicken symmetrically (half of the thickness on each side of the sheet or sheets), select the **Both Sides** check box.

   The resulting solid is the requested thickness, but the original sheet location corresponds to the middle of the resulting solid's thickness.

6. Click **OK**.

   The dialog closes and the sheets are changed into 3D objects of the desired thickness.

Wrap Sheet Command

You can use **Modeler** > **Surface** > **Wrap Sheet** command to wrap a sheet object around a suitable 3D object. The sheet object does not have to be in contact with the 3D object. It should have smaller dimensions than the 3D object.

When the sheet to be wrapped and the target body do not touch, the wrap sheet command chooses among several different methods of bringing them into contact.

- If an edge from the sheet body and the target body are close enough and at a small angle (10 degrees), the edges are brought into alignment before the wrap.

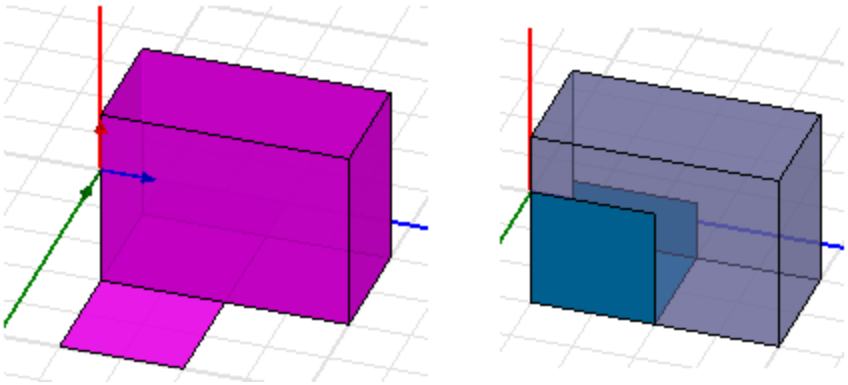
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If a vertex from the sheet body and an edge on the target body are close enough, the vertex is moved to the target edge.
- If the previous criterion are not matched, a vertex from the sheet body is moved to the closest face of the target body.

If the sheet to be wrapped and the target face are both planar and have the same or opposite normal, the closest vertex from the sheet body is moved to the target face. No moving to match or align edges is performed.

If the sheet body has multiple lumps, a single rectangle is fitted around all of the lumps. The rectangle is moved to the target surface using the above procedure.

If the sheet object does not overlap the corners of the 3D object, the wrap is straightforward, as shown in the following figure:



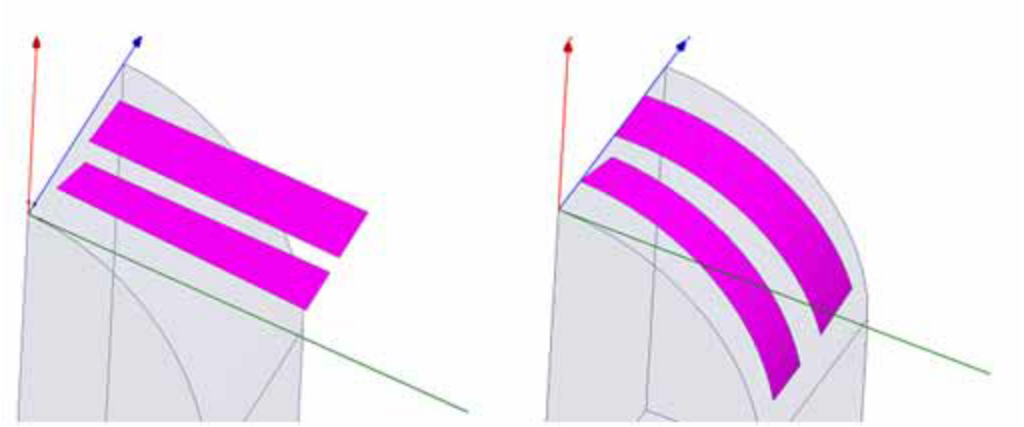
If you delete the 3D object, the wrapped sheet retains the form it took when wrapped.

A sheet object that overlaps corners may not wrap in straightforward fashion, depending on both the angle(s) involved, and the sheet object. While it is possible, it is not recommended.

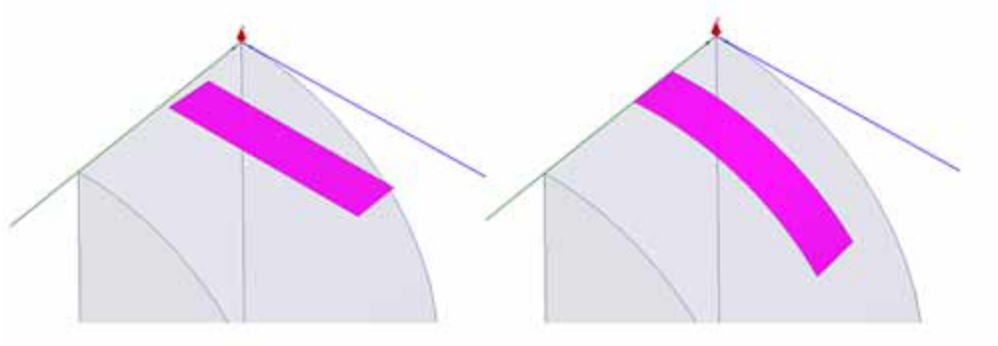
Examples:

The following figure illustrates wrapping on a target surface with the snap to an edge. The surface will snap to an edge if the distance is close relative to the length of the edge. If the angle between the edge is less than 10 degrees, the edge to be wrapped will be aligned to the target body. In the example on the left, the lower sheet has an edge angle more than 10 degrees. The vertex will be snapped to the edge. The top sheet in the picture has an edge angle of less than 10 degrees. The vertex is snapped to the target edge and the sheet edge is aligned to the target. To avoid snapping to an edge or edge alignment, the wrap sheet can be put into contact with the target surface. In that case the sheet vertex will not be moved to the target edge unless it is within modeling tolerances.

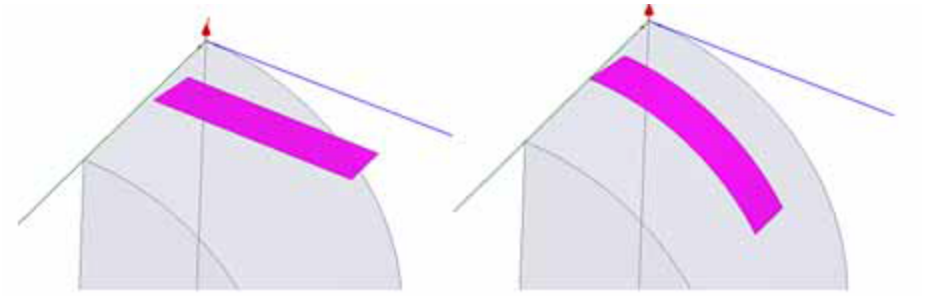
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following figure shows the situation when you move the edge to the edge before wrapping.

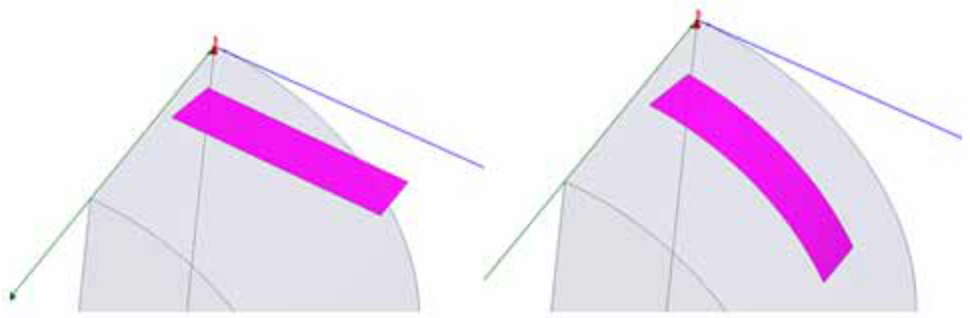


The following figure shows the situation when you move a vertex to an edge before wrapping.

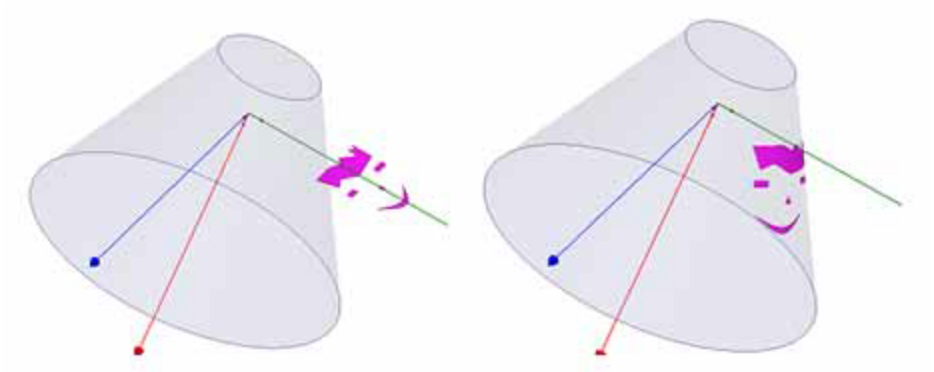


The following figure shows the situation where you move a vertex to a face before wrapping.

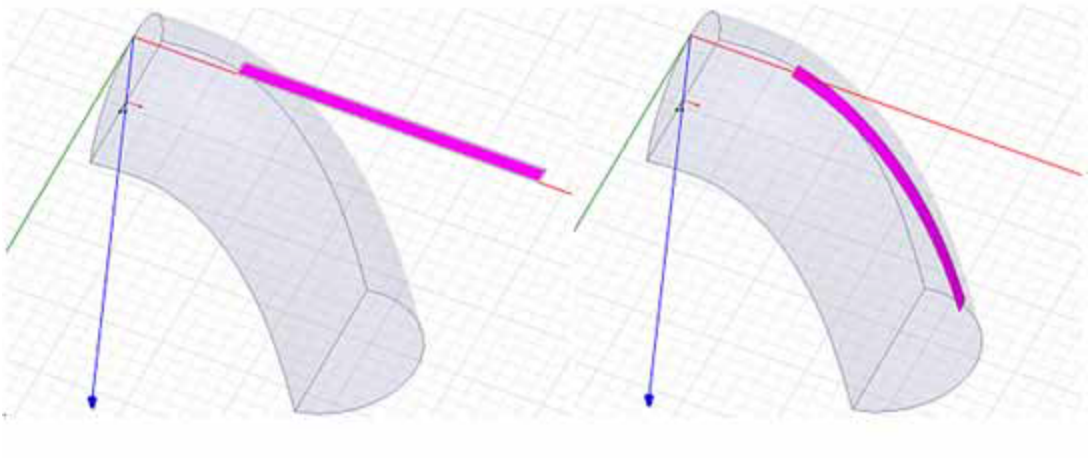
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following figure shows the situation when you move a multilump sheet to touch the target body. A rectangle that encloses all lumps is used to position the sheet on the target.

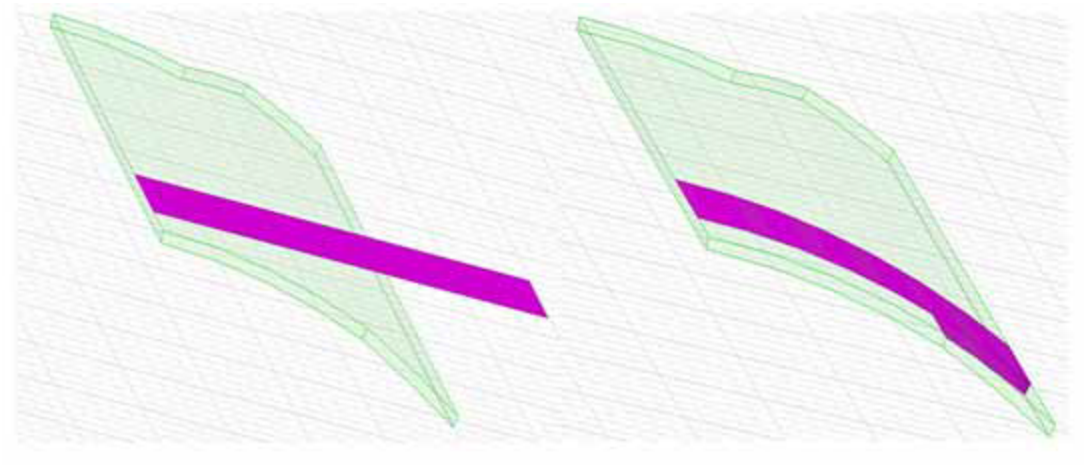


The next figure illustrates wrapping to a non-developable surface. The area of the sheet changed from 0.7000 mm^2 before wrapping to 0.6986 mm^2 after wrapping.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following figure illustrates wrapping on a target surface which has tolerant edges. The tolerant edge is at the beginning of the wrap. The tolerant edge is not evident visually.



Limitations/Error Conditions:

- Wrapping on non-developable surfaces creates distortion in lengths.
- Wrapping is not allowed across curved boundaries between faces.
- Working with tolerant edges is successful only if the gaps can be tightened sufficiently for the operation.
- Wrapping on the pole of a surface (such as sphere or cone) is not supported. A warning is given:

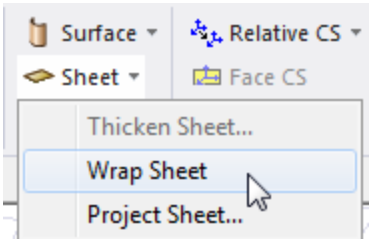
  WrapSheet: Wrapping sheet contacted pole of target surface. Recommend moving sheet and splitting out pole from target surface.

- The choice of method for how the sheet body is moved to touch the target face is not directly selectable by the user. If the sheet to be wrapped and target body touch, then the sheet body is only rotated at point of contact.

To wrap a sheet object:

1. Create a sheet close to an appropriate 3D object.
2. Select both objects.
3. From the menu bar, click **Modeler** > **Surface** > **Wrap Sheet** or, on the **Draw** ribbon tab, select **Sheet** > **Rap Sheet**:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The sheet object wraps around the 3D object. You can select the wrapped sheet object and the 3D object separately, and assign properties separately.

If the object cannot wrap, the Message window contains a warning and description.

You can wrap multiple sheets on the same 3D object.

If you delete the 3D object, the wrapped sheet retains the form it took when wrapped.

You can use the **Tools** > **Options** > **General Options**, 3D Modeler options to automatically perform a **Modeler** > **Boolean** > **Imprint** command after performing a **Wrap.**

Covering Lines

To cover a closed 1D polyline object with a face, use the **Modeler** > **Surface** > **Cover Lines** command. The polyline object becomes a 2D sheet object.

To convert a closed polyline object to a sheet object:

1. Select the closed polyline object you want to cover.
2. From the menu bar, click **Modeler** > **Surface** > **Cover Lines**.

   The object is now covered. It is now a 2D sheet object that can be swept to form a 3D solid object.

> **Note:**
>
> If you want the modeler to automatically cover all closed polyline objects you draw, including circles, ellipses, rectangles, and regular polygons, select the **Automatically cover closed polylines** option in the *3D Modeler > Drawing* section of the *General Options*. A closed polyline object can also be created by using boolean unite operations on two or more polylines.

Covering Faces

To cover object faces, the faces must be united into a 3D sheet object. To cover the face of a 2D or 3D object, use the **Modeler** > **Surface** > **Cover Faces** command.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Covering the face of an open 2D sheet object that had previously been uncovered results in a 3D solid object. For example, for a box, when you select and uncover a face, the solid box becomes a sheet with five faces. When you then select that sheet body box and use the **Cover Faces** command, the box becomes a solid again with six faces.

To cover the faces of objects:

1. Select the faces of the objects you want to cover.
2. From the menu bar, click **Modeler** > **Surface** > **Cover Faces**.

   The object faces are now covered.

Uncovering Faces

Uncover a surface of a 3D object using the **Modeler** > **Surface** > **Uncover Faces** command. Uncovering the surface of a 3D solid object results in an open 2D sheet object.

To uncover the face of a 3D object:

1. Switch to the *Face* selection mode: Click **Edit** > **Selection Mode** > **Faces**.
2. Select a face of the object you want to uncover.

   From the menu bar, click **Modeler** > **Surface** > **Uncover Faces**. The selected face is uncovered, leaving an open face on the object.

> **Note:**
>
> You can uncover one face of a 3D object at a time. If you select multiple faces, only the first face will be uncovered.

Detaching Faces

The **Modeler** > **Surface** > **Detach Faces** command enables you to remove the face of a 3D object, resulting in two separate objects.

To detach the face of an object:

1. Switch to *Face* selection mode: Click **Edit** > **Selection Mode** > **Faces**.
2. Select the face of the object you want to detach. You can select multiple faces to detach.

   Click **Modeler** > **Surface** > **Detach Faces**. The selected face is now detached, resulting in two 2D sheet objects.

Detaching Edges

The **Modeler** > **Edge** > **Detach Edges** command enables you to remove an edge of a wire object, resulting in two separate wire objects.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To detach an edge of an object:

1. Switch to *Edge* selection mode: Click **Edit** > **Selection Mode** > **Edges**.
2. Select the edge of the object you want to detach. You can select multiple edges to detach.
3. Click **Modeler** > **Edge** > **Detach Edges**.

   The selected edge is now detached, resulting in multiple wire objects.

> **Note:**
>
> Only edges from wire bodies can be used in a detach edge operation.

Creating a Cross-Section

You can take a cross section of a 3D object to create a new 2D object. This is done using the **Modeler** > **Surface** > **Section** command.

Use this command to create cross sections of 3D objects on the XY, YZ, or XZ plane. The cross sections are created as 2D closed polyline objects.

To create a cross section of an object:

1. Make sure the working coordinate system you want to use for the cross section plane is set.
2. Select the object from which you want to create a cross section.
3. From the menu bar, click **Modeler** > **Surface** > **Section**.

   The *Section* dialog box appears.

4. Select the **Section Plane** you will use to divide the object: **XY, YZ,** or **XZ**.
5. Click **OK**.

   A closed polyline object is created at the intersection of the 3D object and the specified section plane. The original 3D object is unmodified.

Connecting Objects

Use the **Modeler** > **Surface** > **Connect** command to perform the following operations:

- Connect two or more 1D polyline objects. Ansys Electronics Desktop will modify the first polyline you select to be a 2D sheet object that connects to the second and any subsequently selected polylines. The second and subsequent polylines selected are deleted.
- Connect two or more 2D sheet objects. Ansys Electronics Desktop will modify the first 2D object you select to be a 3D solid object that connects to the second and any subsequently selected objects. The second and subsequent objects selected are deleted.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

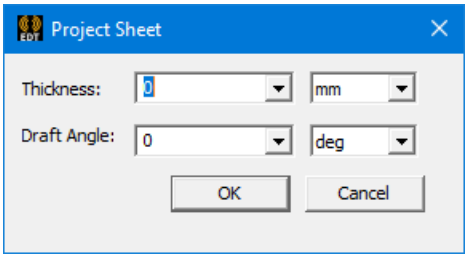
To connect objects:

1. Select the objects you want to connect. Select 3D objects as objects, not as faces.
2. Click **Modeler** > **Surface** > **Connect**.

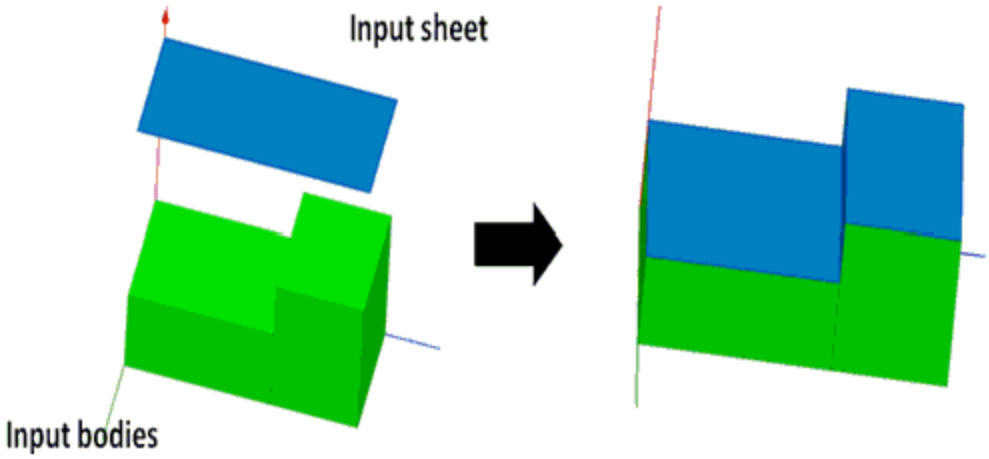
   A new object is created that connects the objects you selected. The first object you selected was modified to create the new object and all subsequently selected objects were deleted.

Project Sheet Object

Use the **Modeler** > **Surface** > **Project Sheet** command to apply a selected planar sheet object to a suitable 3D object. This permits easy modeling of thin conformal deposits. (The **Wrap Sheet** and **Imprint Projection** commands are not suitable for this use.) For modeling conformal deposits when using **Project Sheet**, you can specify a thickness and a draft angle.

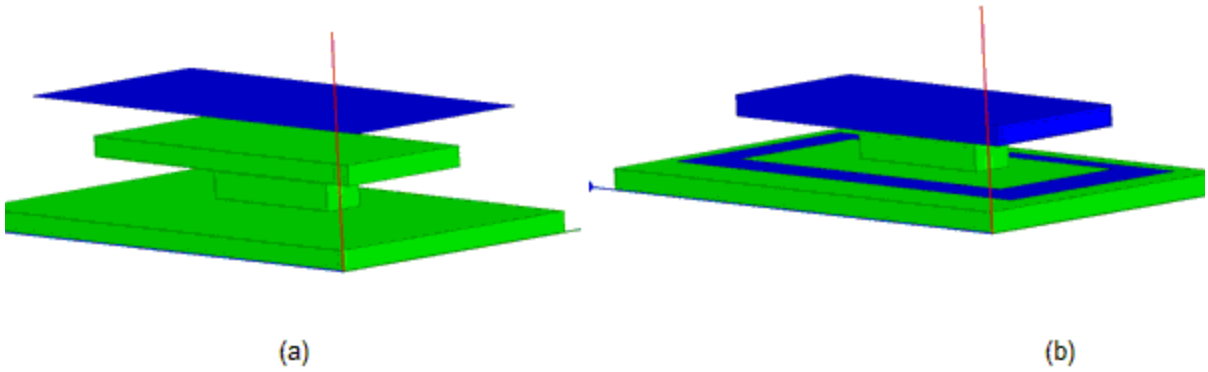


To use the **Project Sheet** command you must select one planar sheet and at least one solid body. If you select multiple solids, the sheet is project on all bodies as if they have been united. After the commend executes, a new sheet that lies on the surface of the selected bodies is created. The solid bodies are not changed.



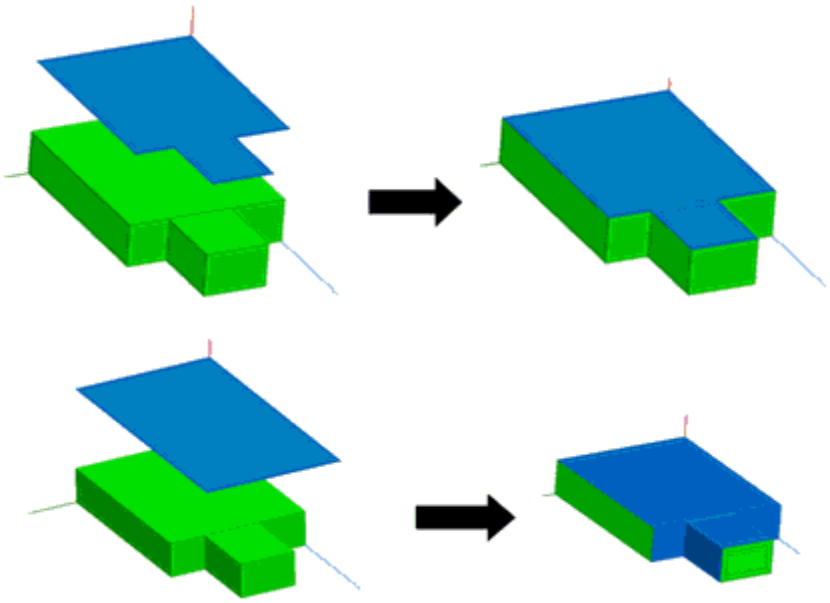
Faces that are hidden from the sheet being projected are not covered, due to the projected sheet.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



(a) (b)

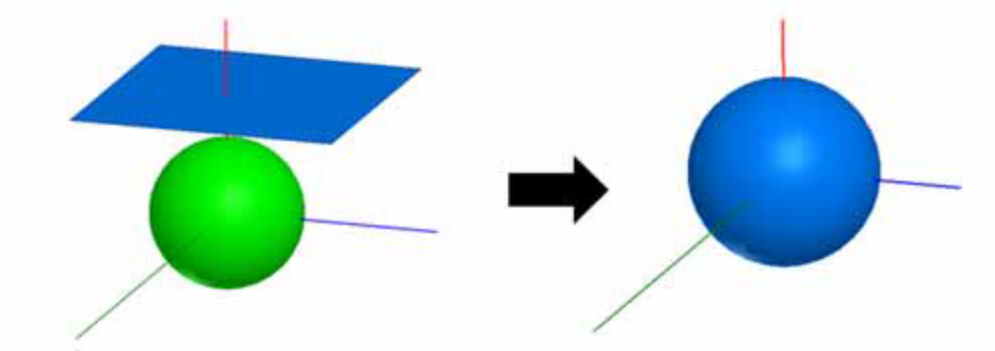
Only vertical faces (or faces parallel to the projection direction) that lie completely within the projection of the sheet will be covered by the projected sheet. Vertical faces that lie on the boundary of the projection are not covered.



If the sheet cross-section exactly matches the body cross-section, no vertical faces are covered. If the projected sheet exceeds the cross-section of solids, faces that lie with in the sheet projection are covered.

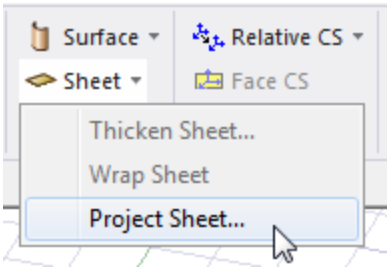
If a surface like a cylinder or sphere lies completely within the sheet projection, the entire surface will be covered, though there may be a warning that “Output may be incorrect because a self-obstructing face was found.”

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To Project a Sheet:

1. Select a planar sheet object and one or more appropriate solid objects.
2. From the menu bar, click **Modeler** > **Surface** > **Project Sheet**, or on the **Draw** ribbon tab, click **Sheet** > **Project Sheet**:

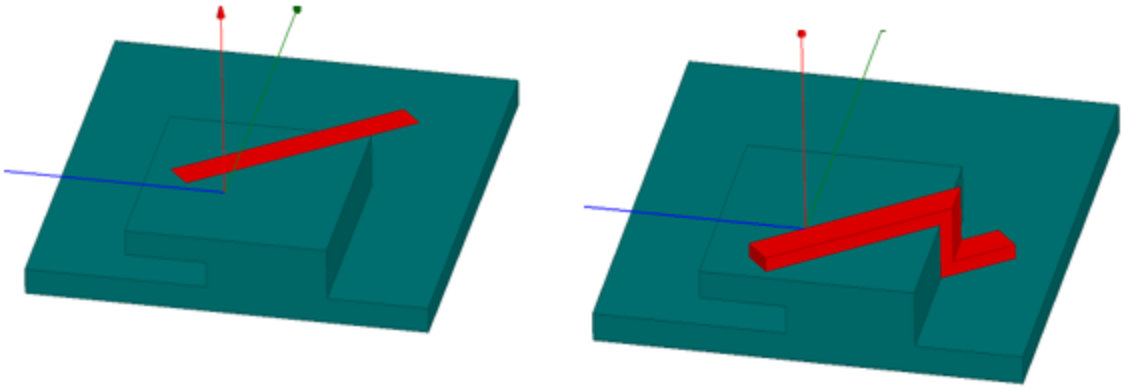


   For modeling conformal deposits, you can also specify a thickness and optional draft angle.

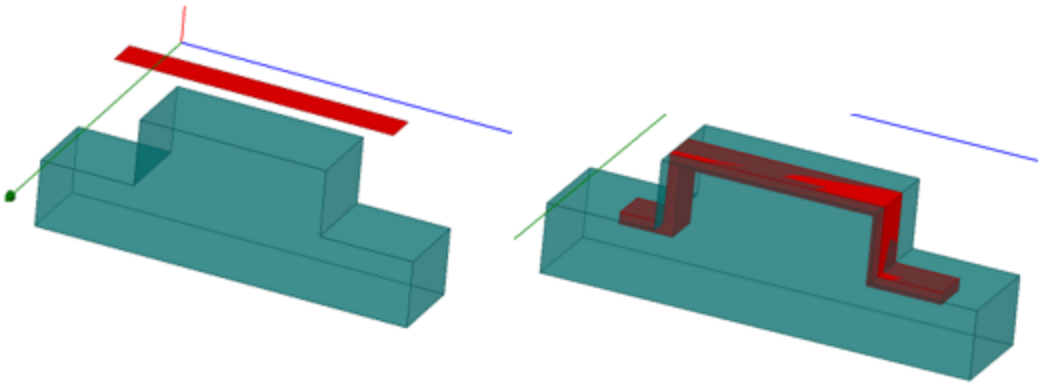
After you perform **Project Sheet**, the History Tree shows the **Project Sheet** command and the create command for the imprinted object. If you select the **Project Sheet** command in the History Tree, you can suppress the command via the docked *Properties* window. If you select the Create <object> icon for the object, you can edit the properties of that object. The changes applied to the object carry over to the imprinting.

Specifying a Thickness value has different effects depending on whether the value is positive or negative. The following figure shows the effect of a positive thickness, with the Projected sheet taking its thickness outward from the target object.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

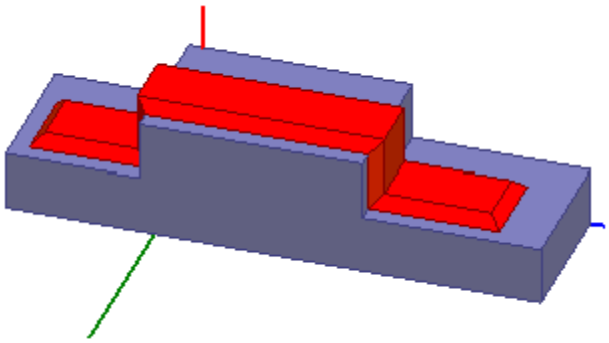


The following figure shows the effect of a negative value for thickness.



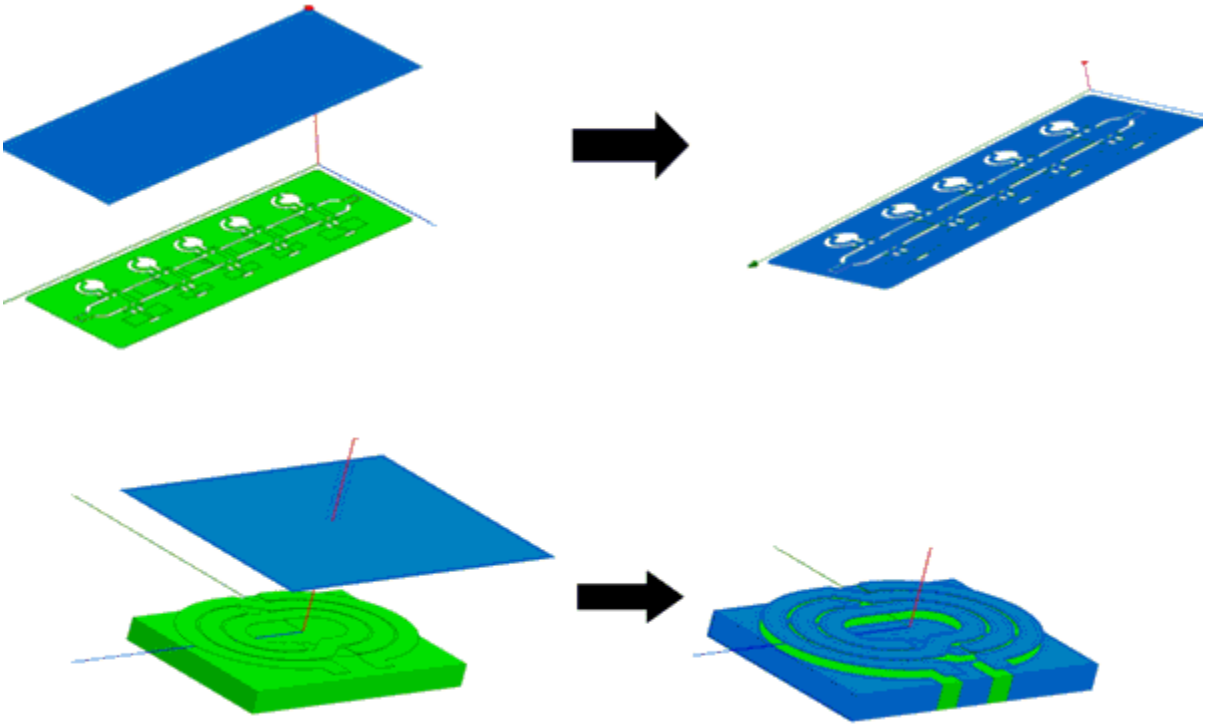
When a negative value is given the thickness will be limited by the target solid body. That is, the thickness can not be beyond the original solid body which was chosen. If you use a negative thickness it is also possible that not all overhangs will be eliminated.

The following figure shows the effect of a 45 degree draft angle.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Other examples of Project Sheet on complex models are show in the following figure.



Moving Faces or Edges

You can move the faces of a 3D object in a normal direction using the **Modeler** > **Surface** > **Move Faces** commands. Moving object faces enables you to resize, reshape, or relocate an object.

Moving Faces Along the Normal

To move a 3D object's face a specified distance in a direction normal to its original plane, use the **Modeler** > **Surface** > **Move Faces** > **Along Normal** command. The faces that adjoin the original face are extended or shortened along their own planes to meet the new face. Note that the adjoining faces will not be sheared or bent.

This command is useful for extruding faces, resizing holes, and removing rounded corners, as shown below.

To move an object face in a normal direction:

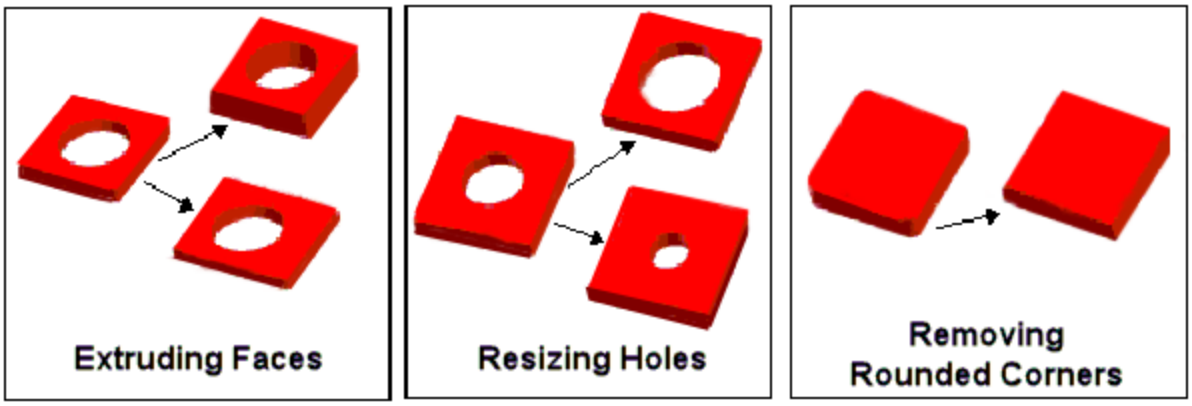
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Right-click in the Modeler window and click **Select Faces** on the shortcut menu.
2. Select the face of the object you want to move.

   If you have created a suitable face selection, right-click the selection and click **Select Assignment** from the shortcut menu. You can operate on faces in the selection.
3. Click **Modeler** > **Surface** > **Move Faces** > **Along Normal**.

   The *Move Faces Along Normal* dialog box appears.
4. Type the distance you want to move the object face.
5. Click **OK**.

   The face will be moved the distance you specified.



To move every face of an object normal to its surface, use the **Edit** > **Arrange** > **Offset** command.

Moving Faces Along a Vector

To move the faces of a 3D object a specified distance along a vector use the **Modeler** > **Surface** > **Move Faces** > **Along Vector** command. Each selected face is moved along the vector, normal to its original plane. The faces that adjoin the original face are extended or shortened along their own planes to meet the new face. Note that the adjoining faces will not be sheared or bent.

This command is useful for relocating holes in an object, as shown below.

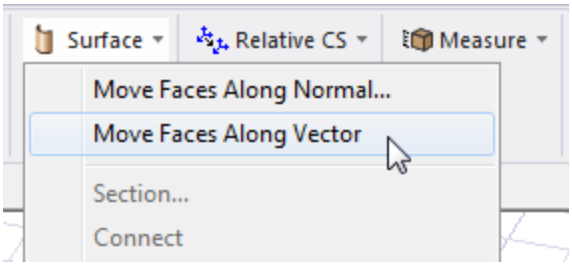
To move an object face along a vector:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Select Faces** on the shortcut menu.
2. Select the face of the object you want to move.

   If you have created a suitable face selection, right-click the list and click **Select Assignment** from the shortcut menu. You can operate on faces in the list.

3. From the menu bar, click **Modeler** > **Surface** > **Move Faces** > **Along Vector**, or, on the **Draw** ribbon tab, click **Surface** > **Move Faces Along Vector**:



4. Specify the vector along which the face will be moved:
   a. Select an arbitrary anchor point in one of the following ways:
      - Click the point.
      - Type the point's coordinates in the **X, Y,** and **Z** boxes.

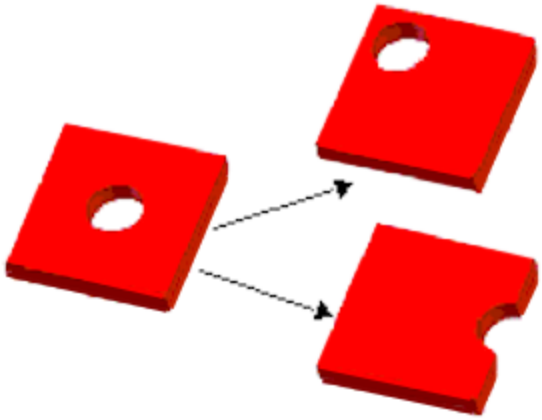
      Any point in the drawing region can be selected; however, selecting an anchor point on the object's edge or within the object makes it easier to select the vector.

   b. Select a second point in one of the following ways:
      - Click the point.
      - Type the coordinates of a point relative to the anchor point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

      This point defines the direction and distance from the anchor point to move the face.

      The face is moved along the vector you specified.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Relocating Holes

To move every face of an object normal to its surface, use the **Edit** > **Arrange** > **Offset** command.

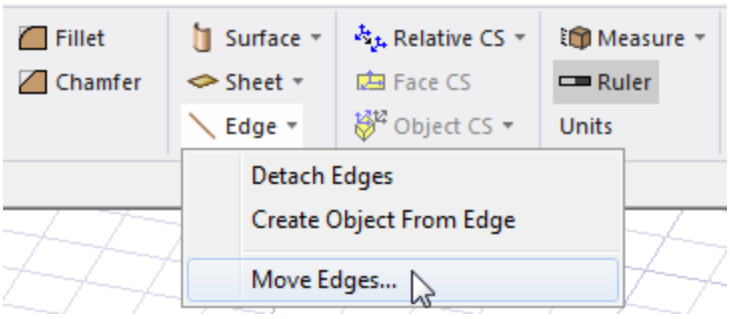
Moving Edges Along Normal

To move a 2D object's edge a specified distance in a direction normal to its original plane, use the **Modeler** > **Edge** > **Move Edge** command. The edge is extended or shortened along its own plane. Note that the adjoining faces will not be sheared or bent. The edge can be on a rectangle, an ellipse, a circle, a regular polygon, or an equation based surface.

This command is useful for extending or shrinking faces and resizing holes.

To move an object edge in a normal direction:

1. Right-click in the Modeler window and choose **Select Edge** on the shortcut menu.
2. Select the edge of the object you want to move.
3. From the menu bar, click **Modeler** > **Edge** > **Move Edge** or, on the **Draw** ribbon tab, click **Edge** > **Move Edges**:



The *Move Faces Along Normal* dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

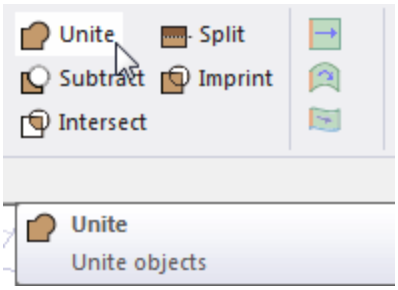
4. Type the distance you want to move the selected edge.
5. Click **OK**.

   The edge of the object is moved based on the value you specified.

Uniting Objects

To join two or more objects into one object, use the **Modeler** > **Boolean** > **Unite** command. The new object has the name, color, boundary, and material assignment of the first object selected. The objects are united at the point of intersection.

1. Select the objects you want to join.
2. From the menu bar, click **Modeler** > **Boolean** > **Unite** or, on the **Draw** ribbon tab, click the **Unite** icon:



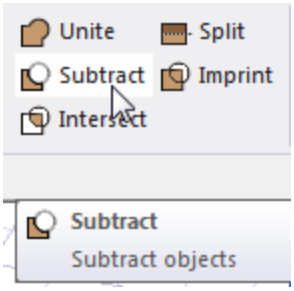
   The objects are united.

> **Note:**
>
> By default, the objects being joined to the first object selected are *not* preserved for later use. If you want to keep a copy of the objects being joined to the first object selected, do one of the following:
>
> - Copy the objects, and then paste them back into the design after uniting them.
> - Select **Clone tool objects before uniting** in the *3D Modeler* > *Operation* section of the *General Options*. This option instructs the modeler to always keep a copy of the original objects being joined.

Subtracting Objects

1. Select the object from which you want to subtract other objects.
2. Hold down the **Ctrl** key and select the objects you want to subtract.
3. From the menu bar, click **Modeler** > **Boolean** > **Subtract**, or on the **Draw** ribbon tab, click the **Subtract** icon:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The *Subtract* dialog box appears.

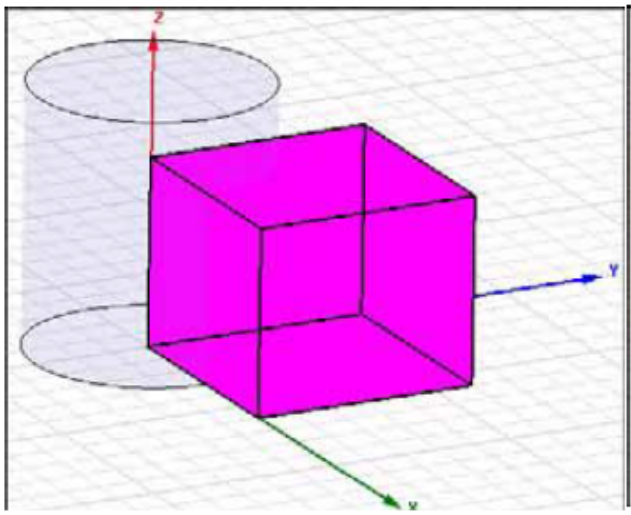
Objects listed in the *Tool Parts* list will be subtracted from the object or objects listed in the *Blank Parts* list.

4. Optionally, select an object name in either list and use the left and right arrow buttons to move the object name to the opposite list.

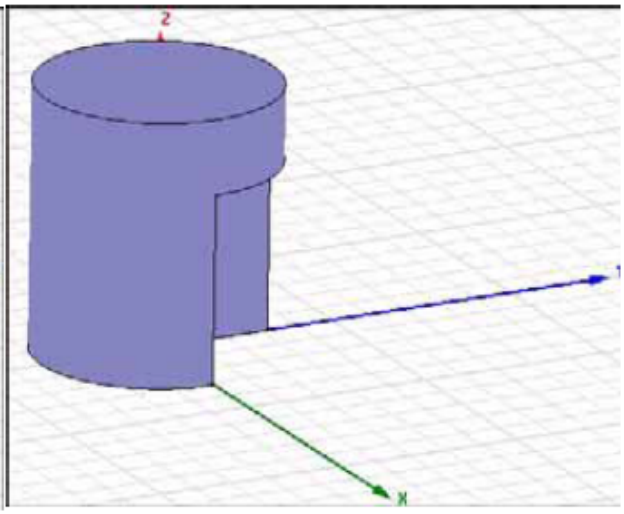
   Though you can type the name of objects into either list, you cannot access the *Subtract* command unless you first select two or more model objects. Therefore, the lists will initially be populated with the selected objects.

5. Optionally, select **Clone tool objects before subtracting**. This option instructs Ansys Electronics Desktop to keep a copy of the original objects being subtracted.
6. Click **OK**.

   The revised *Blank Parts* retain their original name, color, and material.



An intersecting box and cylinder



A box subtracted from a cylinder (the cylinder was selected first)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

By default, the *Tool Parts* (that is, the objects being subtracted from the *Blank Parts*) are *not* preserved for later use. If you want to keep a copy of the objects being subtracted from the blank parts, do one of the following:

- In the *Subtract* dialog box, select **Clone tool objects before subtracting**, as stated in step 5 above. This action is a one-time override of the defdefault behavior.
- In the *3D Modeler* > *Operation* section of the *General Options*, select **Clone tool objects before subtracting**. This option instructs Ansys Electronics Desktop to always (that is, by default) keep a copy of the original objects being subtracted.

Creating Objects from Intersections

To create a new object from the intersection of two or more objects, use the **Modeler** > **Boolean** > **Intersect** command.

To create an object from an intersection:

1. Select the objects from which you want to take the intersection.

   Warning:

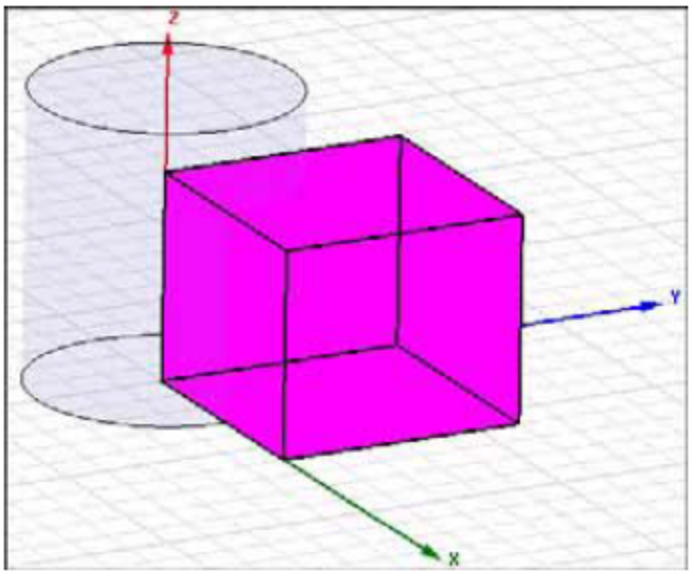
   If the objects you selected do not overlap, the result is a null object, and both objects vanish. When three or more objects have been selected, an intersection is created only if a common volume exists in which all the selected objects overlap. Otherwise, a null object is created and all original objects vanish.

   For example, assume that Part 1 overlaps Part 2, and Part 2 overlaps Part 3, but all three parts do not overlap in a common volume. This situation results in no intersection being created, and all three parts vanishing.

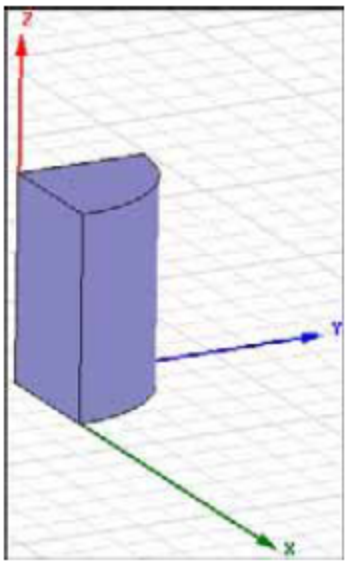
2. From the menu bar, click **Modeler** > **Boolean** > **Intersect**. (You can also access this command from the **Draw** ribbon tab.)

   The original objects vanish, leaving only the new object that was formed from their intersection.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



An intersecting box and cylinder.



Object formed from the intersection of the box and cylinder.

Note:

By default, the original intersecting objects are *not* preserved for later use. If you want to keep a copy of the objects that intersect the first object selected, do one of the following:

- Copy the objects, and then paste them back into the design after creating the new object from the intersection.
- Select **Clone tool objects before intersecting** in the *3D Modeler*> *Operation* section of the *General Options*. This option instructs the modeler to always keep a copy of the original tool parts (the second and subsequent objects selected) that intersect the blank part (the first object selected). The blank part is not preserved. To preserve the blank part, do as instructed in the preceding bullet.

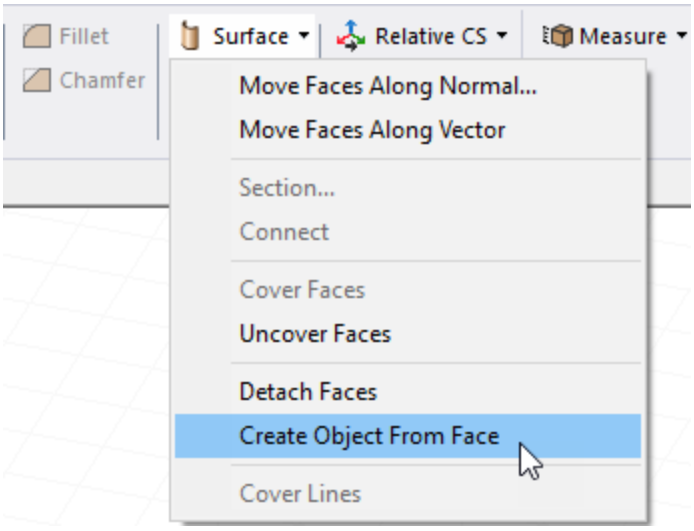
Creating an Object from a Face

The **Modeler** > **Surface** > **Create Object from Face** command copies a selected face, resulting in a new 2D sheet object.

To create a new object from a face:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Right-click in the Modeler window and choose **Select Faces** from the shortcut menu.
2. Select the object face you want to copy. If you select multiple faces, each becomes a new object.
3. From the menu bar, click **Modeler** > **Surface** > **Create Object From Face** or, on the **Draw** ribbon tab, click **Surface** > **Create Object From Face**:



The face is copied, resulting in a new 2D sheet object in the same location.

> **Tip:**
>
> This command is useful for assigning a boundary to the partial intersection of two faces. To do this, first select the faces and create objects from them using the procedure shown above. Next, make sure the **Clone tool objects before intersecting** option is cleared in the *3D Modeler> Operation* section of the *General Options*. Then, select the two sheet objects and use the **Modeler** > **Boolean** > **Intersect** command to create a sheet object that includes only the intersection of the two faces. Finally, assign the boundary to the new sheet object.

Creating an Object from an Edge

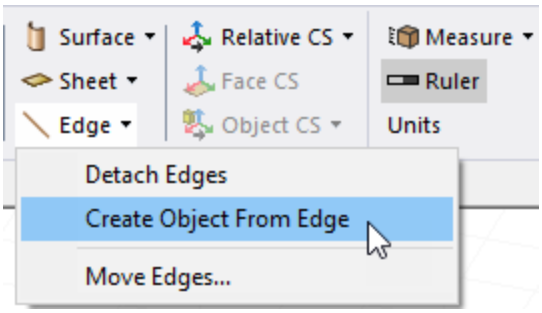
The **Modeler** > **Edge** > **Create Object From Edge** command copies a selected edge, resulting in a new line object.

To create a new object from an edge:

1. Right-click in the modeler window and choose **Select Edges** from the shortcut menu.
2. Select the object edge you want to copy. If you select multiple edges, each edge becomes a new line object.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. From the menu bar, click **Modeler** > **Edge** > **Create Object From Edge** or, on the **Draw** ribbon tab, click **Edge** > **Create Object From Face**:

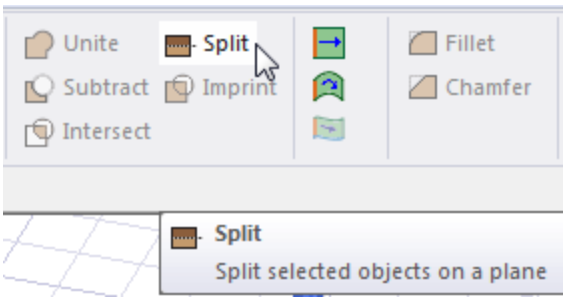


The edge is copied. The resulting object appears in the History Tree as a line object.

Splitting Objects

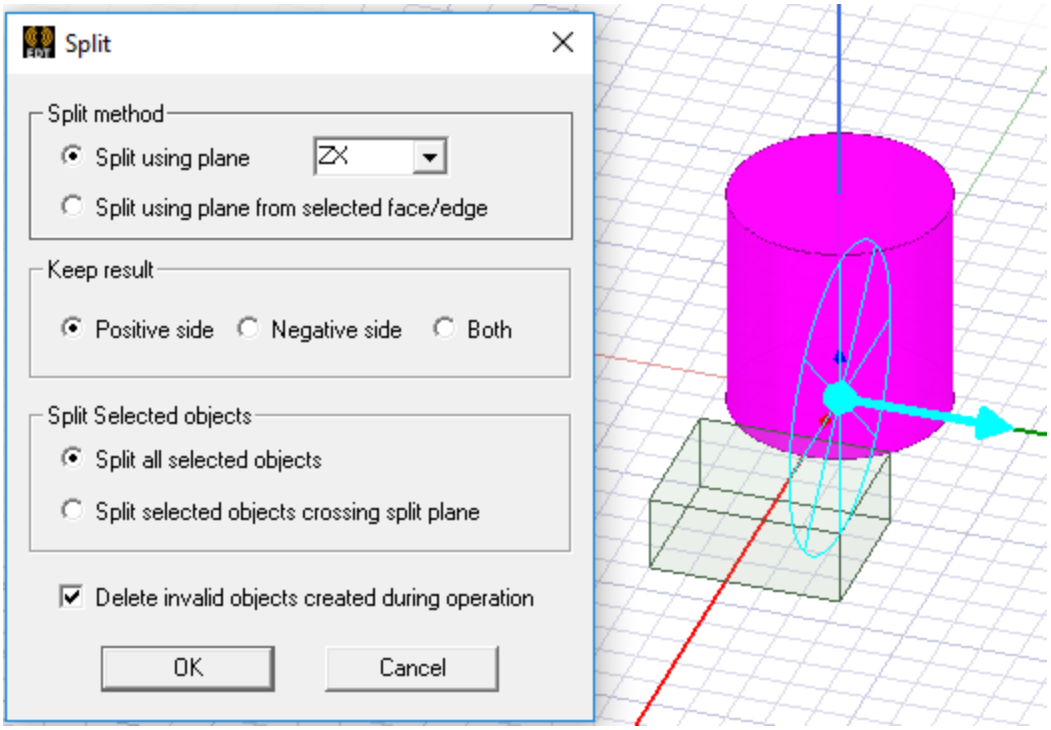
To split an object or several objects using one of the global planes (XY, YZ, or XZ) or a plane based on a selected face or arc, use the **Modeler** > **Boolean** > **Split** command.

1. Select the object you want to split. You can select more than one object.
2. From the menu bar, click **Modeler** > **Boolean** > **Split** or, on the **Draw** ribbon tab, click the **Split** icon:



The *Split* dialog box appears. Also, a cyan colored wheel shows the current split plane and points toward the positive side of the plane.

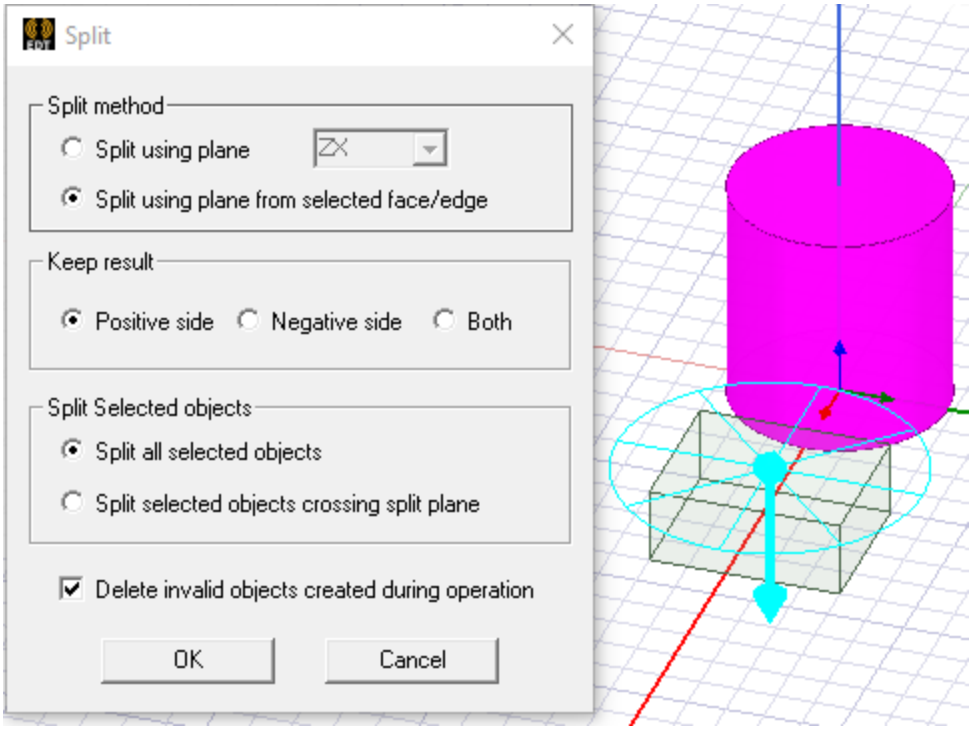
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Choose the desired *Split method* and specify the splitting plane:
   - **Split using plane:** Select the desired splitting plane (**XY, YZ,** or **XZ**) from the drop-down menu, or
   - **Split using plane from selected face/edge:** Select a single face or edge (specifically, a planar face or arc) to define the splitting plane.

     The following example shows the top face of the box being used to define the splitting plane for the cylinder:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

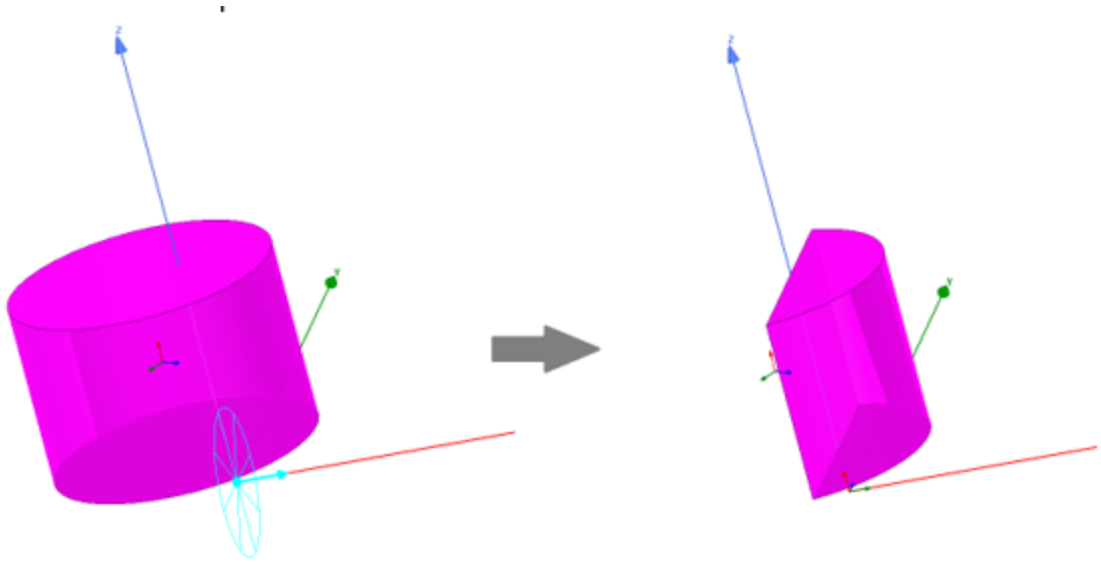


4. Select one of the following *Keep result* options to specify which object fragments you want to keep:
   - **Positive side:** Keep objects on the positive side of the splitting plane.
   - **Negative side:** Keep objects on the negative side of the splitting plane.
   - **Both:** Keep objects on both sides of the splitting plane.

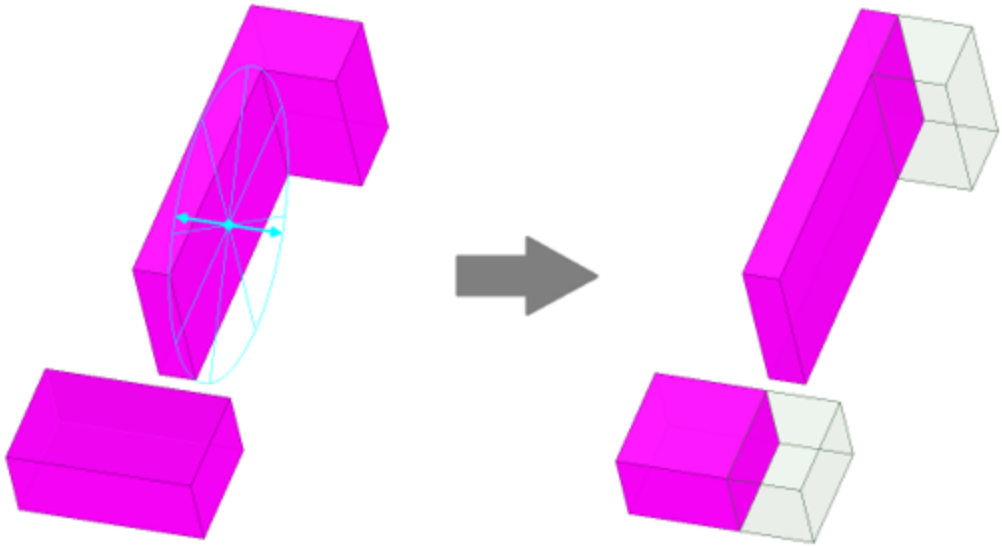
   The split plane is shown using a cyan colored wheel with spokes, as seen in the images above and below left. A cyan arrow indicates which side of the split plane is the positive side.

   In the following example, the *YZ* plane (green and blue axes) is the splitting plane, and the result on the *positive side* was kept. The picture on the left is before the split. The picture on the right is the result of the completed *Split* command:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The next example shows the second *Split method* (*Split using plane from selected face/edge*). You can only select one entity, a planar face or an arc (selected as an edge). The face or edge used to define the splitting plane need not belong to one of objects selected to be split. You can select any face or arc edge in the model.. You can pan, zoom, or rotate the model viewpoint as needed to make the selection easier. Also, the *Keep result* for *Both* sides option was selected. (Notice the arrow is in both directions for the split plane graphics.) The picture on the left is before the split. The picture on the right is the result of the completed *Split* command:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Select one of the following *Split Selected objects* options:
   - **Split all selected objects:** Select this option if you do not want to preserve objects that are not crossing the split plane but are part of the selection.
   - **Split all selected objects crossing split plane:** This option allows you to identify selected objects that do not cross the split plane and ignore them for the operation. For a multiple selection, only those objects that cross the split plane are split; others are kept intact. By design, splits in existing designs from previous software versions are not changed.
6. Optionally, select the **Delete invalid objects created during operation**. If you choose to *Split all selected objects*, the operation could create invalid objects (if split plane does not cut any of selected objects and any are on the side not being kept). This option deletes any such invalid objects that might be created.
7. Once the proper selections have been made, click **OK**.

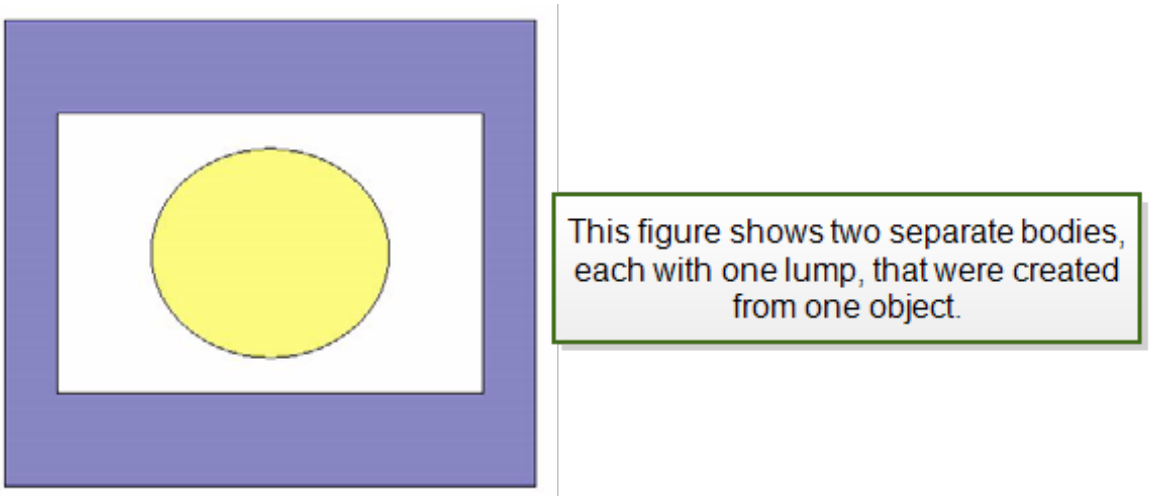
   The objects are divided and retained as specified.

Separating Bodies

To separate an object with multiple lumps (geometric bodies) into individual single-body objects:

1. Select the object you want to separate.
2. From the menu bar, click **Modeler** > **Boolean** > **Separate Bodies**.

   The bodies are separated into multiple objects.



Converting Polyline Segments

A polyline is a single object that includes any combination of straight line, arc, or spline segments. You can convert a polyline segment from one type to another. The following

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

conversions are supported:

- Straight line segments to arc or spline segments.
- Arc segments to straight line or spline segments.
- Spline segments to straight line segments.

To convert polyline segments:

1. In the History Tree, locate the polyline that contains the segment you want to convert. Expand this part of the tree.
2. Click the polyline segment operation you want to change to select it.

   The segment properties appear in the *Segment* tab of the docked *Properties* window.
3. Click in the **Value** text box in the **Segment Type** row and select the desired polyline segment type from the drop-down menu.

   The polyline segment you selected is changed to the new type.

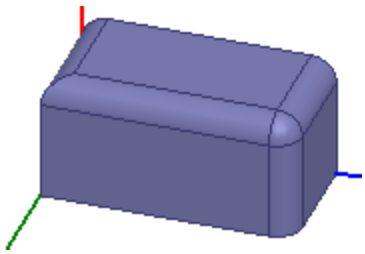
   > **Note:**
   >
   > Converting an arc or spline segment to a straight line segment results in two straight line segments; one segment is created between the start point and midpoint and one segment is created between the midpoint and endpoint.
4. By default, curved segments are generated as smooth (true) curves. If you want to approximate an arc or spline using multiple discrete straight segments, enter an integer of 2 or greater in the **Number of Segments** value. A value of zero produces a smooth curve.

   Segmented curves can be useful when sweeping polylines into surfaces. Segmented surfaces provide some regularity to the mesh and can enable you to visualize surface twisting when you sweep along a path with a twist angle specified.
5. If the changes are not what you expected, undo the change using the **Edit** > **Undo** command or by pressing **Ctrl+Z**.

Rounding the Edge of Objects (Fillet Command)

The fillet command rounds the object at the original edges and vertices. The edges and vertices are replaced by new rounded surfaces that are tangential to the original adjacent surfaces, providing a smooth transition.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Vertices are only going to be replaced by new rounded surfaces if all the edges connecting to the original vertex are selected; otherwise, the vertex is preserved but moved (if necessary). The edges are replaced by quarter-cylindrical surfaces, of which the radius can be customized (see the **Fillet Radius** property). Vertices are replaced by more complicated new faces. You can control the setback distance.

The fillet command is disabled if an edge is not selected.

To switch to edge selection mode:

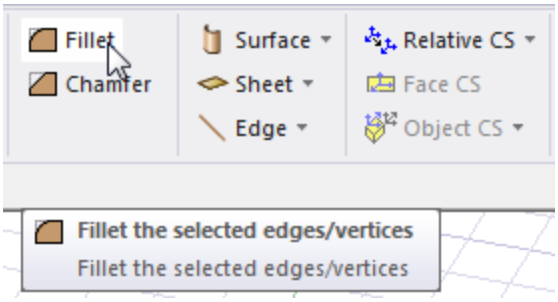
- Right-click the desktop, and choose **Select Edges** from the shortcut menu.

To round the edge of an object:

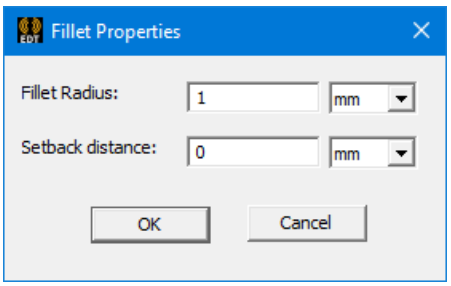
1. Select the edge you want to change.

   The edge is highlighted and the *Fillet* command is enabled.

2. From the menu bar, click **Modeler** > **Fillet**, or on the **Draw** ribbon tab, click the **Fillet** icon:



   The *Fillet Properties* dialog box is displayed:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Enter a value for the **Fillet Radius** and select units from the drop-down menu.
4. Enter a value for the **Setback distance**.

   The setback distance controls the shape of the vertex. It is the distance of the cross curve from the vertex at the end of the edge. If it is less than the fillet radius the *Setback distance* has no effect. Also, you will get an error if it is greater then the length of the edge. Note that the setback feature only works on corners where three or more edges meet and only if all edges meeting at the vertex are selected.

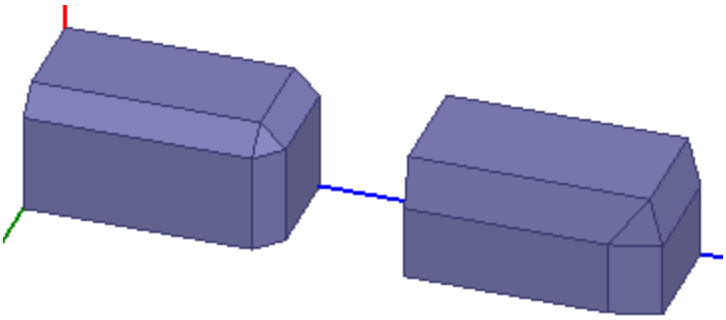
5. Click **OK** to apply the change to the edge.

   The dialog box closes and the object edges and vertices are rounded as specified.

Flattening the Edge of Objects (Chamfer Command)

The chamfer command flattens the selected edges and vertices of the object. Selected edges and vertices are replaced by new flat surfaces known as chamfers.

Chamfers can be symmetric, or asymmetric:



The angle between the original faces and the chamfer's face varies depending on the angle between the original faces, and it also depends on the left and right chamfer distances. For example, for two faces at 90° with an equal distance (symmetrical) chamfer, the edge chamfer's face will be 45° from both of the original faces.

Vertices are only going to be replaced by new flat faces if all the edges connecting to the original vertex are selected; otherwise, the vertex is preserved but moved (if necessary).

The chamfer command is disabled if an edge is not selected.

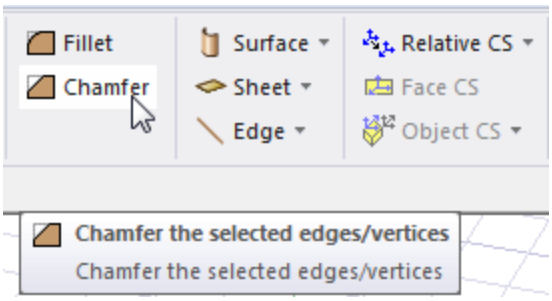
To flatten an object's edge for a symmetric chamfer:

1. Right-click the desktop, and select **Select Edges** from the shortcut menu.
2. Select the edge (or edges) you want to change.

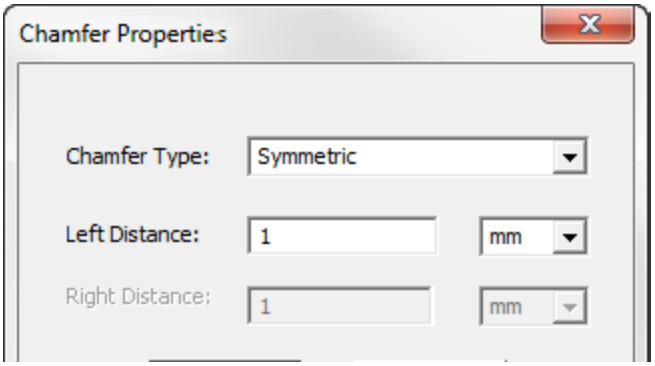
   The edge(s) is highlighted, and the **Chamfer** command is enabled.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. From the menu bar, click **Modeler** > **Chamfer** or, on the **Draw** ribbon tab, click the **Chamfer** icon:



The *Chamfer Properties* dialog box appears.



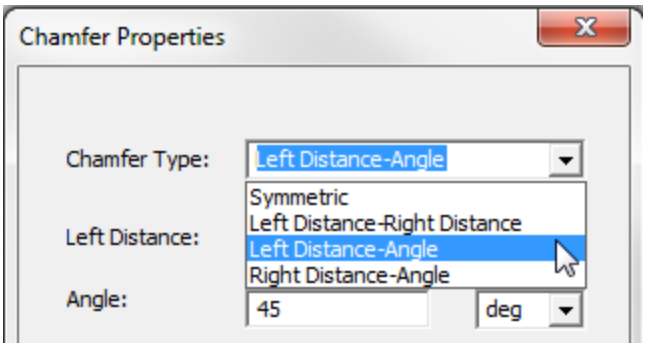
4. Type a **Left Distance** value and select the units from the drop-down menu.
5. Click **OK** to apply the change to the selected edges.

To flatten object's edge for an asymmetric chamfer:

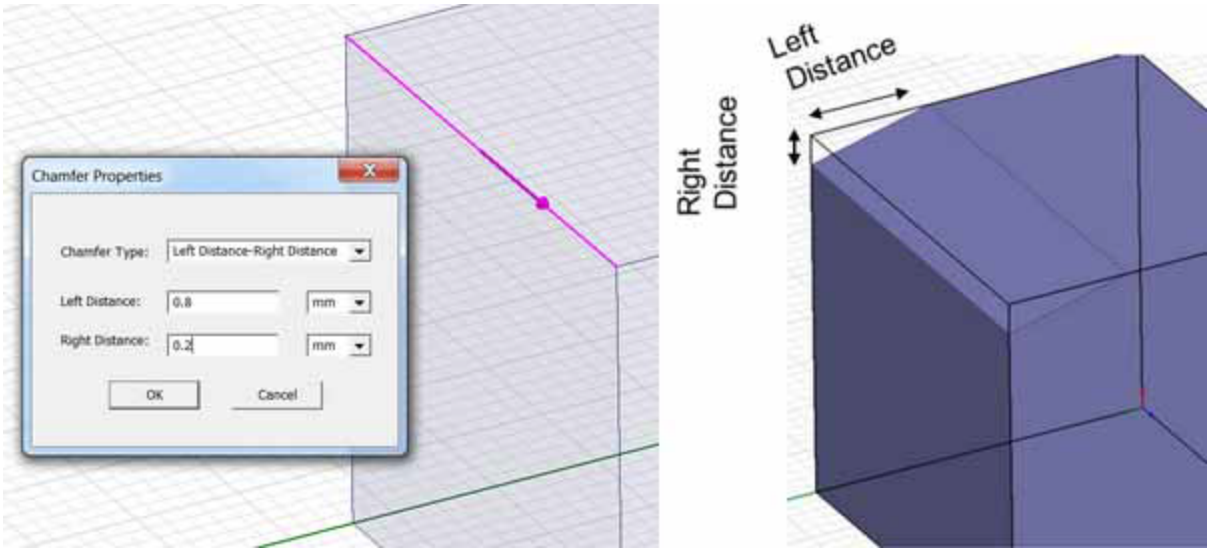
1. Select one or more edges and click **Modeler** > **Chamfer** to open the *Chamfer Properties* dialog box.
2. Use the **Chamfer type** drop-down menu to select the type:

   You control an asymmetric chamfer by selecting a type that defines the chamfer asymmetry as **Left Distance-Right Distance**, as **Left Distance-Angle**, or as **Right Distance-Angle**. Notice that red directional arrows on the selected edges provide the direction against which left distance and right distance are determined.

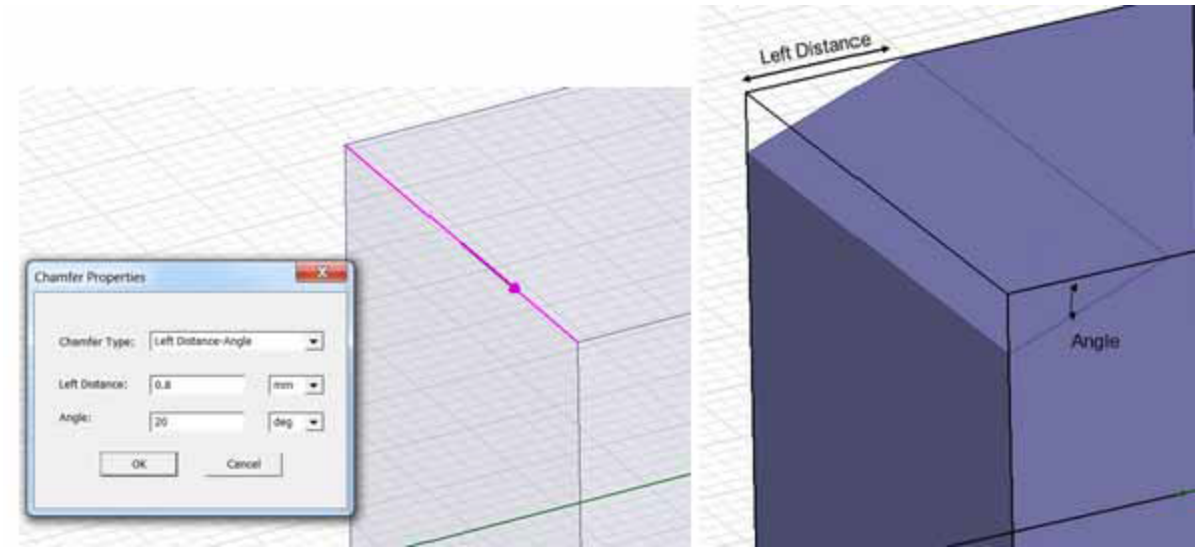
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Example 1: Left Distance-Right Distance

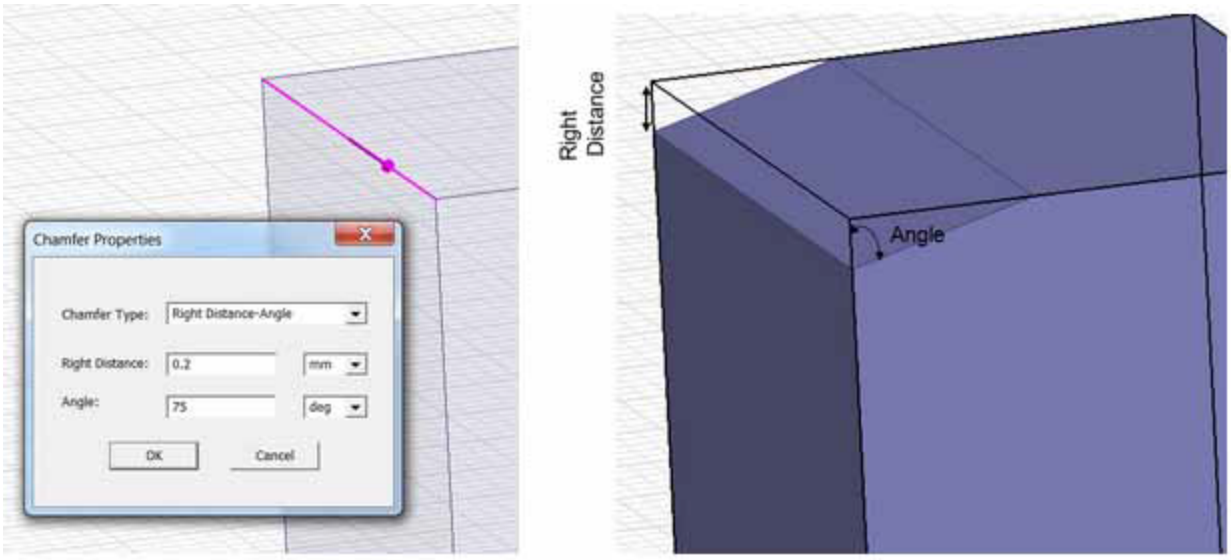


Example 2: Left Distance-Angle



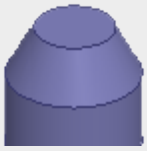
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example 3: Right Distance-Angle



Note:

Only symmetric and left distance-right distance chamfers are supported for edges formed by a curved surface.



Angle-distance chamfers are not supported in such cases and do not appear on the Chamfer Type drop-down menu.

3. Click **OK** to apply the chamfer to the selected edges.

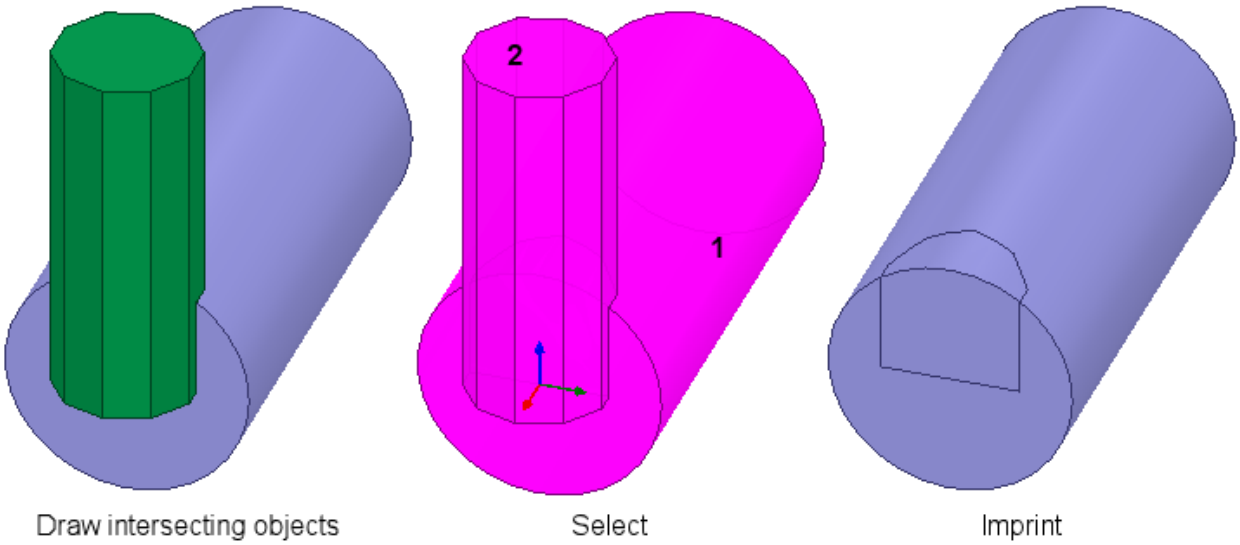
Imprinting an Object

The **Boolean** > **Imprint** command lets you imprint the geometry of one object upon another. For example, you could draw a polyhedron intersecting a cylinder, and then imprint the intersecting lines on the cylinder. This process splits the larger face into two or more faces, with the intersecting lines representing the edges of the imprinted face. The original face is cut so that it no longer extends into the imprinted area.

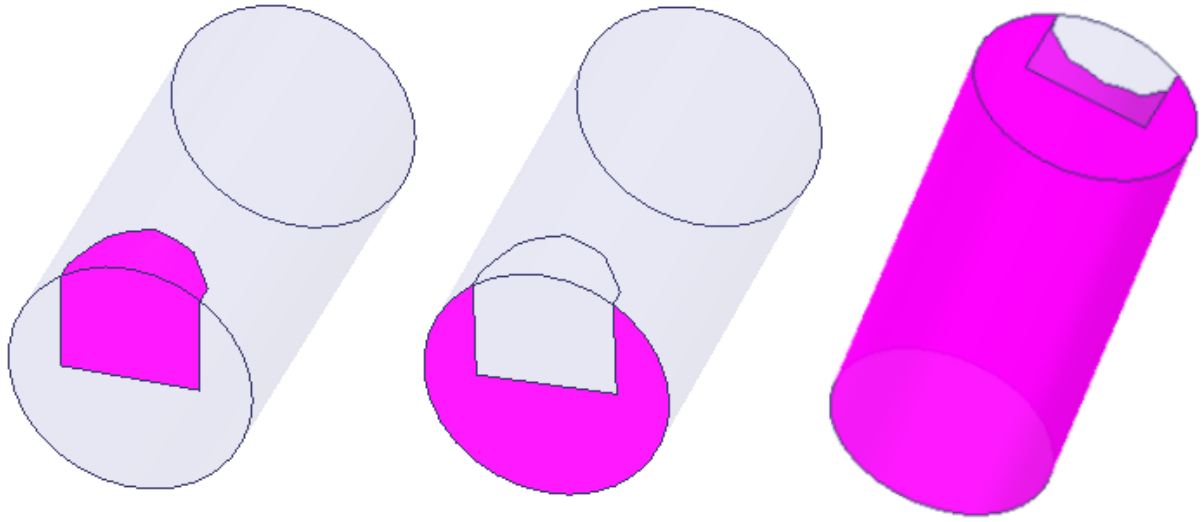
Imprinting is often an essential capability for finite element analysis. It enables a boundary or excitation to be applied to the exposed area of a model face while excluding it from areas of intersection with other parts. One example is a thermal analysis where convection is applied to

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the exposed surfaces of a heat sink. The area in contact with a semiconductor (the source or heat) is not exposed to ambient air and should not receive a convection boundary. In this case, you would imprint the semiconductor on the heat sink, producing a separate heat sink face in the contact area. This face is excluded from the selection set when applying the convection boundary.



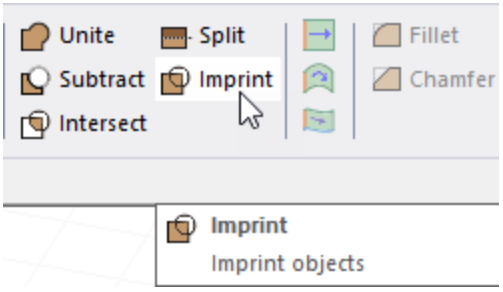
You can select the faces of the imprinted surface separately and assign properties as needed.



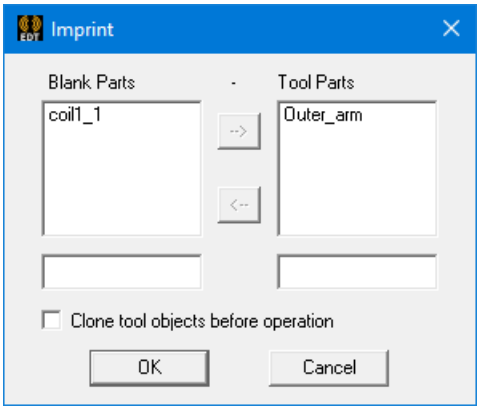
To imprint one object with another:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the intersecting objects.
2. From the menu bar, click **Modeler** > **Boolean** > **Imprint...** or, on the **Draw** ribbon tab, click the **Imprint** icon:

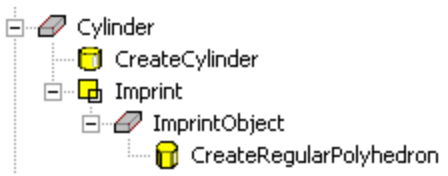


The **Imprint** dialog box displays in which you designate which objects are the *Blank Parts*, and which the *Tool Parts*. If necessary, you can select the objects in lists, and use the arrow keys to move them. If desired, you can clone the tool objects before the imprint operation.



3. Optionally, select **Clone the tool objects before operation** to retain the tool part. Otherwise, only its imprint on the blank part will remain.
4. Click **OK** to close the dialog box and perform the Boolean imprinting.

   The History Tree retains the ImprintObject command and the Create<object> command for the imprinted tool object:



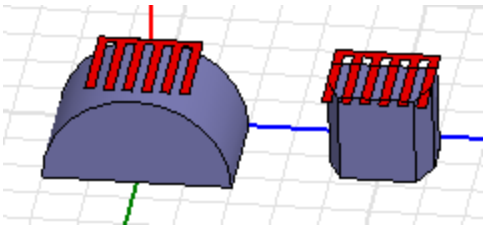
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you select the Imprint command in the History tree, you can suppress the command via the docked *Properties* window. If you select the *Create<object>* icon, you can edit the properties of that object. The changes you apply carry over to the imprinting operation, modifying the imprinted face accordingly.

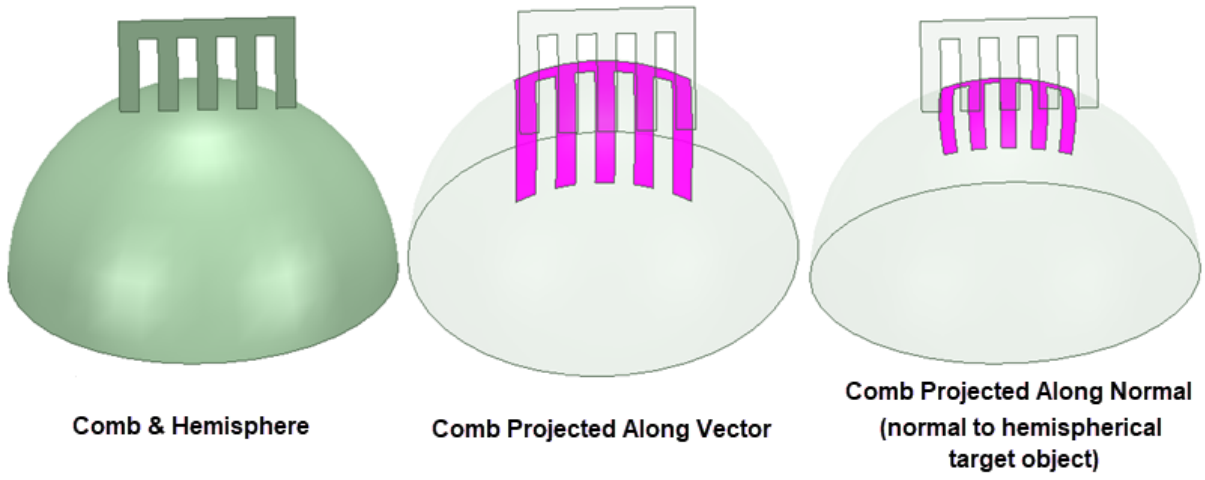
In the *3D Modeler* > *Operation* section of the *General Options*, you can also enable the **Automatically imprint wrapped sheets** option to automatically perform an *Imprint* operation command after executing the **Modeler** > **Surface** > **Wrap** command.

Imprint Projection Commands

The **Boolean** > **Imprint Projection** commands lets you project the form of a sheet object onto the face or faces of another object (either solid or sheet). The receiving object faces can be flat, curved, or faceted.

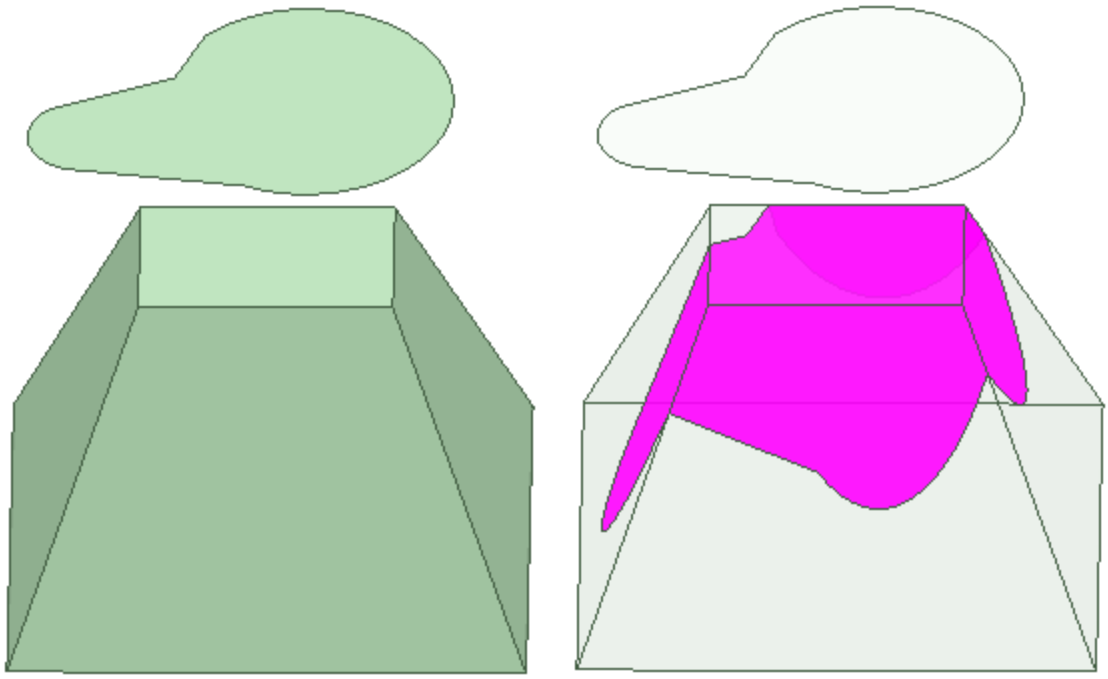


If the receiving surface is curved or inclined, and you choose to project *Along Normal* direction, the dimensions of the projection are affected. The reason is that the projection direction is normal to the curved or inclined face of the receiving object and not normal to the object being projected.

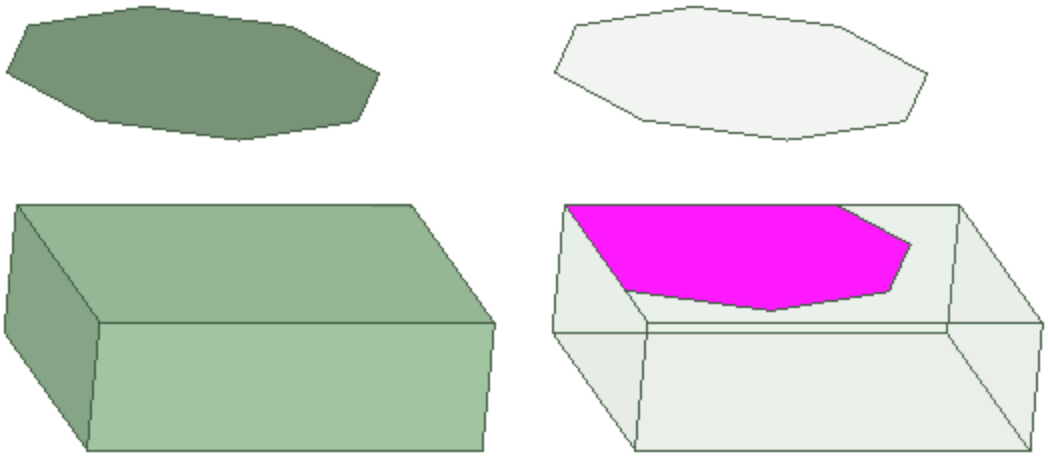


The projected shape will wrap around the target object, from the face nearest to the center of the imprinting area and extending to the adjacent target object faces:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If the projection of the imprinting object extends beyond the extents of the receiving object, the imprinted shape is truncated:

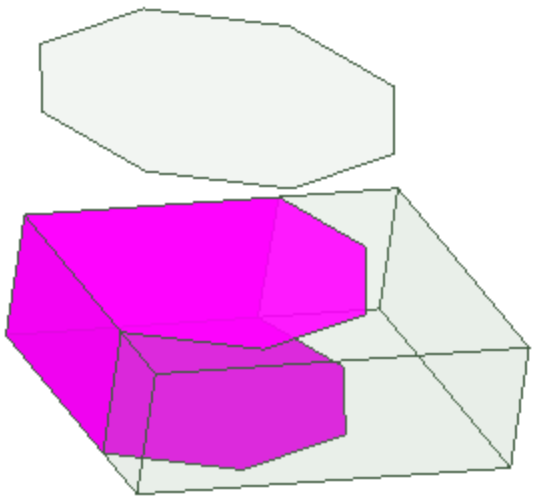


You can select the imprinted faces on the target object separately, and perform subsequent operations (such as creating sheet objects from the faces, assigning boundaries or excitations, or selecting for field overlays).

To perform an Imprint Projection:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

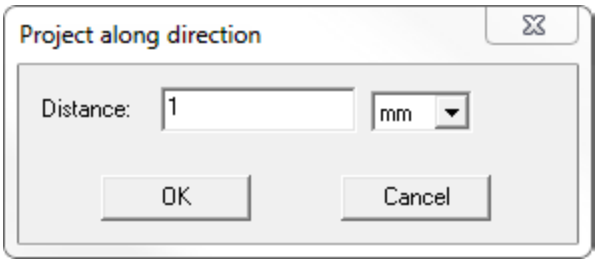
1. If you want to preserve the tool object that will be imprinted on the target object, use one of the following methods:
   - Copy the tool object to the clipboard so that you can paste it back into the model after the imprinting operation.
   - In the *3D Modeler > Operation* section of the *General Options*, select the **Clone tool objects before projecting** option.
2. Select the projecting and target *objects*.
3. From the menu bar, click **Modeler** > **Boolean** > **Imprint Projection** > **Along Normal** or **Modeler** > **Boolean** > **Imprint Projection** > **Along Direction...**
   - If you select **Along Normal**, the projection occurs along the direction normal to the target faces and the operation is completed immediately. Imprinting occurs only on target faces visible from the imprinting object's point of view.
   - If you select **Along Direction**, you must specify two points that define the imprinting vector direction (step 3 below). Once you have done this, you see a dialog box for specifying the projection **Distance**. This option allows you to limit the projection distance so that only near faces are affected. Unlike the *Along Normal* option, the *Along Direction* option can imprint onto the target object faces that are not visible from the imprinting object's point of view (that is, those on the opposite side of the target object):



4. If performing an imprint projection operation using the **Along Direction** option:
   a. Specify the imprinting vector direction. You can click in the Modeler window or use the coordinate entry text boxes in the Status Bar.

      The *Project along direction* dialog box appears:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



b. Specify the maximum projection **Distance** and select the length unit from the drop-down menu.
c. Click **OK**.

   The dialog box closes and the Boolean imprinting operation is performed.

After you perform the imprinting, the History Tree shows the ImprintProjection command and the create command for the imprinted object.

If you select the *Imprint Projection* command in the History Tree, you can suppress the command via the docked *Properties* window. If you select the *Create<object>* icon for the object, you can edit the properties of that object. The changes applied to the object carry over to the imprinting operation.

Purging Object History

Each object is a sequence of modeler-based operations. The history for each object is shown under its name in the History Tree. You can use the **Purge History** command to remove the history of operations while not affecting the geometry itself. This is useful when you wish to perform healing operations on the object. If there is an object for which you want to keep the history, you should make a copy of the object for that purpose before purging.

To purge the history:

1. Select the object.
2. From the menu bar, select **Modeler** > **Purge History**. The history for the model is purged, and the context for the **Undo** and **Redo** commands is updated.

Related Topics

Working with the History Tree

Generate History

Generate History to Reproduce Portions of Model

If a primitive object (cylinder, cone, box, circle, 3D polygon, etc.), polyline object (line, spline, or arc), circle, or ellipse is imported, or if the history was previously purged, you can select the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

object and use the **Generate History** command to reproduce, in the History Tree, the primitive object definition or the individual line segments used to create a polyline. *Generated History* uses the original coordinate system in which the objects were created. *Generate History* also retains boundary conditions.

To reproduce the line segments in the model history tree:

1. Select the primitive or polyline object.
2. From the menu bar, click **Modeler** > **Generate History**.

Selecting Items in the 3D Modeler Window

To modify or learn more about an item's properties, you must first select it. All commands you choose while an item is selected are performed on or in reference to the selected item.

What selection mode do you want to use?

- Select Submodels
- Select Objects
- Select Faces
- Select Edges
- Select Vertices
- Select Multi
- Coordinates in the drawing space
- Select By Area including optional filters
- Select by History Tree Group
- Select by Submodel
- Select by intersection error message
- Selecting the Face or Object Behind

Selection menu for:

The **Edit** > **Select Objects** menu and **Modeler** window shortcut menu includes selection for:

- By Name
- By Material
- By Variable
- All Model Objects
- All Non-model Objects
- All Primitive Shapes
- All Non Primitive Shapes
- All Solids

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- All Sheets
- All Lines

Selections enabled for features of selected Object:

Some **Edit** > **Extend Selection** menu commands are enabled when you select an appropriate object:

- All Object Faces
- All Object Edges
- All Face Edges
- Faces on Plane
- Select Connected Vertices
- Select Connected Edges
- Select Connected Faces
- Select Edge Chain
- Select Face Chain
- Select Sheet Edges

History Tree Icon Menu Selection

You can also right-click **History** tree icons to **Select All** for ungrouped:

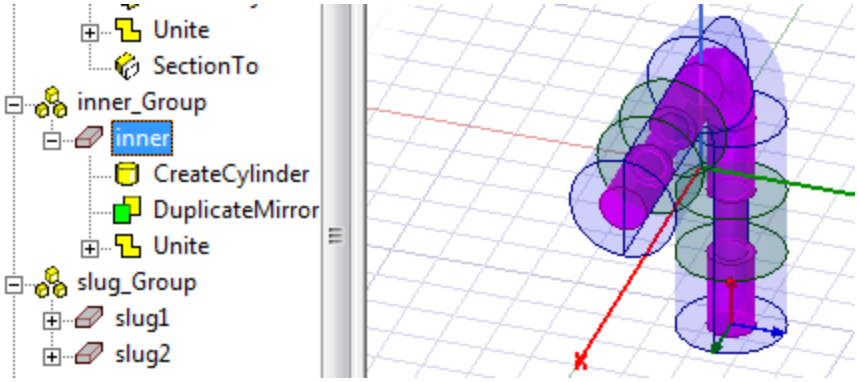
- Model
- Group
- Solids
- Selected Material, for example copper or air.
- Sheets
- Lines

> **Note:**
>
> If selected objects do not display correctly, for some graphics card, you can improve performance by changing settings in the *NVIDIA Control Panel* application. In the *Select a Task* tree, navigate to **3D Settings** > **Manage 3D Settings**. Then, under the **Global Settings** tab, choose **Workstation App - Dynamic Streaming** from the **Global presets** drop-down menu.
>
> For more information about graphics cards, navigate to the appropriate Windows or Linux Ansys Installation Guide and search for **OpenGL**.

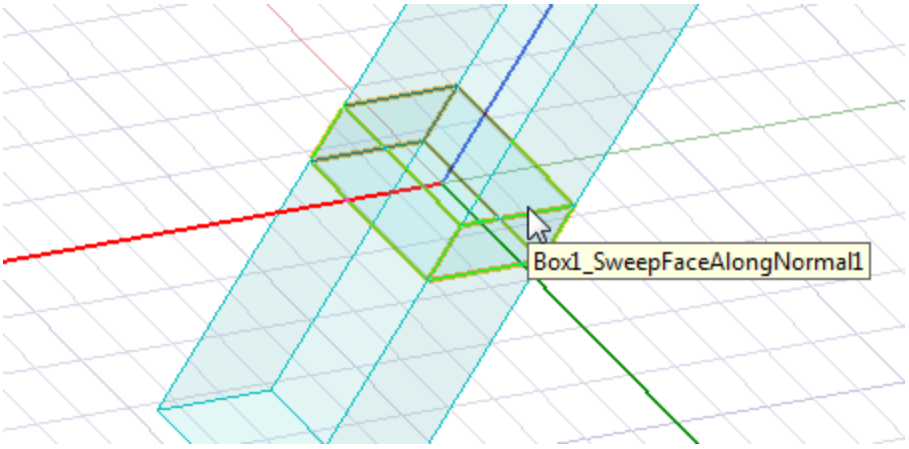
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting Objects

By default, the modeler is in *object selection mode*. Simply click an object in the view window or an object name in the history tree and it will be selected. All other objects become relatively transparent. Selected objects become the color specified under the **Display** tab of the **Modeler Options** dialog box.



When the mouse hovers over an object in the view window, that object is highlighted, which indicates that it will be selected when you click. As you hover the cursor over an entity, tooltips indicate its type/ID (object name in the case of objects, Face_id in the case of faces, and so on). This feature helps you distinguish between face-of-sheet-object pick versus sheet-object pick.

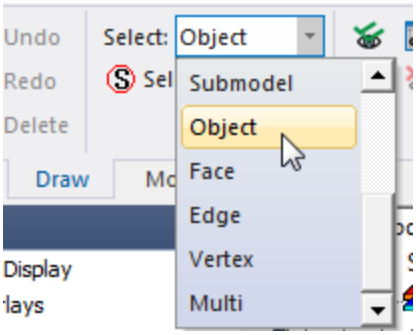


If the modeler is not currently in object selection mode, you can switch to it using one of the following methods:

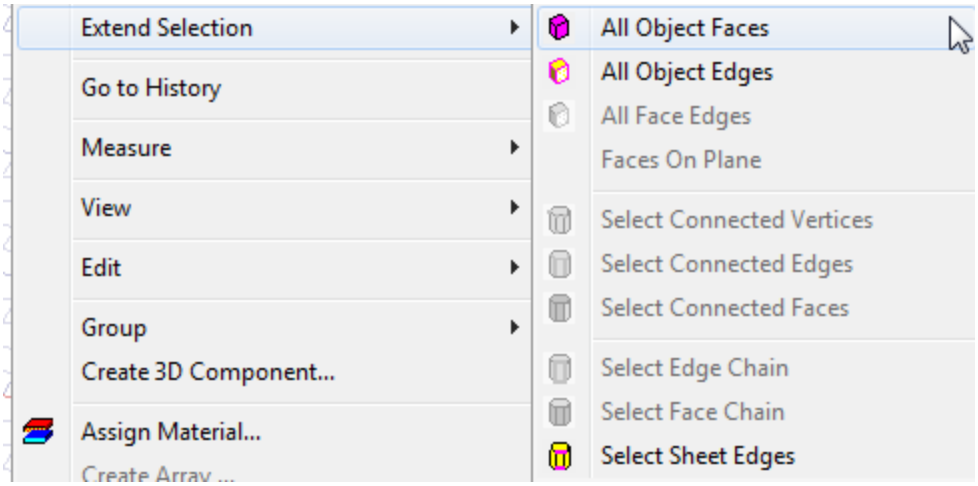
- Press the **O** shortcut key while the Modeler window is active.
- Right-click in the Modeler window and click **Selection Mode** > **Select Objects**.
- From the menu bar, click **Edit** > **Selection Mode** > **Objects**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- On the **Draw** or **Model** ribbon tabs, choose **Object** from the **Select** drop-down menu:



When you've selected an object, the menu bar's **Edit** > **Extend Selection** submenu and the **Extend Selection** submenu in the Modeler window's shortcut menu show the following options:

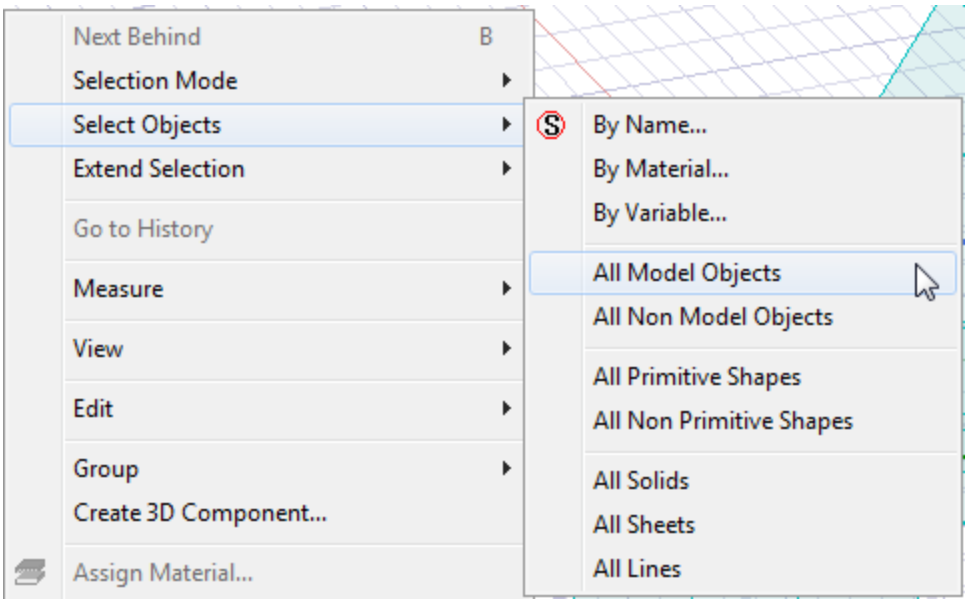


Selecting Several Objects

1. If you are selecting objects in the Modeler window make sure that the modeler is in object selection mode by pressing the **O** shortcut key. You can always select objects in the History Tree.
2. Select several objects in one of the following ways:
   - Hold down **Ctrl** and click the objects in the Modeler window that you want to select.
   - Hold down **Ctrl** and click the object names in the History Tree that you want to select.
   - In the History Tree, select a range of objects by first clicking one object to select it, and then Shift+click to extend the selection of visible items.
   - In the History Tree, under *Lists*, select **AllObjects**, which is an automatically created list of all model objects.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- From the menu bar, click **Edit** > **Select All** to select all objects that were drawn in the active Modeler window, including objects that are not currently visible.
- Press **Ctrl+A** or, from the menu bar, click **Edit** > **Select All Visible** to select all visible objects in the active Modeler window.
- From the menu bar, click **Edit** > **Select Objects** > or right-click in the Model window and go to the **Select Objects** > submenu. Then, choose a command from the submenu suitable for the objects you want to select:



- Right-click a History Tree icon for Model, Group, object, category (such as Solid or Sheet), or material to perform context-sensitive selections. See Select Objects in the History Tree.

To deselect all objects, do one of the following:

- From the menu bar, click **Edit** > **Deselect All**.
- Press **Ctrl+Shift+A**.

Selecting Objects by Name or Material

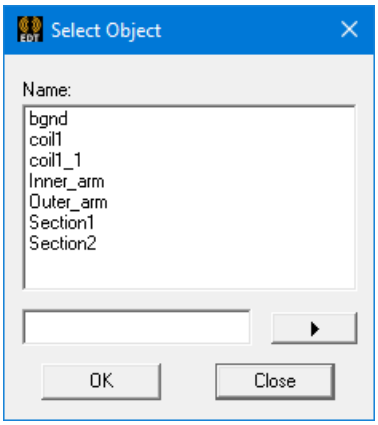
1. Make sure that the modeler is in the *Object* selection mode by pressing the **O** shortcut key or by choosing **Object** from the **Select** drop-down menu on the **Draw** or **Model** ribbon tab. Then, proceed to one of the following two procedures.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selection by Object Name

2. Access the *Select Object* dialog box using one of the following three methods:
   - From the menu bar, click **Edit** > **Select Objects** > **By Name**.
   - On the **Draw** or **Model** ribbon tab, click **Select by Name**.
   - Right-click in the *Modeler* window and choose **Select Objects** > **By Name** from the shortcut menu.

   The *Select Object* dialog box appears, populated with a list of the objects in the model, by name:



3. In the *Name* list, click the name of the object you want to select. Use the **Ctrl** key to select more than one.

   Alternatively, type the name of an object in the empty text box, click the arrow to the right of the text box, and choose **Select** or **Deselect**.

4. Click **OK**.

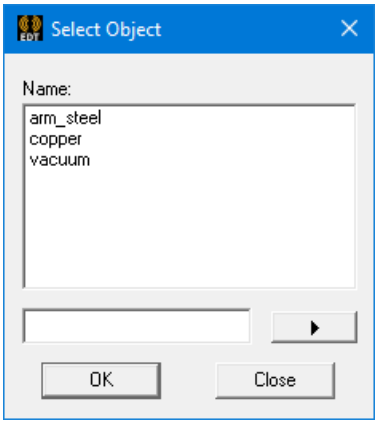
   The object or objects specified are selected.

Selection by Object Material

2. Access the *Select Object* dialog box using one of the following two methods:
   - From the menu bar, click **Edit** > **Select Objects** > **By Material**.
   - Right-click in the *Modeler* window and choose **Select Objects** > **By Material** from the shortcut menu.

   The *Select Object* dialog box appears, populated with a list of the materials comprising the model objects:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. In the *Name* list, click the material of the object or objects you want to select. Use the **Ctrl** key to select more than one material.

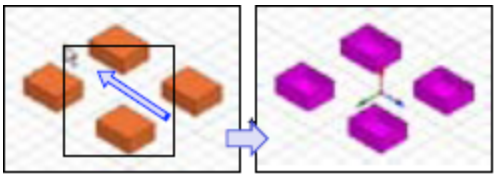
   Alternatively, type the name of a material in the empty text box, click the arrow to the right of the text box, and choose **Select** or **Deselect**.

4. Click **OK**.

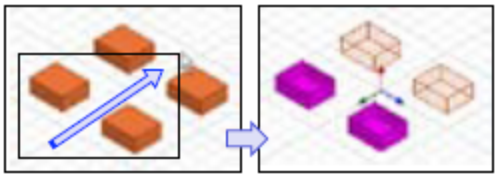
Select By Area

You can select items by area in the Modeler window by clicking and dragging a rubber band (selection box) around objects. Rubber band selection works differently depending on the direction you drag the mouse:

- **From right-to-left:** Selects all items that are wholly or partly enclosed within the rubber band.



- **From left-to-right:** Selects only the items that are wholly enclosed within the rubber band.

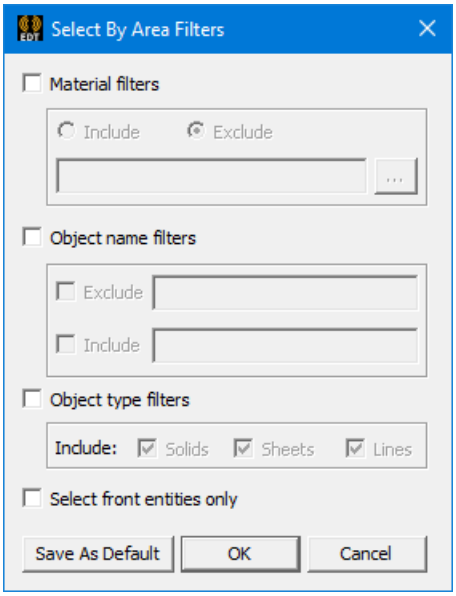


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Area selection works with Object, Face, Edge, and Vertex selection modes, but not for the Multi selection mode.

You can control which objects to include or exclude from area selections based on material, object name, or object type. To do this:

1. From the menu bar, click **Edit** > **Select By Area Filter** to display the following dialog box:



2. Check **Material filters** to enable the **Include** and **Exclude** radio buttons. Use the text box to specify filters by name, or use the ellipsis [...] button to display the *Select Definition* dialog box, from which you can browse and select materials.
3. Check **Object name filters** to enable the **Exclude** and **Include** check boxes and associated text boxes in which you can specify object names.
4. Check **Object type filters** to enable the *Include* check boxes for including **Solids, Sheets,** and/or **Lines**.
5. Check **Select front entities only** to prevent entities in the background (that is, those hidden behind other entities) from being selected.
6. If you click **Save As Default**, the settings persist for the project (until you change the settings and *Save as Default* again.
7. Click **OK** to close the dialog box.

   Now when you click and drag around an area, those objects which meet the filter criteria are highlighted in the Modeler window, and those objects are shown as selected in the History Tree.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Default Color and Transparency of Selected Objects

To set the color of objects when they are selected:

1. From the menu bar, click **Tools** > **Options** > **General Options** or, on the **Draw** ribbon tab, click **General Options**.

   The *Options* dialog box appears.

2. In the tree at the left side of the dialog box, select the **3D Modeler** > **Display** > **Rendering** group of settings.
3. In the *Rendering Defaults* section, choose **Select** from the **Default color** pull-down menu.
4. Click the color button to the right of *Select*.

   The *Color* palette appears.

5. Select a color and then click **OK**.

   Any objects you select after this point will temporarily become the default color you selected.

By default, the modeler shows selected objects as nearly opaque and shows non-selected objects as nearly transparent. This features helps you to easily distinguish between selected and non-selected objects and to see selected objects on the back side of the model.

To set the transparency of selected and non-selected objects:

1. From the menu bar, click **View** > **Options**.

   The *3D UI Options* dialog box appears. The *When there is a selection* region contains check boxes for setting the transparency for selected and non-selected objects.

2. Ensure that the check box is selected for any value you want to change.

   Each check box enables the associated value box. The default transparency for selected objects is 0.1, which makes them almost opaque. The default transparency for non-selected objects is 0.9, which makes them highly transparent.

3. Enter a new value, and click **OK**.

Setting the Default Color of Highlighted Objects

1. From the menu bar, click **Tools** > **Options** > **General Options** or, on the **Draw** ribbon tab, click **General Options**.
2. In the tree at the left side of the dialog box, select the **3D Modeler** > **Display** > **Rendering** group of settings.

   The *Options* dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In the *Rendering Defaults* section, choose **Highlight** from the **Default color** pull-down menu.
4. Click the color button to the right of *Highlight*.

   The *Color* palette appears.

5. Select a color and then click **OK**.

   Outlines of all object you hover over temporarily become the default color you specified above.

Creating an Object Selection

Create an object selection when you want to group multiple objects for convenient selection and processing. Objects in a selection can still be treated as separate objects. The same object can be included in several different selections.

To create an object selection:

1. If you are selecting in the Modeler window, make sure that the modeler is in select object mode by pressing the **O** shortcut key. This is not necessary for selecting in the History Tree.
2. Select the objects you want to include in the selection.
3. Click **Modeler** > **Named Selection** > **Create** > **Object Selection**.

   The object selection is created with the default name ObjectSelection*n*. It appears in the History Tree under **Named Selection**. Selecting an object selection displays the properties of that selection in the docked *Properties* window. One of the properties is a list of objects contained in the selection.

   To rename the object selection, edit its Name in the docked *Properties* window. Object selections are sorted in alphanumeric order.

There is an automatically created selection called *AllObjects* included in the **Named Selection** branch. Click on this entity to select all objects in the model.

The object selection is treated as one volume when you plot and perform fields calculations. It will be listed in the *Geometry* window of the Fields Calculator when you select **Volume**.

If a selection contains mixed types of geometry, for example, volume and sheet objects, the volume calculation only uses the geometry of the highest dimension in plots or integral, and so forth.

- Line objects in the selection do not affect where the fields are saved. If all objects in the selection are lines, fields will be saved on all objects.
- To postprocess on line objects when saving fields on an object/face selection, include both the line and its containing object/face in the selection.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example: To plot the E-field on a surface formed by the intersection of the XY-plane and several objects, first define a selection of these objects. Then, when plotting fields, select the object selection name from the **Geometry** window of the *Fields Calculator*. Fields will be plotted only at the intersection of the plane and the objects in the selection.

Reassigning Objects to Another Object Selection

You can assign objects after you have created an object selection. Creating an object selection is a convenient way to identify and select a group of objects for a field plot or calculation. Objects in a selection can still be treated as separate objects. The same object can be included in several different selections.

To reassign objects to an existing object selection:

1. If you are selecting in the Modeler window, make sure that the modeler is in select object mode by pressing the **O** shortcut key. This is not necessary for selecting in the History tree.
2. Select the objects you want to reassign.
3. Click **Modeler** > **Named Selection** > **Reassign**.

   A dialog box with the existing object selections is displayed. (They appear in the History Tree under **Named Selections**.)

4. Click the selection to which you want to assign the selected object(s) and click **OK**.

   The object is reassigned to the active selection, replacing previous list members. The *Objects* property of the selection shows the objects contained in the selection, with each name separated by a comma.

The object selection will be treated as one volume when you are plotting and performing fields calculations. It will be listed in the *Geometry* window of the *Fields Calculator*, when you select **Volume**.

Using or Viewing Selection Objects

To view the objects included in an object selection:

1. In the History Tree, expand the **Named Selection** branch.
2. Right-click the desired selection, and click **Select Assignment**.

   The objects that are included in that selection are highlighted in the Modeler window, and the attributes appear in the docked *Properties* window.

To view the properties of the object selection (including a list of the objects in the selection):

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the History Tree, expand the **Named Selection** branch.
2. Right-click the selection you want to view and click **Properties**.

   The *Properties* dialog box appears. The objects included in the selection appear as comma-separated names in the *Objects* row.

3. Click **OK** or **Cancel** to close the *Properties* dialog box.

To use an object from a selection in another operation:

1. In the History Tree, expand the **Named Selection** branch.
2. Right-click the desired selection and click **Select Assignment**.

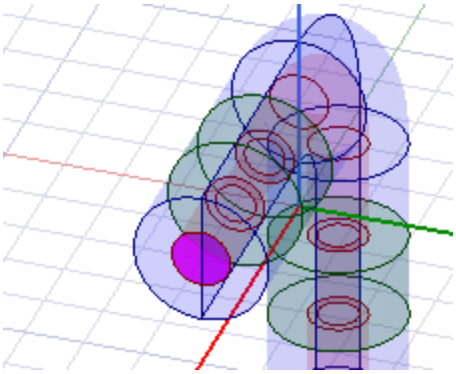
   The objects that are included in that selection are highlighted in the Modeler window, and the attributes appear in the docked *Properties* window.

3. Holding **Ctrl**, select any additional objects you might want to use in the operation.
4. Complete the operation.

   For example, you could select an object selection and an additional object, and then execute one of the Boolean commands (such as *Unite* or *Subtract*).

Selecting Faces

If the modeler is in face selection mode, click an object face in the view window to select it. The selected face is rendered in the section color (magenta by default) and is relatively opaque by default. All other objects become relatively transparent.



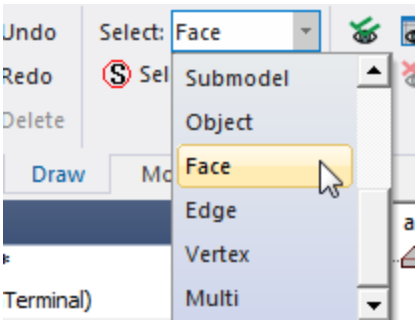
When the mouse hovers over a face in the view window, that face is highlighted, which indicates that it will be selected when you click. Selected faces become the color specified under the *3D Modeler > Display > Rendering* section the *General Options*.

To select multiple faces, hold the **Ctrl** key as you click the faces. You also have the option to create face selections, which define a list of object faces, or you can make face selections from a Face ID list in the Select Face dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

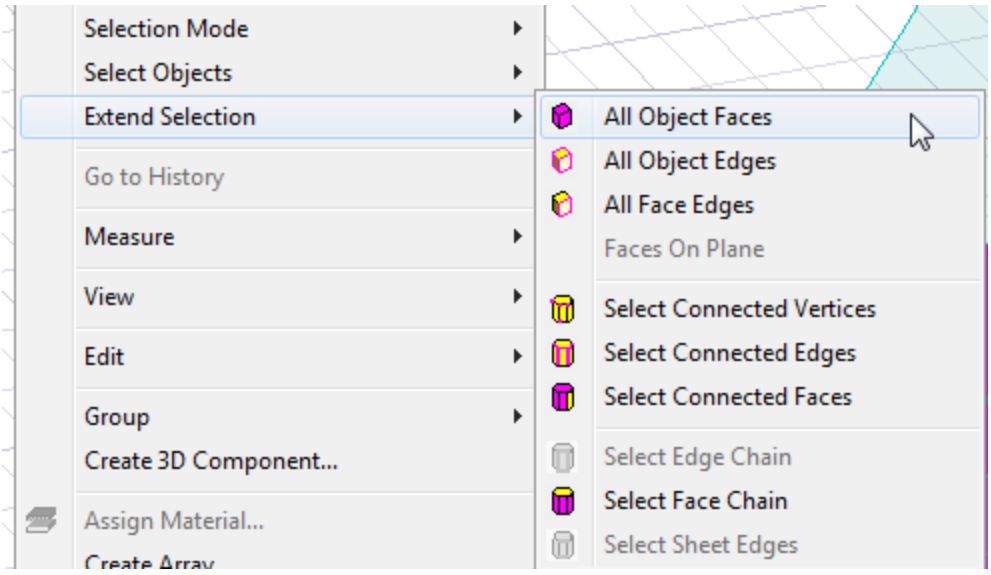
Switch to face selection mode using one of the following methods:

- From the menu bar, click **Edit** > **Selection Mode** > **Select Faces**
- Right-click in the Modeler window and choose **Selection Mode** > **Select Faces** from the shortcut menu.
- With the Modeler window active, press the **F** shortcut key.
- On the **Draw** ribbon tab, choose **Faces** from the **Select** drop-down menu.



You can also select faces in the Select Multi mode.

When you've selected a face, the menu bar's **Edit** > **Extend Selection** submenu and the **Extend Selection** submenu in the Modeler window's shortcut menu show the following options:



Various tools are available for face selection, as described in the *Related Topics* listed below.

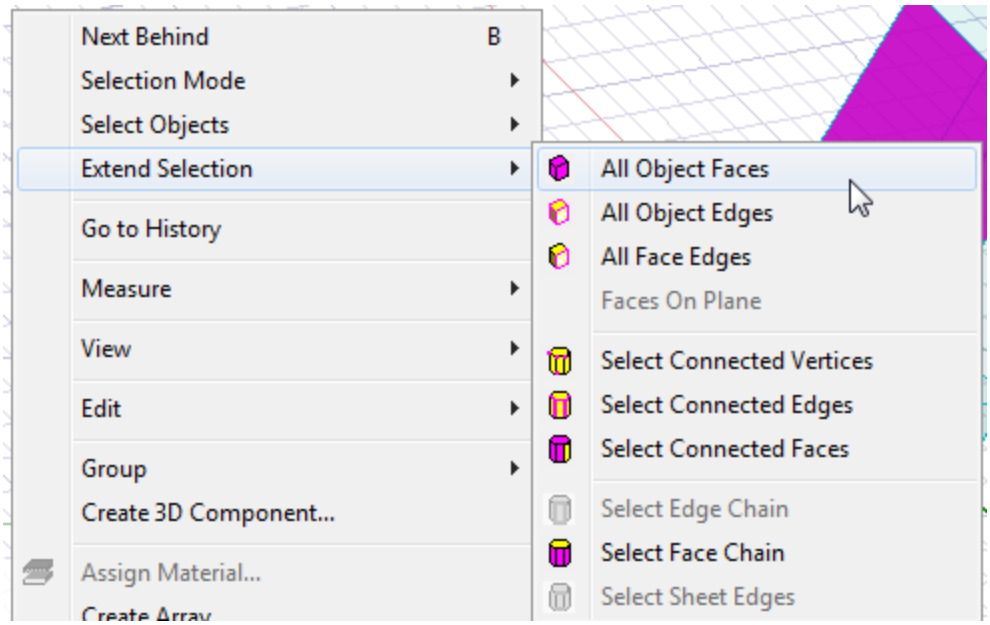
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting All Faces of an Object

1. With the Modeler window active, switch to the face selection mode by pressing the **F** shortcut key.
2. Click a face on the object of interest to select it.

   You can select a face on each of two or more objects if desired.

3. Use one of the following methods of selecting all faces of the objects of interest:
   - From the menu bar, click **Edit** > **Extend Selection** > **All Object Faces.**
   - Right-click in the Modeler window and choose **Extend Selection** > **All Object Faces** from the shortcut menu:



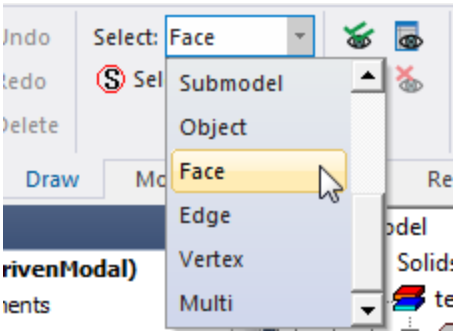
   The selection is extended to include all faces of each object that had at least one face selected originally.

Selecting Faces by Name

1. Make sure that the modeler is in face selection mode using one of the following methods:
   - With the Modeler window active, press the **F** shortcut key.
   - From the menu bar, click **Edit** > **Selection Mode** > **Faces**

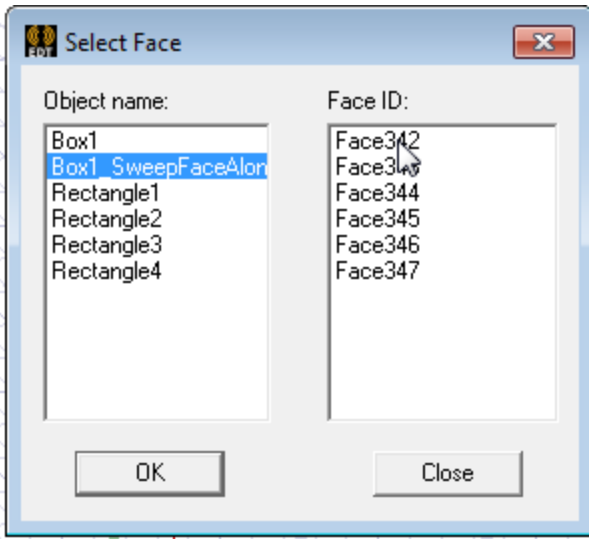
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- On the **Draw** or **Model** ribbon tabs, select **Face** from the **Select** drop-down menu.



2. From the menu bar, click **Edit** > **Select Objects** > **By Name** or, on the **Draw** or **Model** ribbon tabs, click **Select by Name**.

   The *Select Face* dialog box appears:



3. In the *Object name* box, click the name of the object with the face or faces you want to select.

   The object's faces are listed in the *Face ID* box.

4. Click the ***face*** you want to select in the *Face ID* column. Use **Ctrl+click** to select more than one face or **Shift+click** to select a range of faces.

   The selected faces are highlighted in the list and in the Modeler window.

5. Click **OK**.

Selecting Faces by Plane

To select a face that is aligned with a global plane, use one of the following two methods.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

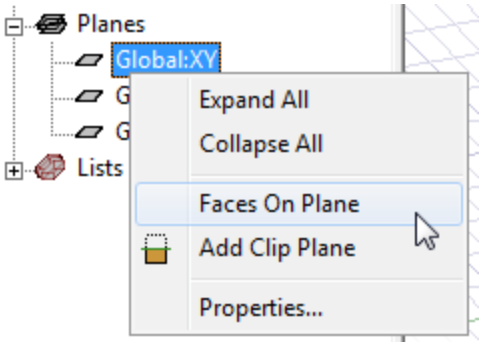
Method 1:

1. Make sure that the modeler is in face selection mode by pressing the **F** shortcut key.
2. In the History Tree, expand the **Planes** branch and select a plane to display (a global plane, a user-defined plane, or a plane associated with a relative coordinate system).
3. From the menu bar, click **Edit** > **Extend Selection** > **Faces on Plane** or right-click in the Modeler window and choose **Extend Selection** > **Faces on Plane**.

   All faces that lie on the chosen plane are selected.

Method 2:

1. In the History Tree, expand the **Planes** branch.
2. Right-click one of the listed planes (a global plane, a user-defined plane, or a plane associated with a relative coordinate system).
3. On the shortcut menu, click **Faces on Plane**.



   All faces that lie on the chosen plane are selected.

Creating a Face Selection

Create a face selection when you want to group multiple faces for convenient selection and processing. Creating a face selection is a convenient way to identify and select a specific set of surfaces for field plots, calculations, boundaries or excitations. The same face can be included in several different selections.

To create a face selection:

1. Make sure that the modeler is in select face mode by pressing the **F** shortcut key.
2. Select the faces you want to include in the face selection.
3. From the menu bar, click **Modeler** > **Named Selection** > **Create** > **Face Selection**.

   The face selection is created. It is listed in the History Tree under **Named Selection**. The default name is FaceSelection*n*. The selections appear in alphanumeric order.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Optionally, to change the name of a face selection (for example, to a name describing the listed faces as ports or boundaries), select its entry in the History Tree and change the **Name** value in the docked *Properties* window.

   If necessary, the selections in the History Tree are resorted to restore the proper alphanumeric order.

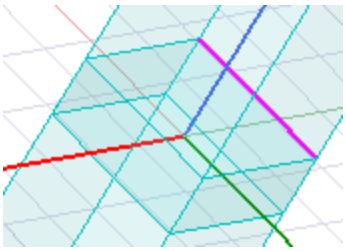
The face selection will be treated as one group of surfaces when you are plotting and performing fields calculations. The face selection will be listed in the *Geometry* window of the *Fields Calculator*, when you select the *Surface* option.

- Line objects in the selection do not affect where the fields are saved. If all objects in the selection are lines, fields will be saved on all objects.
- For postprocessing on line objects, when saving fields on a object/face selection, include both the line and its containing object/face in the selection.

If you right-click an existing face selection and choose **Select Assignment** from the shortcut menu, you can make a boundary assignment in the Modeler window to all faces in the selection, or you can execute the **Move Faces** geometry operations on the selection as a whole.

Selecting Edges

If the modeler is in edge selection mode, simply click an object's edge in the view window and it will be selected. The edge is rendered as nearly opaque using the default *Select* color. All other objects become relatively transparent.



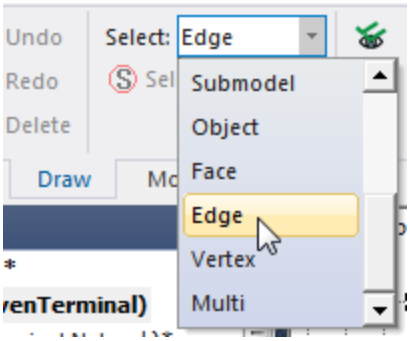
When the mouse hovers over an edge in the view window, that edge is highlighted, which indicates that it will be selected when you click.

Switch to edge selection mode using one of the following methods:

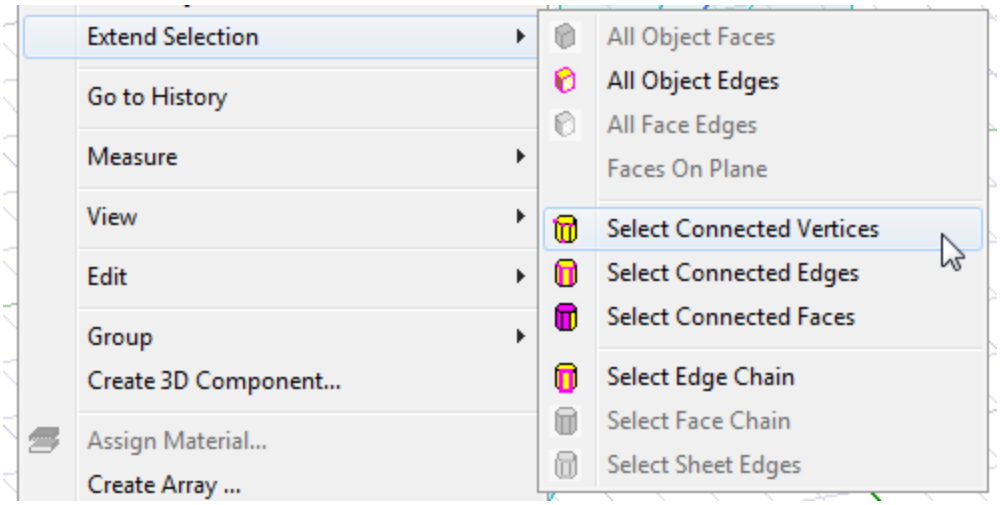
- From the menu bar, click **Edit** > **Selection Mode** > **Edges**.
- Right-click in the Modeler window and click **Selection Mode** > **Edges**.
- With the Modeler window active, press the **E** shortcut key.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- On the **Draw** or **Model** ribbon tab, choose **Edge** from the **Select** drop-down menu.



Selecting an edge enables the following commands in the menu bar's **Edit** > **Extend Selection** >**...**submenu and the **Extend Selection** >**...** submenu that appears when you right-click in the Modeler window.



You can use these commands to modify the current selection.

- All Object Edges
- Select Connected Vertices
- Select Connected Edges
- Select Connected Faces
- Select Edge Chain

You can also select multiple edges by holding the **Ctrl** key as you click additional edges.

> **Note:** If you have excluded the background object from your model, do not assign boundary conditions or sources to it.

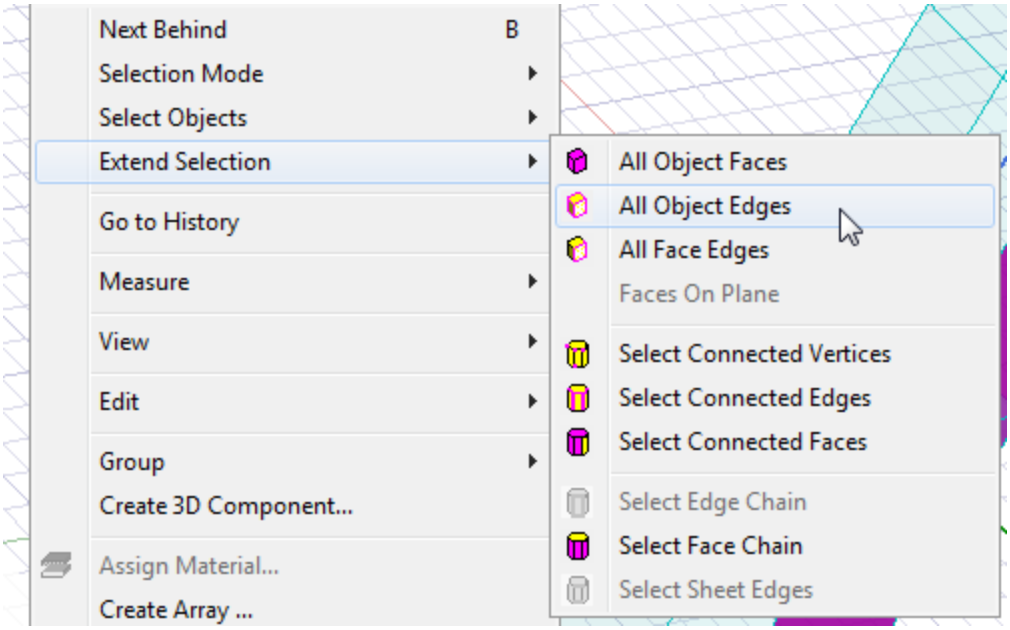
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting All Edges of an Object or Face

To select all edges of an object or face:

1. Select a single edge or face that belongs to the object for which you wish to select all edges.

   You may also select the object itself. The type of entity you initially choose affects which selection tools will be available in the next step:



2. If you selected an **object** or an **edge** in step 1, the following *All Edge* command is available:
   - From the menu bar click **Edit** > **Extend Selection** > **All Object Edges** or right-click in the Modeler window and choose **Extend Selection** > **All Object Edges** from the shortcut menu.

     All edges of the object are selected.

3. If you selected a **face** in step 1, the following two *All Edge* commands are available:
   - From the menu bar click **Edit** > **Extend Selection** > **All Face Edges** or right-click in the Modeler window and choose **Extend Selection** > **All Face Edges** from the shortcut menu.

     All edges belonging to the face are selected.
   - From the menu bar click **Edit** > **Extend Selection** > **All Object Edges** or right-click in the Modeler window and choose **Extend Selection** > **All Face Edges** from the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

shortcut menu.

All edges of the object to which the face belongs are selected.

If you selected multiple objects, all edges of those objects are selected.

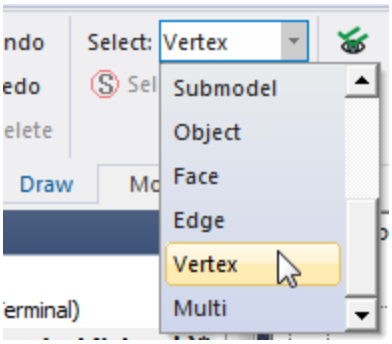
Selecting Vertices

If the modeler is in vertex selection mode, simply click an object's vertex in the view window and it will be selected. To select multiple vertices, hold the **Ctrl** key as you click the vertices.

When the mouse hovers over a vertex in the view window, that vertex is highlighted, which indicates that it will be selected when you click. Selected vertices are rendered as using the default *Select* color. All other objects become relatively transparent.

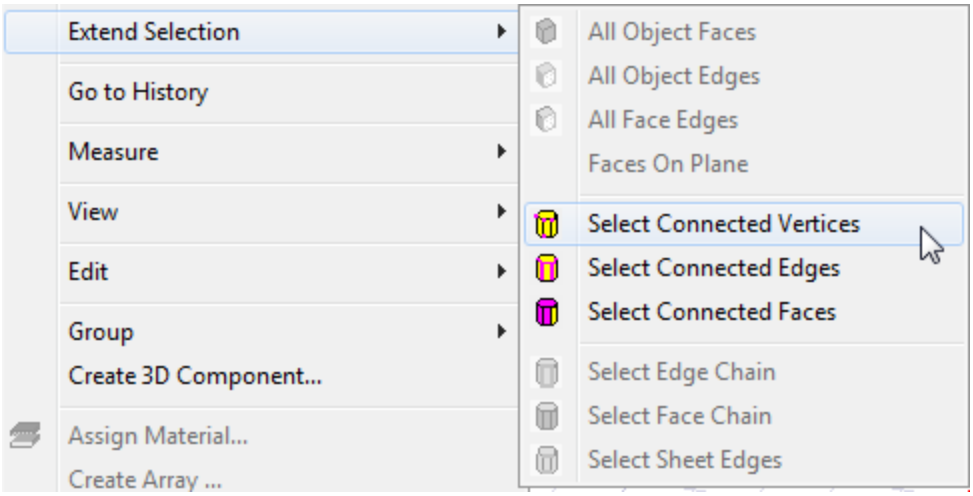
Switch to vertex selection mode using one of the following methods:

- From the menu bar, click **Edit** > **Selection Mode** > **Vertices**.
- Right-click in the Modeler window and click **Selection Mode** > **Vertices**.
- With the Modeler window active, press the **V** shortcut key to begin the vertex selection mode.
- On the **Draw** or **Model** ribbon tabs, choose **Vertex** from the **Select**drop-down menu.



Selecting a vertex enables the following commands in the menu bar's **Edit** > **Extend Selection** >**...** submenu and the **Extend Selection** >**...** submenu that appears when you right-click in the Modeler window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can use these commands to modify the current selection:

- **Select Connected Vertices:** Selects the vertices of all edges that touch the initially selected vertices.
- **Select Connected Edges:** Selects all edges that touch the initially selected vertices.
- **Select Connected Faces:** selects all faces touching the initially selected vertices.

> **Note:**
>
> Multiple vertices can exist at the same coordinates, one vertex for each object that intersects at that point. There's no visual way to discern to which object a selected vertex belongs. If you click a vertex and use any of the commands listed above, only the entities that belong to the same object will be selected. To select the associated entities for all objects, start by box-selecting the vertices at the point of interest (clicking and dragging from left-to-right). This method selects all coincident vertices at the point. Then, use one of the above three commands.
>
> You can also pick a vertex and press **B** (*Next Behind*) to select a different vertex at the point of interest. Use the trial and error method to select the related vertices, edges, or faces of the desired object. For example, you may have to press *B* two or more times to select the vertex of the desired object.

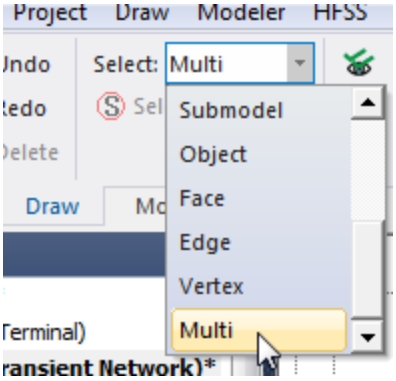
Selecting Multi

The Select Multi mode permits you to select a mixture of objects, faces, vertices, or edges, depending on where you click. This mode can be very useful in conjunction with Measure Mode,

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for measuring the distances between different entities. Enter the *Multi* selection mode using one of the following methods:

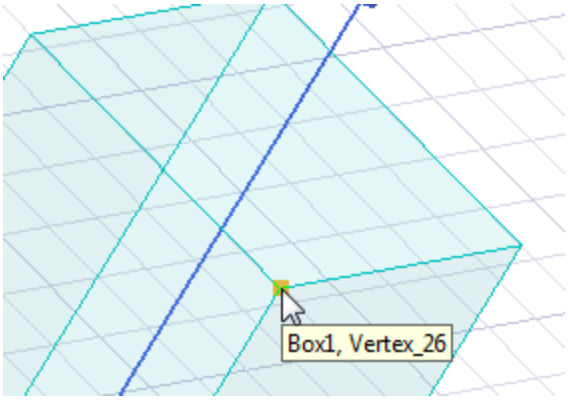
- With the Modeler window active, press the **M** shortcut key.
- Right-click in the Modeler window and choose **Selection Mode** > **Multi** from the shortcut menu.
- From the menu bar, click **Edit** > **Selection Mode**> **Multi**.
- On the **Draw** or **Model** ribbon tabs, choose **Multi** from the **Select** drop-down menu.



With Multi mode active:

- To select a vertex, click near a vertex, within 10 pixel radius.
- To select an edge, click near an edge (and at least 10 pixels away from any vertex).
- To select an object, click little farther from any edge (between 10 and 20 pixels away).
- To select a face, click anywhere else on the interior of a face.

As you hover the cursor over an entity, a Tooltip indicates the type/ID of the entity (object name in the case of objects, Face_id in the case of faces, and so on). For example, this feature helps you to distinguish between the selection of the face of a sheet object versus a sheet object.

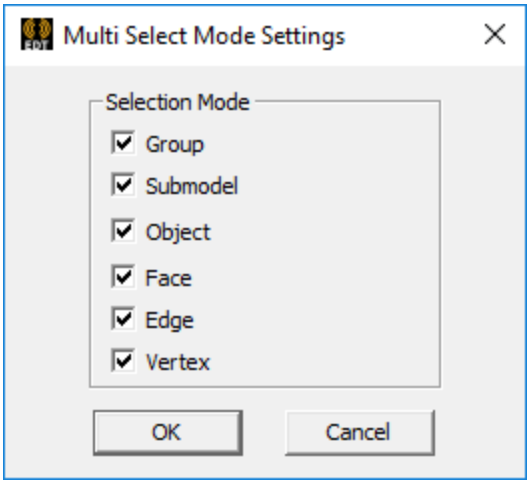


By holding down the **Ctrl** key, you can make multiple selections.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controlling the Selection in Multi Mode

You can control the behavior of this selection mode. From the menu bar, click **Edit** > **Selection Mode** > **Multi Mode Settings**. This action displays a dialog box with check boxes for enabling the selection of specific types of entities (Group, Submodel, Object, Face, Edge, and Vertex):



Clear a box filters out the associated item, preventing that type of entity from being selected while the *Multi* selection mode is active.

Selecting Groups and Submodels

With the *Group* selection mode, the parent group of selected object is selected. Selecting a group highlights all children under the group in the Modeler window. If you click an object that has not been assigned to a group, nothing is selected. To select multiple groups, hold the **Ctrl** key as you click the groups.

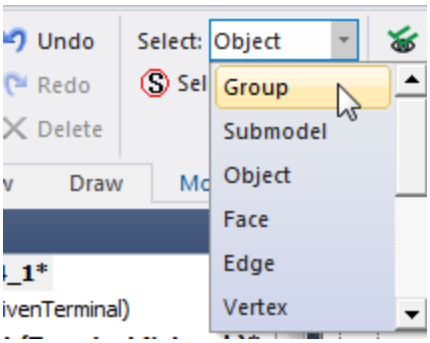
The *Group* selection mode is only available when the model contains at least one defined group.

Use one of the following methods of setting the selection mode to *Group*:

- From the menu bar, click **Edit** > **Selection Mode** > **Groups**.
- With the Modeler window active, press the **G** shortcut key.
- Right-click in the Modeler window and choose **Selection Mode** > **Groups** from the shortcut menu.
- On the **Draw** or **Model** ribbon tabs, choose **Group** from the **Select** drop-down menu. You

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

may have to scroll up to see this option.



If your design contains submodels, you can set the selection mode to *Submodel*:

- From the menu bar, click **Edit** > **Selection Mode** > **Submodels**.
- With the Modeler window active, press the **U** shortcut key.
- Right-click in the Modeler window and choose **Selection Mode** > **Submodels** from the shortcut menu.
- On the **Draw** or **Model** ribbon tabs, choose **Submodel** from the **Select** drop-down menu. You may have to scroll up to see this option.

Clearing a Selection

To clear an object, face, edge, or vertex selection, do one of the following:

- Click in an empty area of the Modeler window.
- To clear an object selection, click a point away from the object name in the History Tree.
- From the menu bar, click **Edit** > **Deselect All**.
- Press **Shift+Ctrl+A**.

The items are no longer selected.

Selecting the Face, Edge, Vertex, or Object Behind

To select the face, edge, vertex, or object, behind another selected face, edge, vertex, or object, do one of the following:

- From the menu bar, click **Edit** > **Next Behind**.
- Right-click in the Modeler window and choose **Next Behind** from the shortcut menu.
- Press the **B** shortcut key.
- Press **Ctrl+B**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> The desired entity must be behind the point where you clicked, not just behind the initially selected face, edge, vertex, or object.

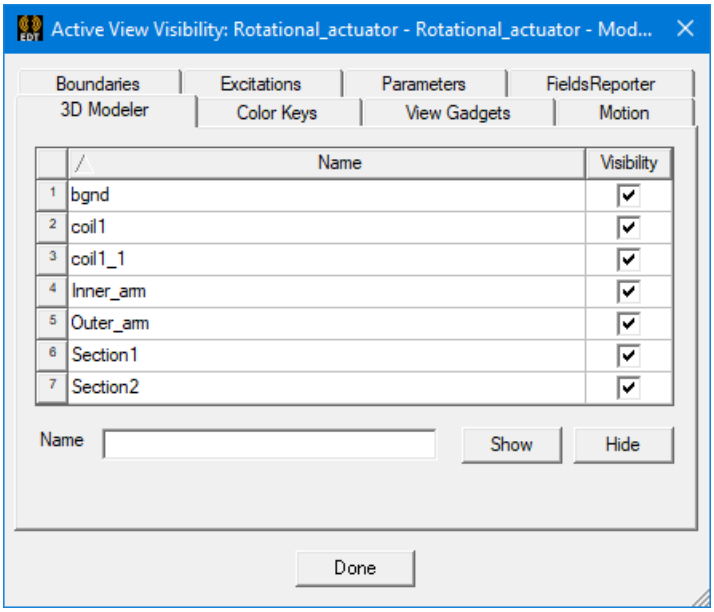
This option is useful when you are trying to select a face, edge, vertex, or object that is in the interior of a model, or when you do not want to change the model viewpoint to select an item.

Selecting Objects and Surfaces That Lie Inside Other Objects

To select objects and faces that lie inside other objects (such as an object that lies within an air box, a conductive shield, or the background object), do one of the following:

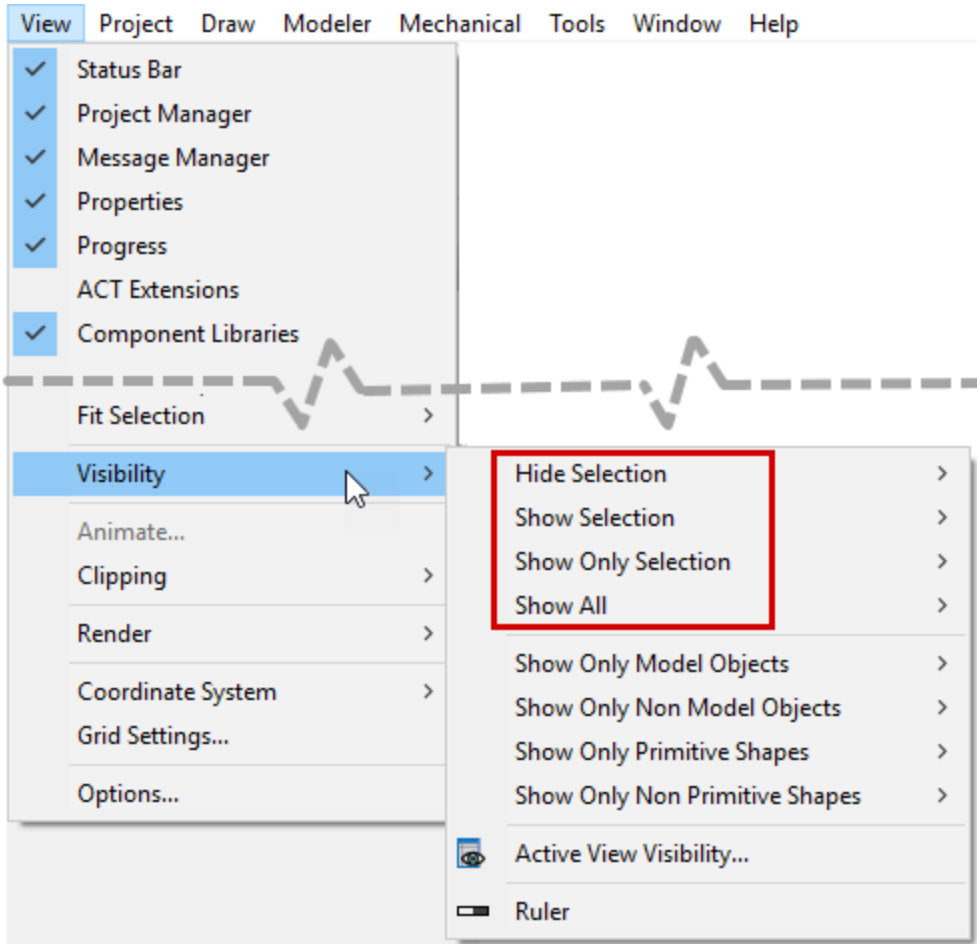
- From the menu bar, **View** > **Visibility** > **Active View Visibility...** to access the *Active View Visibility* dialog box. Then, under the *3D Modeler* tab, use the check boxes in the **Visibility** column to control which objects are visible and which are hidden.

  This method, and the method that follows, is useful when you want to select objects using the mouse, since the mouse cannot select invisible objects.

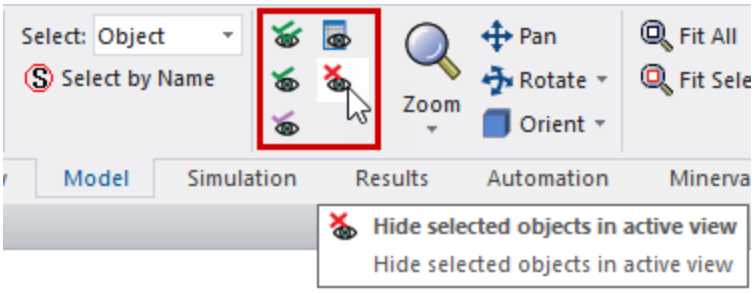


- Make the objects on the outside of the model invisible using the menu bar's other **View** > **Visibility** > commands:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



or the same commands on the **View, Draw,** or **Model** ribbon tabs:



- Use the **Select by Name** command to select Objects or Faces inside the model.
- Use the **Next Behind** command in the *Edit* menu or Modeler shortcut menu, or use the **B** shortcut key. This command selects the object that lies behind the one you initially selected. This command does nothing if no objects have previously been selected or if no object is behind the one you initially selected at the location you clicked.

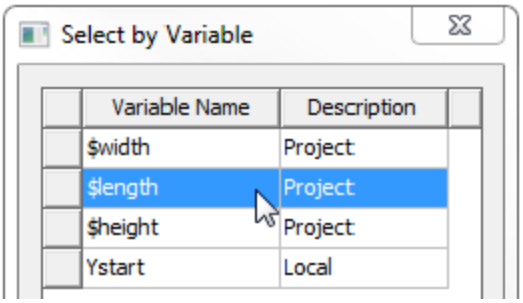
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting Objects by Variable

You can select an object based on a variable that affects it. If your design includes variables, do so as follows:

1. From the menu bar, click **Edit** > **Select Objects** > **By Variable**.

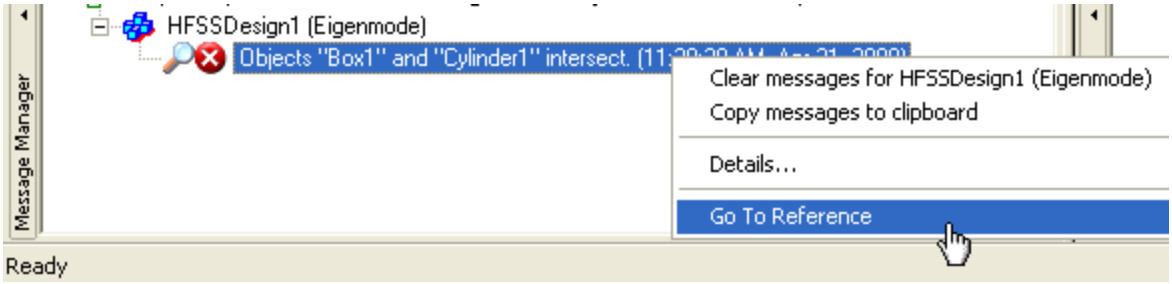
   The *Select by Variable* dialog box appears, which lists the variables in your design. Both columns are sortable by clicking the header. You can resize and move the dialog box. When you next open it, it remembers that size and location.



2. Select the variable of interest and click **OK**.

   The dialog box closes, and the object affected by the variable is highlighted in the Modeler window.

   The Message window contains a reference that you can select and use to go to the affected object.



   If you execute the command again, without clearing the current selection(s), the additional object can be highlighted. You can resize and move the dialog box.

Assigning Coordinates to New Objects

When you insert and draw new 3D, 2D, or 1D objects in model, you may need to define coordinate systems, locations of points, distances between objects, and other geometry items.

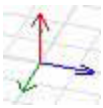
- Setting the Reference Point
- Defining Cartesian Coordinates

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Defining Spherical Coordinates
- Defining Relative Coordinates
- Defining Absolute Coordinates

Setting the Reference Point

When you draw objects, the cursor's location is always relative to a reference point. The reference point is displayed with a mini xyz-axis:



To change the reference point:

1. Select the drawing command to use.

   The Measure Data dialog box opens. As you move the cursor over the Modeler window, the top line in the *Measure Data* dialog box shows the coordinates of the current reference point.

2. Move the cursor to the desired reference point and press **Ctrl+click** or right-click and select **Set Reference Point** from the short cut menu.

   This action moves the reference point marker to the new location. The *Measure Data* dialog box updates. The coordinates boxes in the Status Bar change to accept Relative distance information. If you prefer, rather than setting the reference point with the cursor, you can press **Tab** to jump to a coordinate box in the status bar, where you can specify the coordinates numerically. Navigate among the three coordinate boxes by pressing the **Tab** key again.

Defining Cartesian Coordinates

When you draw an object, define a point using Cartesian coordinates by typing the point's x, y, and z distances from the origin in the **X, Y,** and **Z** text boxes, respectively. When defining a second point, specify its x, y, and z distances from the previously selected point in the **dX, dY,** and **dZ** text boxes, respectively.

1. Select the desired drawing command.
2. Select **Cartesian** from the drop-down menu in the Status Bar.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Type the point's x, y, and z coordinates in the **X, Y,** and **Z** text boxes.

   > **Tip:**
   >
   > Press **Tab** to move from one coordinate text box to the next. Press **Ctrl+Tab** to move to the previous coordinate text box.

   - Alternatively, click the point in the Modeler window.
4. When drawing objects other than polylines and helices, the second point you select is relative to the first point. Type the second point's x, y, and z distances from the previously selected point in the **dX, dY,** and **dZ** text boxes, respectively.

Defining Cylindrical Coordinates

To define a point using cylindrical coordinates, specify the point's radius, measured from the origin, in the **R** text box, the angle from the x-axis in the **Theta** text box, and the distance from the origin in the z direction in the **Z** text box. When defining a second point, specify its distance from the previously selected point in the in **dR, dTheta,** and **dZ** text boxes.

1. After clicking the desired drawing command, select **Cylindrical** from the pull-down menu in the Status Bar.
2. Type the point's r, theta, and z coordinates in the **R, Theta,** and **Z** boxes.

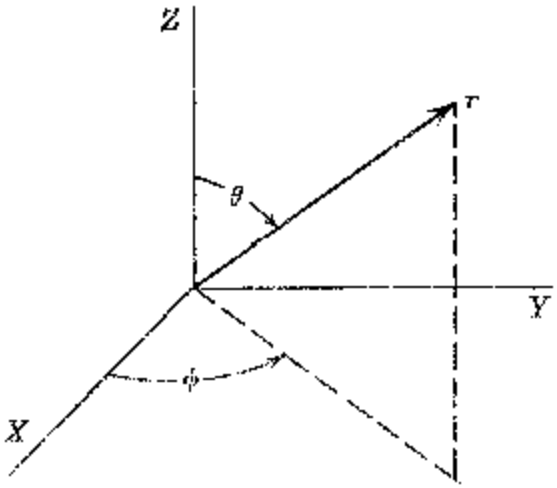
   > **Tip:**
   >
   > Press **Tab** to move from one coordinate text box to the next. Press **Ctrl+Tab** to move to the previous coordinate text box.

   - Alternatively, click the point in the Modeler window.
3. When drawing objects other than polylines and helices, the second point you select is relative to the first point. Type the second point's distance from the previously selected point in the **dR, dTheta,** and **dZ** text boxes.

Defining Spherical Coordinates

To define a point in spherical coordinates, specify the point's radius, measured from the origin, in the **Rho** text box, the angle from the positive z-axis in the **Theta** text box, and the angle from the positive x axis in the **Phi** text box. When selecting a second point, specify its distance from the previously selected point in the in **dRho, dTheta,** and **dPhi** text boxes.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> Even though you are inputting spherical coordinates, all data is internally stored in Cartesian coordinates.

To define a point in spherical coordinates:

1. After clicking the desired drawing command, select **Spherical** from the drop-down menu in the status bar.
2. Type the point's r, theta, and phi coordinates in the **Rho, Theta,** and **Phi** text boxes in the Status Bar.

   > **Tip:**
   >
   > Press **Tab** to move from one coordinate text box to the next. Press **Ctrl+Tab** to move to the previous coordinate text box.

   - Alternatively, click the point in the Modeler window.
3. When drawing objects other than polylines and helices, the second point you select is relative to the first point. Type the second point's distance from the previously selected point in the **dRho, dTheta,** and **dPhi** text boxes.

Using Absolute Coordinates

When entering a point's location, you can specify it using *absolute* or *relative* coordinates. Absolute coordinates are relative to the working coordinate system's origin (0, 0, 0). This entry mode is the default setting for the first point you select after clicking a drawing command. Relative coordinates are relative to the reference point, or the previously selected point.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To enter a point's absolute coordinates:

1. Click the desired drawing command.
2. Select **Absolute** from the **Absolute/Relative** drop-down menu in the Status Bar.
3. Specify the point's coordinates in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the appropriate text boxes in the Status Bar.

> **Note:**
>
> When drawing objects other than polylines and helices, by default, the second point you select is relative to the first point; **Relative** is automatically selected in the **Absolute/Relative** drop-down menu in the Status Bar. Be sure to select **Absolute** from this drop-down menu if you want the second point to be relative the origin of the working coordinate system.

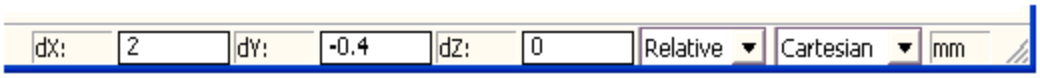
Using Relative Coordinates

When entering a point's location, you can specify it using *absolute* or *relative* coordinates. Relative coordinates are relative to the reference point or the previously selected point. Absolute coordinates are relative to the working coordinate system's origin (0, 0, 0).

To enter a point's relative coordinates:

1. Click the desired drawing command.
2. Select **Relative** from the **Absolute/Relative** drop-down menu in the Status Bar.

   When you are in *Relative* mode, the text boxes for a coordinate show an "d" before the coordinate description, to indicate "distance from" the working reference. For example:

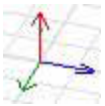


3. Specify the point's coordinates in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the appropriate text boxes in the Status Bar.

Choosing the Movement Mode

When drawing objects, the cursor's location is always relative to a reference point. The reference point is displayed with a mini xyz-axis when you click a drawing command:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To change the reference point, move the cursor to the desired point and **Ctrl+click** the point.

When creating 1D, 2D, or 3D geometry graphically (that is, by clicking points instead of by defining coordinates numerically), points that you can select are dependent on the current *Movement Mode* setting. You can move the cursor and place a point at one of the following locations:

- In the same plane as the reference point (In Plane movement mode)
- Perpendicular to the reference point (Out of Plane movement mode)
- If an object is present, you can snap to one of its points – any snapable point in 3D space (3D movement mode).
- Along the X-axis of the working coordinate system (CS)
- Along the Y-axis of the working CS
- Along the Z-axis of the working CS

Drawing commands that you choose may temporarily override the current movement mode. For example, when you draw a box or cylinder, the first two points you click (defining the base) may be in the drawing plane, but the third (defining the height) must be out of plane relative to the base. In this case, you do not have to manually change the movement mode; it is automatically overridden for the third click.

> **Note:**
>
> For all six movement modes covered in this section, the following rules apply regarding mode persistence:
>
> - If you change the movement mode *before* executing a drawing command, the new mode setting will be retained until you change it again.
> - If you change the movement mode *after* executing a drawing command, it will be treated as a temporary override. Upon completion of the drawing operation, the mode will revert to the previous setting.

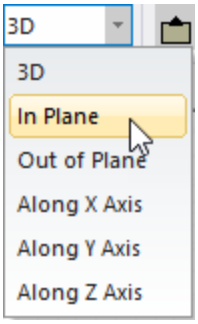
Moving the Cursor In Plane

Use the *In Plane* movement mode to specify points of objects being drawn that are *on the same plane* as the reference point:

Before or after clicking a desired drawing command, change the movement mode using one of the following methods:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

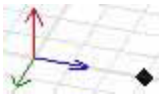
- From the menu bar, click **3D Model** > **Movement Mode** > **In Plane**.
- On the **Draw** ribbon tab, choose **In Plane** from the **Movement mode** drop-down menu:



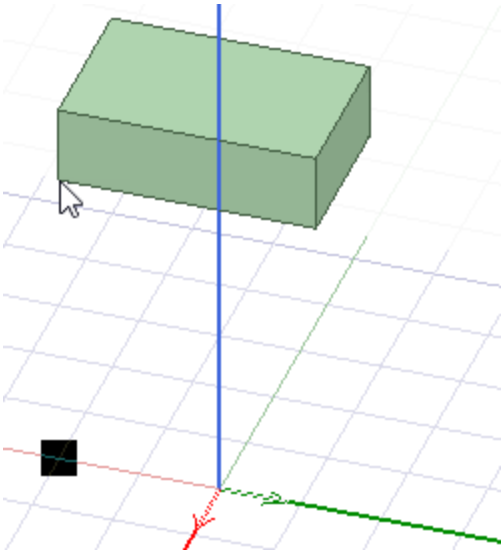
The following additional method is available only after clicking a drawing command:

- Right-click in the Modeler window and choose **Movement Mode** > **In Plane** from the shortcut menu.

The next point you select will be on the same plane as the reference point. The cursor's location is displayed with a black diamond that indicates where it will snap to the drawing plane grid.



If you click a snapping point on an object in 3D space (that is, one not on the drawing plane), the point location is projected to the drawing plane along a perpendicular vector. In the following example, a filled black square indicates the projected snapping point:



The symbol used for projected point matches the type of snapping point on the object (vertex, edge center, face center, and so on).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

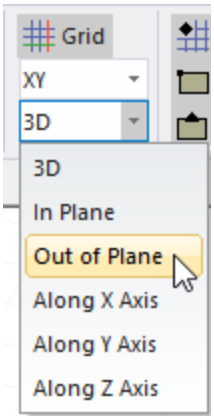
The *Measure Data* window and the coordinate text boxes in the Status Bar provide precise cursor location information.

Moving the Cursor Out of Plane

Use the *Out of Plane* movement mode to specify points of objects being drawn that are along a vector *perpendicular* to the drawing plane and intersecting it at the reference point:

Before or after clicking a desired drawing command, change the movement mode using one of the following methods:

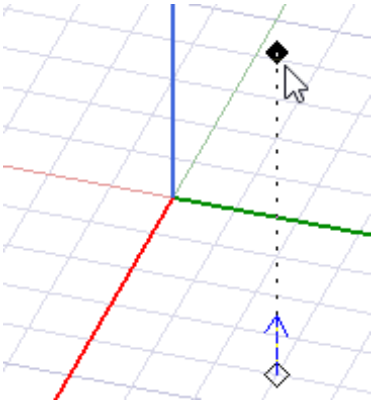
- From the menu bar, click **3D Model** > **Movement Mode**> **Out of Plane**.
- On the **Draw** ribbon tab, choose **Out of Plane** from the **Movement mode** drop-down menu:



The following additional method is available only after clicking a drawing command:

- Right-click in the Modeler window and choose **Movement Mode** > **Out of Plane** from the shortcut menu.

A dashed line perpendicular to the drawing plane is displayed, and a filled black diamond indicates the cursor's location:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

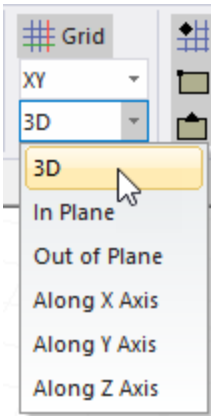
The *Measure Data* window and the coordinate text boxes in the Status Bar provide precise cursor location information.

Moving the Cursor in 3D Space

To snap to drawing plane grid points or to object snapping points anywhere in 3D space, use the *3D* movement mode.

Before or after clicking a desired drawing command, change the movement mode using one of the following methods:

- From the menu bar, click **3D Model** > **Movement Mode** > **3D**.
- On the **Draw** ribbon tab, choose **3D** from the **Movement mode** drop-down menu:

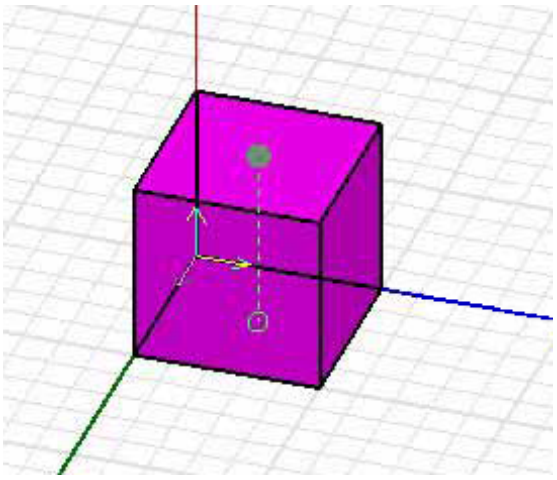


The following additional method is available only after clicking a drawing command:

- Right-click in the Modeler window and click **Movement Mode** > **3D** from the shortcut menu.

If one of an object's snapping points (vertex, edge center, face center, and so on) is within snapping range of the cursor, the geometry you are drawing will snap to that point. The symbol will indicate the type of snapping point. In the following example, the cursor will snap to the face center point of the boxes top face:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If an object is not within snapping range of the cursor, the *3D* movement mode is identical to the *In Plane* mode. That is, the geometry being drawn will snap to a grid point on the drawing plane.

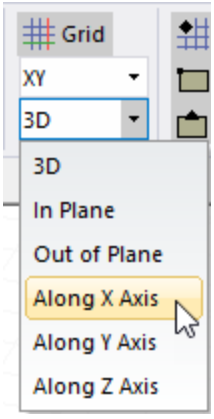
In either case, the *Measure Data* window and the coordinate text boxes in the Status Bar provide precise cursor location information.

Moving the Cursor Along the X-Axis

Use this movement mode to constrain the cursor to a moving only in the +/- X direction relative to the reference point. Movement is constrained to the X direction as defined by the working coordinate system, not necessarily the global X direction. The line of motion passes through the reference point, so it may be parallel to the X-Axis and not necessarily along the actual X-axis.

Before or after clicking a desired drawing command, change the movement mode using one of the following methods:

- From the menu bar, click **3D Model** > **Movement Mode** > **Along X Axis**.
- On the **Draw** ribbon tab, choose **Along X Axis** from the **Movement mode** drop-down menu:



The following additional methods are available only after clicking a drawing command:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Press and hold the **X** key while clicking the next point.
- Right-click in the Modeler window and choose **Movement Mode** > **Along X Axis** from the shortcut menu.

The next point you select will be on the same plane as the reference point and in the positive or negative X direction relative to the reference point.

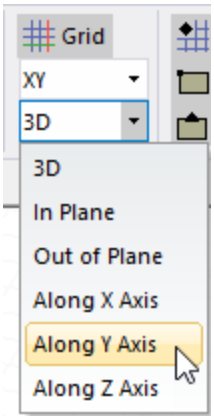
The *Measure Data* window and the coordinate text boxes in the Status Bar provide precise cursor location information.

Moving the Cursor Along the Y-Axis

Use this movement mode to constrain the cursor to a moving only in the +/- Y direction relative to the reference point. Movement is constrained to the Y direction as defined by the working coordinate system, not necessarily the global Y direction. The line of motion passes through the reference point, so it may be parallel to the Y-Axis and not necessarily along the actual Y-axis.

Before or after clicking a desired drawing command, change the movement mode using one of the following methods:

- From the menu bar, click **3D Model** > **Movement Mode** > **Along Y Axis**.
- On the **Draw** ribbon tab, choose **Along Y Axis** from the **Movement mode** drop-down menu:



The following additional method is available only after clicking a drawing command:

- Right-click in the Modeler window and choose **Movement Mode** > **Along Y Axis** from the shortcut menu.
- Press and hold the **Y** key while clicking the next point.

The next point you select will be on the same plane as the reference point and in the positive or negative Y direction relative to the reference point.

The *Measure Data* window and the coordinate text boxes in the Status Bar provide precise cursor location information.

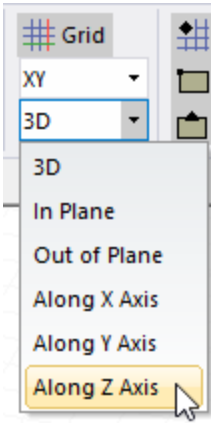
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Moving the Cursor Along the Z-Axis

Use this movement mode to constrain the cursor to a moving only in the +/- Z direction relative to the reference point. Movement is constrained to the Z direction as defined by the working coordinate system, not necessarily the global Z direction. The line of motion passes through the reference point, so it may be parallel to the Z-Axis and not necessarily along the actual Z-axis.

Before or after clicking a desired drawing command, change the movement mode using one of the following methods:

- From the menu bar, click **3D Model** > **Movement Mode** > **Along Z Axis**.
- On the **Draw** ribbon tab, choose **Along Z Axis** from the **Movement mode** drop-down menu:



The following additional methods are available only after clicking a drawing command:

- Press and hold the **Z** key while clicking the next point.
- Right-click in the Modeler window and choose **Movement Mode** > **Along Z Axis** from the shortcut menu.

The next point you select will be on the same plane as the reference point and in the positive or negative Z direction relative to the reference point.

The *Measure Data* window and the coordinate text boxes in the Status Bar provide precise cursor location information.

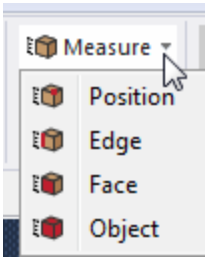
Measure Modes

The Measure modes lets you measure the position, length, area, and volume of objects. With two faces selected, with two edges selected, or with an edge and a face selected, the Measure Mode displays the angle and distance between them. The Measure Position mode dynamically measures the distance between a reference point and the cursor location. You can pause

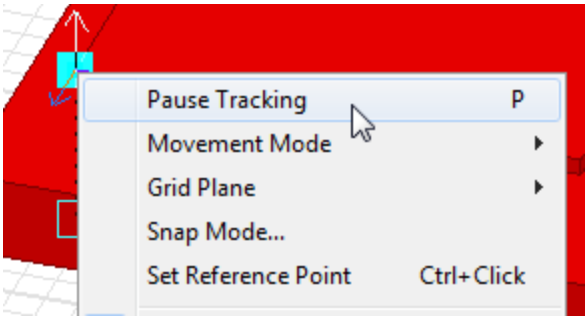
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

dynamic tracking in order to copy text information from the *Measure Data* window, and then Resume dynamic updating.

1. To access a Measure mode, do one of the following:
   - From the menu bar, click **Modeler** > **Measure** > [**Position, Edge, Face,** or **Object**].
   - Right-click in the Modeler window and choose **Measure** > [**Position, Edge, Face,** or **Object**] from the short-cut menu.
   - On the **Draw** ribbon tab, choose **Position, Edge, Face,** or **Object** from the **Measure** drop-down menu.



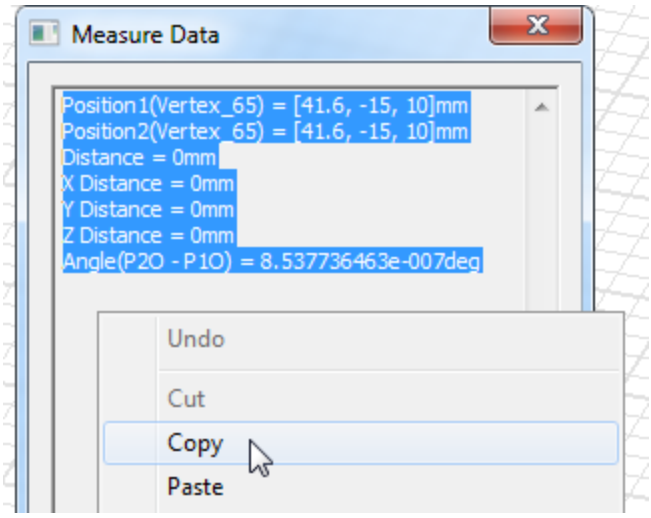
2. Select items to measure as described below under Measurement Modes and Data.
3. Optionally, to *Pause* the dynamic tracking, do one of the following:
   - Right-click in the Modeler window and select **Pause Tracking**:



   - Press **P**.

   While paused, you can move the cursor without changing the *Measure Data* window contents. You can also copy and paste the data to a text file:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To resume measurement tracking, do one of the following:

- Right-click in the Modeler window and choose **Resume Tracking**.
- Press **P** again.

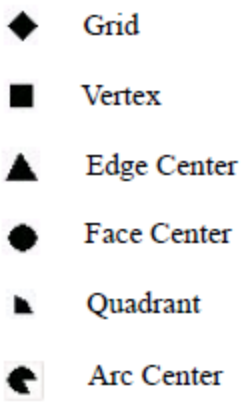
4. To exit the Measure mode, do one of the following:
   - Click **Close** on the *Measure Data* window.
   - Press **Esc**.

Measurement Modes and Data:

- **Position:** Use the following procedure to measure the position of points and the distance between them:
  1. Click a snap point anywhere on the model, which becomes *Position 1* (the measurement reference point).
  2. Hover over another snap point but do not click the mouse (or **Ctrl+click** the second point to redefine the reference point but retain the measurement data).

     The following symbols indicate the available types of snap points:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> As you move the cursor, the Measure Data window displays the reference point, current cursor location, and measurement information. Clicking on a new vertex or other snap point updates the reference to the new location and resets the measurement results to zero. To resent the reference point location without clearing the measurement data, **Ctrl+click** the point.

The *Measure Data* window gives you the following information:

- Position1 coordinates
- Position2 coordinates
- Distance between Position1 and Position2
- X Distance: X-component of the Distance.
- Y Distance: Y-component of the Distance.
- Z Distance: Z-component of the Distance.
- Angle: The angle of the line connecting Position1 and Position2

- **Edge:** The *Measure Data* window gives you the following information:
  - Edge name
  - Length
  - Reference coordinate system
  - Start Vertex Position (with Vertex name)
  - End Vertex Position (with Vertex name)
  - Additional data for circles and arcs only:
    - Arc Center Position
    - Arc Radius
    - Arc Diameter
    - Plane

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Face:** Each item listed below only appears in the *Measure Data* window when applicable to the selected face.
  - Face ID
  - Area
  - Perimeter
  - Cylinder Radius or Sphere Radius
  - Cylinder Diameter or Sphere Diameter
  - Plane (orientation of the plane – the face may be parallel to the listed plane and does not necessarily lie on it)
  - Plane Normal (normal vector for planes not aligned with a global or relative coordinate system)
  - Reference coordinate system
  - Center Position
  - No. of Loops
  - No. of Edges
  - No. of CoEdges
  - No. of Vertices
- **Object:** Each item listed below only appears in the *Measure Data* window when applicable to the selected object.
  - Volume(with *ObjectName* — zero for shell objects)
  - Area(with *ObjectName* — total surface area for solids and shells)
  - No. of Lumps
  - No. of Shells
  - No. of Faces
  - No. of Loops
  - No. of Edges (the total number of unique edges)
  - No. of CoEdges (the total number of coincident edges, where adjacent faces intersect)
  - No. of Vertices

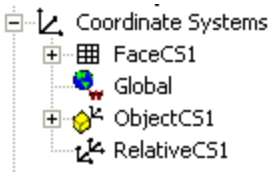
Setting Coordinate Systems

The modeler has four types of coordinate systems (CS) that enable you to easily orient new objects:

- Global CS
- Relative CS
- Face CS
- Object CS

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Every coordinate system has an x-axis that lies at a right angle to a y-axis, and a z-axis that is perpendicular to the xy plane. The origin (0,0,0) of every CS is located at the intersection of the x-, y-, and z-axes. The default Global coordinate system and any additional coordinate systems that you create for a project appear in the History Tree of the Modeler window.



- The *Global* coordinate system (CS) is the fixed, default CS for each new project. It cannot be edited or deleted.
- A *Relative CS* is user-defined. Its origin and orientation can be set relative to an existing CS. Relative CSs enable you to easily draw objects that are located relative to other objects. If you modify a relative CS, all objects drawn on that CS will be affected and change position and/or orientation accordingly. You can define a relative CS to be *Offset* and/or *Rotated* from an existing CS. This feature provides a way for objects made of the same anisotropic materials to have different orientations.

  When you set a new relative coordinate system, you specify whether to express the coordinates as Absolute or Relative Coordinates. Absolute uses the specified values in terms of the global coordinate system. Relative interprets the values as differences from the current working CS.

  You have choices for expressing the coordinates as Cartesian, Cylindrical, or Spherical. These are evaluated as Cartesian for the coordinate system properties.

- A *Face CS* is also user-defined. Its origin is specified on a planar object face. Face CSs enable you to easily draw objects that are located relative to an object's face.
- An *Object CS* is user-defined as attached to a specific object.

Switch between global, relative, object and face CSs by changing the *working CS*. Simply click the CS you want to use in the History Tree. The working CS is indicated by a red *W* that appears at the lower-left corner of the CS name in the History Tree (as shown on Global in the figure above). The docked **Properties** window lists the CS associated with an object as the *Orientation*. By default, the orientation is Global, but if you have created the object under a different coordinate system, that CS will be shown for the orientation. You can click on the current orientation to see a drop-down menu of other orientations that you can assign for an object.

User-defined CSs are saved with the active project. When you open a project, the CS designated as working CS when you last saved it is active.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Working Coordinate System

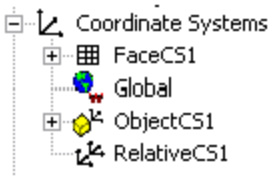
The working coordinate system (CS) is the current CS with which objects being drawn are associated. The working CS can be the Global CS or a user-defined Relative, Object, or Face CS. Select the working CS by clicking its name in the History Tree, or follow this procedure:

1. Click **Modeler** > **Coordinate System** > **Set Working CS**.

   The *Select Coordinate System* dialog box appears.

2. Click a CS in the list.
3. Click **Select**.

   A red *W* appears at the lower-left corner of the CS name in the History tree, indicating that it is the working CS. In this following figure the Global is the working CS.



   Objects that you draw hereafter will be associated with the CS you selected.

Creating a Relative Coordinate System

When creating a relative CS, you have the following options:

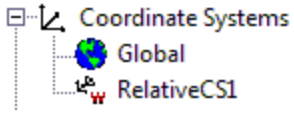
- You can create an offset relative CS, that is, a relative CS whose origin lies a specified distance from another CS's origin. By moving a CS's origin, you can enter coordinates relative to an existing object, without having to add or subtract the existing object's global coordinates.
- You can create a rotated relative CS, that is, a relative CS whose axes are rotated away from another CS's axes. By rotating the axes of a CS, you can easily add an object that is turned at an angle relative to another object.
- You can also create a relative CS that is both offset and rotated.
- You can click **Tools** > **Options** > **General Options** to display the *Options* dialog box.

  Under the *3D Modeler* > *Drawing* options, choose your preferred *Relative Coordinate System Creation Mode*. The **Axis/Position** option uses the cursor or coordinate entry boxes in the Status bar to define the CS. The **Euler angle** option causes the *Set Origin, Set Euler Angles,* or *Set Origin and Euler Angles* dialog box to display (depending on the type of Offset CS you are creating). You define the origin and angles numerically in the dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also press **F3** or **F4** after clicking a *RelativeCS* command to switch between the two entry modes on the fly.

After you have created a relative CS it appears in the History Tree:

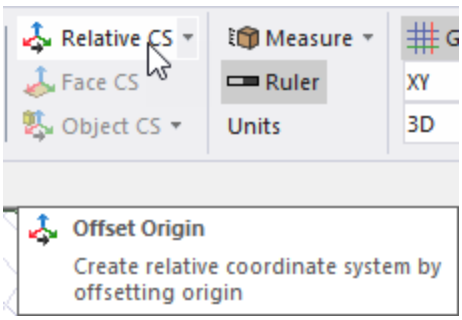


Selecting the CS in the History Tree causes the docked *Properties* window to show the CS properties.

Creating an Offset Relative CS

To create a relative CS with an origin that lies a specified distance from another CS's origin:

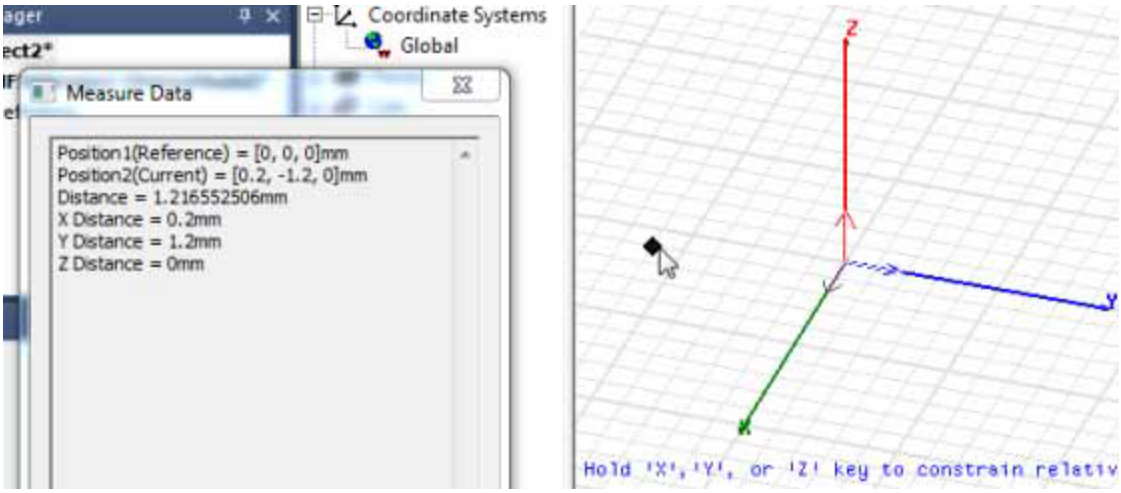
1. In the History Tree, click the CS upon which you want to base the new relative CS, making it the working CS.
2. From the menu bar, click **Modeler** > **Coordinate System** > **Create** > **Relative CS** > **Offset** or, on the **Draw** ribbon tab, click **Relative CS**. (You do not have to access the drop-down menu because *Offset* is the default option for a *Relative CS* in the ribbon:



3. Depending on your *Relative Coordinate System Creation Mode* choice (**Tools** > **Options** > **General Options**, *3D Modeler* > *Drawing*), or whether you press **F3** or **F4**, you can select the origin in one of the following two ways:
    - **Axis/Position** option selected, or **F3** pressed – Use one of the following two coordinate entry methods:
        - Specify the coordinates graphically (click to select):

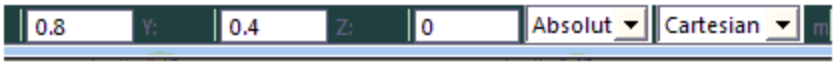
          Use the cursor to click the point. You can see the coordinate information in the *Measure Data* window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To select a point that does not lie in the current plane, use the *Movement Mode* commands or choose a different drawing plane.

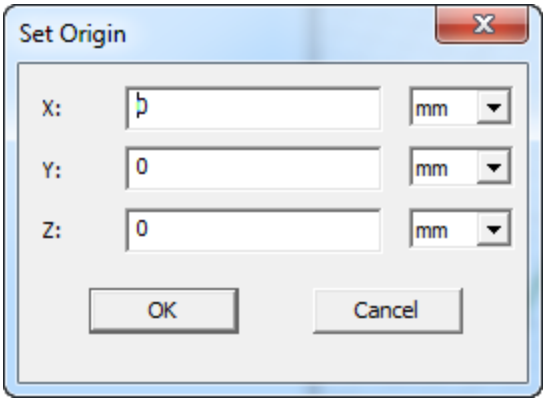
- Specify the coordinates numerically (using the coordinate entry text boxes in the Status Bar):



  a. In the Status Bar, choose the type of coordinate system from the rightmost drop-down menu (**Cartesian, Cylindrical,** or **Spherical**).
  b. Select either Relative or Absolute coordinates from the other drop-down menu.
  c. Type the CS origin coordinates in the **X, Y** and **Z** (or **dX, dY,** and **dZ**) text boxes.

- **Euler angle** option selected, or **F4** pressed:

  In the *Set Origin* dialog box, specify the **X, Y,** and **Z** coordinates and units and click **OK**.



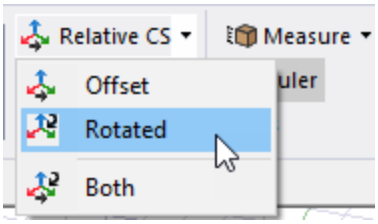
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The new relative CS is created. Its origin has moved from the previous working CS, but its axes remain the same. It is listed in the History Tree under **Coordinate Systems**. It automatically becomes the working CS; objects that you draw hereafter will be based on the coordinates of this relative CS. Default planes are created on its xy, yz, and xz planes.

Creating a Rotated Relative CS

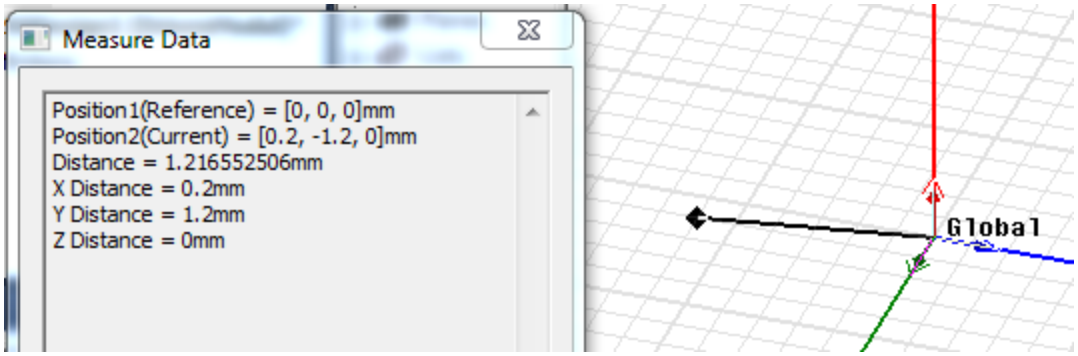
To create a new relative CS with its axes rotated away from another CS's axes:

1. In the History Tree, select the CS upon which you want to base the new relative CS, making it the working CS.
2. Click **Modeler** > **Coordinate System** > **Create** > **Relative CS** > **Rotated** or, on the **Draw** ribbon tab, click **Relative CS** > **Rotated**:



.

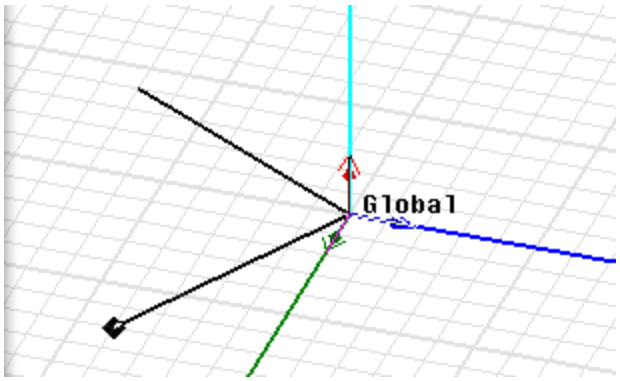
3. Depending on your *Relative Coordinate System Creation Mode* choice (**Tools** > **Options** > **General Options** – *3D Modeler> Drawing*), or whether you press **F3** or **F4**, you can specify the axes in one of the following two ways:
   - **Axis/Position** option selected, or **F3** pressed – Use one of the following coordinate entry methods:
     - Specify the X-axis and XY plane graphically (click to select), as follows:
       a. Use the cursor to click a point along the desired X-axis. You can see the coordinate information in the *Measure Data* window.



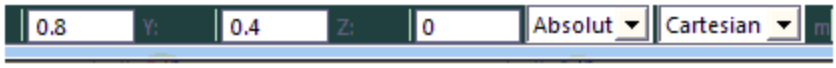
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To select a point that does not lie in the current plane, use the *Movement Mode* commands or choose a different drawing plane.

b. In the same manner, select any point lying on the XY plan that's not on the X-axis.



- Enter the coordinates numerically in the Status Bar, as follows:



a. In the Status Bar, choose the type of coordinate system from the rightmost drop-down menu (**Cartesian, Cylindrical,** or **Spherical**).

b. Select either Relative or Absolute coordinates from the other drop-down menu.

c. Type the coordinates of a point on the desired X-axis in the X, Y and Z (or dX, dY, and dZ) text boxes and press **Enter**.

d. In the same manner, type the coordinates of a point lying on the XY plane, but not on the X-axis, and press **Enter** again.

You do not need to specify the Z-axis. It is automatically calculated to be perpendicular to the new XY plane.

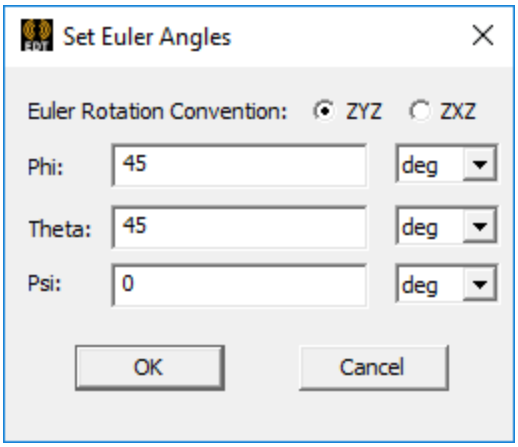
- **Euler angle** option selected, or **F4** pressed:

a. In the *Set Euler Angles* dialog box, select the preferred **Euler Rotation Convention** (**ZYZ** or **ZXZ**).

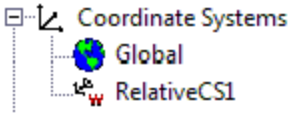
b. Specify the **Phi, Theta,** and **Psi** values and select units for each from the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

drop-down menus.



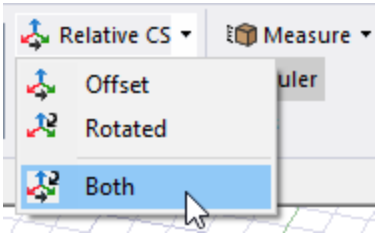
The new relative CS is created. It has the same origin as the previous working CS, but its axes are rotated. It is listed in the History Tree under *Coordinate Systems*. It automatically becomes the working CS; objects that you draw hereafter will be based on the coordinates of this relative CS. Default planes are created on its XY, YZ, and XZ planes.



Creating an Offset and Rotated Relative CS

To create a new relative CS that is both offset and rotated from an existing CS:

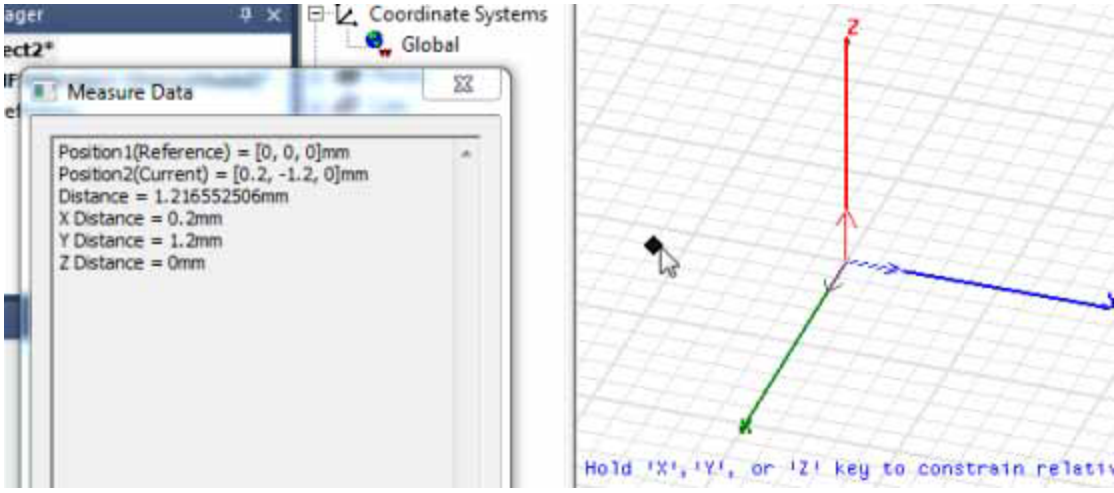
1. In the History Tree, select the CS upon which you want to base the new relative CS, making it the working CS.
2. Click **Modeler** > **Coordinate System** > **Create** > **Relative CS** > **Both** or, on the **Draw** ribbon tab, click **Relative CS** > **Both**:



3. Depending on your *Relative Coordinate System Creation Mode* choice (**Tools** > **Options** > **General Options** – *3D Modeler* > *Drawing*), or whether you press **F3** or **F4**, you can specify the origin and axes in one of the following two ways:

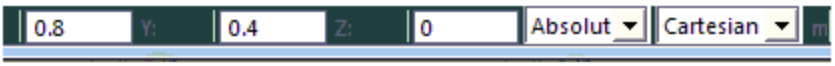
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Axis/Position** option selected, or **F3** pressed – Use one of the following coordinate entry methods:
  - Specify the Origin, X-axis, and XY plane graphically (click to select), as follows:
    a. Use the cursor to click the Origin point. You can see the coordinate information in the *Measure Data* window.



    To select a point that does not lie in the current plane, use the *Movement Mode* commands or choose a different drawing plane.

    b. In the same manner, click a point to define the X-axis.
    c. Select a third point, one lying on the XY plan that's not on the X-axis, to complete the CS definition.
  - Specify the Origin, X-axis, and XY plane numerically in the Status Bar, as follows:

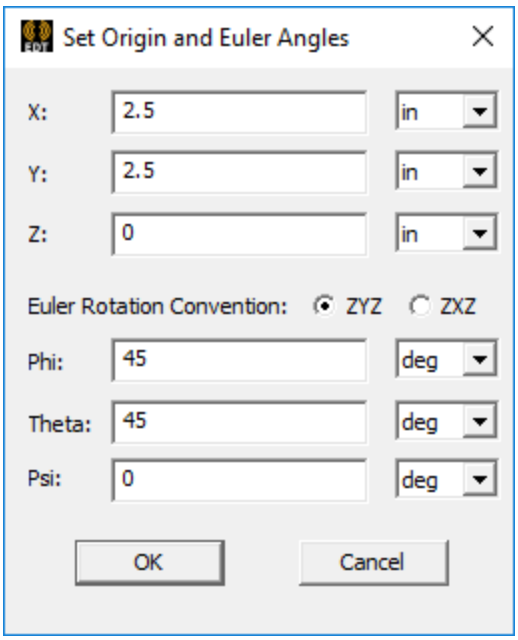


    a. In the Status Bar, choose the type of coordinate system from the rightmost drop-down menu (**Cartesian, Cylindrical,** or **Spherical**).
    b. Select either Relative or Absolute coordinates from the other drop-down menu.
    c. Type the CS origin coordinates in the **X, Y** and **Z** (or **dX, dY,** and **dZ**) text boxes.
    d. In the same manner, type the coordinates of a point on the X-axis and press **Enter**.
    e. Type the coordinates of a third point, one lying on the XY plan that's not on the X-axis, and press **Enter** to complete the CS definition.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You do not need to specify the Z-axis. It is automatically calculated to be perpendicular to the new XY plane.

- **Euler angle** option selected, or **F4** pressed:
  a. In the *Set Origin and Euler Angles* dialog box, specify the **X, Y,** and **Z** coordinates of the Origin and select units for each from the drop-down menus.
  b. Select the preferred **Euler Rotation Convention** (**ZYZ** or **ZXZ**).
  c. Specify the **Phi, Theta,** and **Psi** values and select units for each from the drop-down menus.

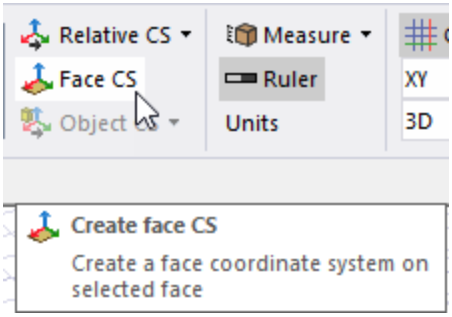


The new relative CS is created. It is listed in the History Tree under *Coordinate Systems*. It automatically becomes the working CS; objects that you draw hereafter will be based on the coordinates of this relative CS. Default planes are created on its XY, YZ, and XZ planes.

Creating a Face Coordinate System

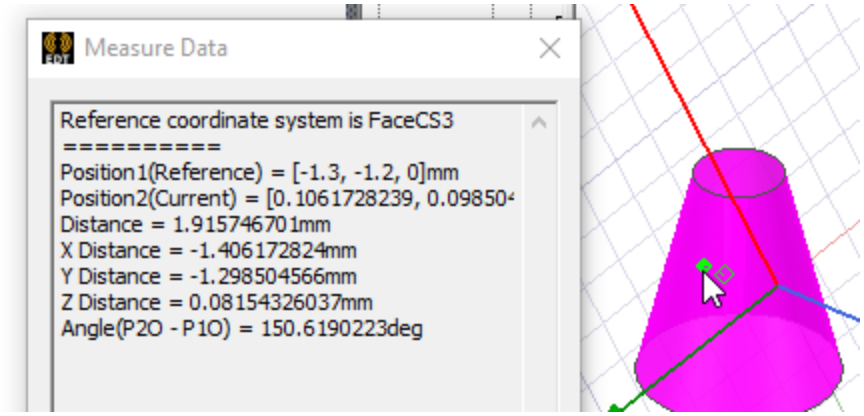
1. Select the object face upon which you want to create the face CS. The CS face can be planar or curved.
2. Click **Modeler** > **Coordinate System** > **Create** > **Face CS** or, on the **Draw** ribbon tab, click the **Create Face CS** icon.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



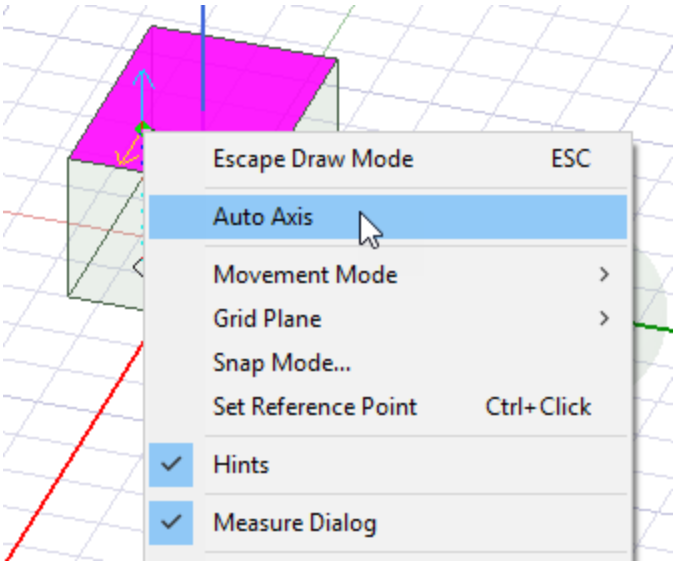
3. Select the Origin in one of the following ways:
   - Click a point on the selected face to define the Origin. If the point is not on the face, you will receive an error message.

     The *Measure Data* window shows the current cursor coordinates. The cursor shape will change to identify grid points, vertices, edge center, face center, quadrant, and arc center points, but you can click any point on the selected face.



   - Type the point's coordinates in the **X, Y,** and **Z** text boxes on the Status Bar.
4. Specify the positive X-axis direction in one of the following ways:
   - Click a point on the selected face. If the point is not on the face, you will receive an error message.
   - Type the coordinates of a point that is relative to the previously selected point in the **dX, dY,** and **dZ** text boxes, where **d** is the distance from the previously selected point.

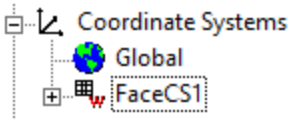
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Right-click in the Modeler window and select **Auto Axis** from the shortcut menu. The modeler selects the X-axis based on the principal curvature of the face or the dimensions of a flat face. The *axial* direction of a curved face or the long dimension of a flat face is selected as the X-axis direction.

You do not need to specify the Y or Z axes. The modeler assumes that the Z-axis is normal to the object face and the Y-axis is automatically calculated to be perpendicular to the XZ plane.

The new face CS is listed in the History Tree under *Coordinate Systems*. It automatically becomes the working CS; objects that you draw hereafter will be referenced to the coordinates of this face CS. Default planes are created on its XY, YZ, and XZ planes.



Only operations listed in the History Tree *before* the face CS's creation will affect the face CS and, in turn, affect objects dependent upon that face CS. A face CS, or any object created on it, is *not* affected by operations that occur after the face CS is created. Also see the Move CS to End command.

For example, suppose you create a box, then a face CS on a face of the box, and then a cylinder on the face CS. If you then edit the box's dimensions in the docked *Properties* window, the cylinder will move accordingly. But if you rotate the box using the **Edit** > **Arrange** > **Rotate** command, the box will move, but the cylinder will not move (because the rotation operation occurs later in the History Tree).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Automatically Creating Face Coordinate Systems

You can instruct the modeler to automatically create a new face CS every time you draw on an object's face.

1. Click **Tools** > **Options** > **General Options**.

   The *Options* dialog box appears.

2. In the tree on the left side of the dialog box, select **3D Modeler** > **Operation**.
3. In the *Coordinate System* section, select **Automatically switch to face coordinate system**.
4. Click **OK**.

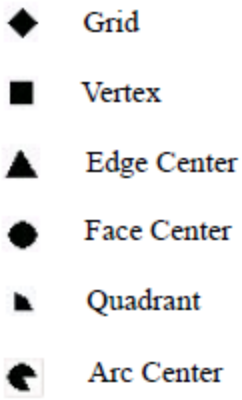
Now, when you select a face, and then click a drawing command, a new face CS will be created on the selected face. The modeler automatically sets the new face CS as the working CS, and the object you draw is oriented according to the new face CS.

> **Note:**
>
> The modeler will not automatically create a new face CS if a face CS has already been assigned to the selected face.

Creating an Object Coordinate System

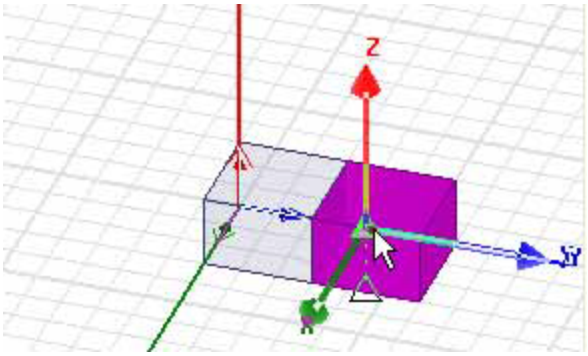
You can create coordinate systems based on any object of solid, sheet, or wire type. The **Modeler** > **Coordinate System** > **Create** > **Object CS** command, and the equivalent ribbon command, is enabled when you select an object. An Object CS can be Offset, Rotated or Both. Executing one of the Object CS commands changes the cursor to the selection marker mode.

As you drag the selection marker over an object, it follows the 3D surfaces of the object, dropping a dashed reference line to a point on the current plane. The cursor changes shape to provide information about the object at the corresponding coordinate:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, in this case, the cursor shows Edge Center triangles as valid selection points for a **Modeler** > **Coordinate Systems** > **Create Object CS** > **Offset** command.



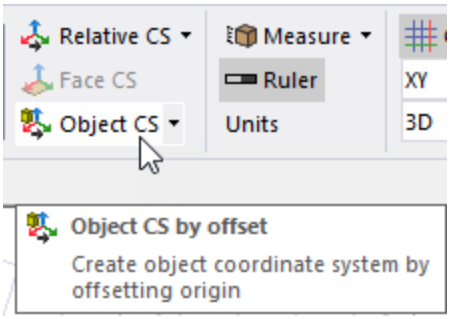
Only operations listed in the History Tree *before* the Object CS's creation will affect the Object CS, and in turn, affect objects dependent upon that Object CS. An Object CS, or objects created on it, is *not* affected by operations that occur after it is created. Also see the Move CS to End command.

For example, suppose you create a box, then an Object CS on a face of the box, and then a cylinder on the Object CS. If you then edit the box's dimensions in the docked *Properties* window, the cylinder will move accordingly. But if you rotate the box using the **Edit** > **Arrange** > **Rotate** command, the box will move, but the cylinder will not move because the operation occurs later in the History Tree.

Creating an Offset Object CS

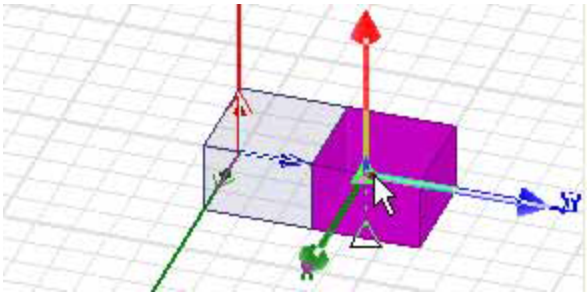
To create an Offset Object coordinate system (CS):

1. Select the working CS to use as the basis of the new Object CS.
2. Select the object to use as the basis of the new Object CS.
3. From the menu bar, click **Modeler** > **Coordinate System** > **Create** > **Object** > **Offset** or, from the **Draw** ribbon tab, click **Object CS**. (You do not have to access the *Object CS* drop-down menu because *Offset CS by offset* is the default type of Object CS):



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click a point on the selected object to locate the Origin for the new CS.
    - You can select any snap point on the object (depending on the current snapping mode options) to select origin of the Object CS. When you hover the mouse over a valid point, a coordinate system preview is shown.

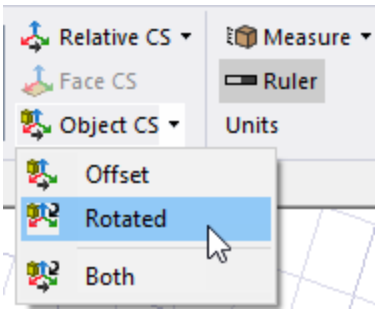


    - The point must be on the selected object.
    - When you select the Origin point, the location is validated, and an appropriate message indicates when a point is invalid. Points where a CS preview appears are always valid.
    - The X-axis {1,0,0}, Y-axis {0,1,0} and Z-axis {0,0,1} are parallel to the corresponding axes of the working CS that was active when you executed the Object CS command, but the Origin has been relocated.
    - The axis coordinates can be later edited through the docked *Properties* window or the *Properties* dialog box.

Creating a Rotated Object CS

To create a Rotated Object coordinate system (CS):

1. Select the working CS to use as the basis of the new Rotated Object CS.
2. Select the object to use as the basis of the new Rotated Object CS.
3. From the menu bar, click **Modeler** > **Coordinate System** > **Create** > **Object** > **Rotated** or, on the **Draw** ribbon tab, click **Object CS** > **Rotated**:



You are prompted to *Select X axis*. A preview of X axis is shown for valid selections.

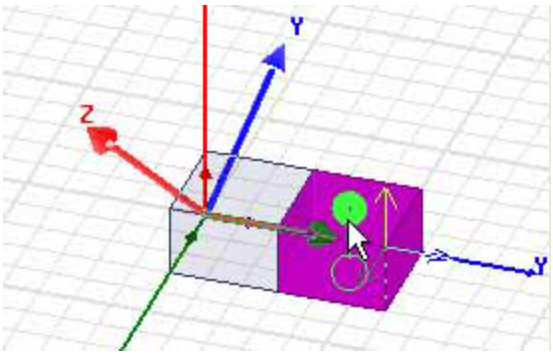
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click a point on the selected object to locate the X-axis direction for the new CS.
   - The Origin {0,0,0} remains at the location defined for the working CS that was active when you executed the Rotated Object CS command. You can edit the coordinates later through the docked *Properties* window or the *Properties* dialog box.
   - The clicked point must be on the selected object.

   You are prompted to *Select to define XY plane*.

5. For 3D models, click a second point on the selected object to define the XY plane.

   A CS preview displays as the cursor hovers over valid selections.



   In the 2D modeler, you are prompted to select only the X axis. The 2D model lies on the XY plane, and the Y-axis is 90 degrees from the X-axis.

Creating an Object CS that is Both Offset and Rotated

To create an Object CS that is both offset and rotated:

1. Set the working CS to use as the basis of the new Object CS.
2. Select the object to use as the basis of the new Object CS.

   The requirements to define this type of Object CS are a combination of *Offset* and *Rotated* Object CS definitions. You are first prompted to select the Origin (refer to the Creating an Offset Object CS page).

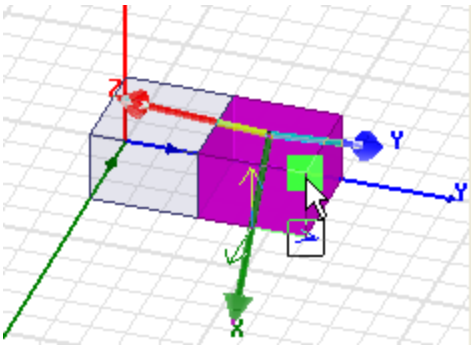
3. Select a point on the object to define the Origin.

   You are then prompted to define the XY plane (refer to the Creating a Rotated Object CS page.

4. For 3D models, select a second point on the object to define the XY plane.

   A CS preview appears when you hover over valid snapping points:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



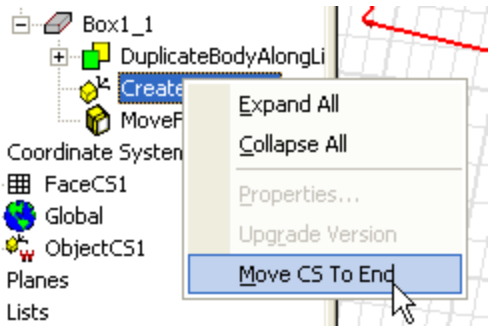
In the 2D modeler, you are asked only to select the origin and the X axis. The 2D model lies on the XY plane, and the Y-axis is 90 degrees from the X-axis.

Move CS to End Command for History Tree

Only operations listed in the History Tree *before* the Object CS's creation will affect the Object CS and, in turn, affect objects dependent upon that Object CS. An Object CS, or objects created on it, is *not* affected by operations that occur after it is created. It is sometimes useful to have the coordinate system affected after any other operations that might have edited, moved, or rotated the object.

If you have at least one operation after a *CreateFaceCS* or *CreateObjectCS* in the History Tree:

Then, selecting a Face or Object CS enables the **Move CS to End** command in the **Modeler** > **Coordinate System** submenu and in the shortcut menu that appears when you right-click the operation in the History Tree:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Executing this command moves the selected CreateObject CS to the end position in the History Tree and updates associated items (other CS, object history, any dependent parts, and so on).

Modifying Coordinate Systems

Keep in mind that when you edit a CS, the following will also be affected:

- All objects drawn on the CS.
- All CSs that were defined relative to that CS.
- All objects drawn on a CS that were defined relative to that CS.

There are two ways to modify a coordinate system. You can select the coordinate system in the History Tree and edit its parameters in the docked *Properties* window. This approach does not allow you to change whether the coordinate system is Absolute or Relative. Nor does it allow you to change how you express the coordinates (as Cartesian, Cylindrical, or Spherical).

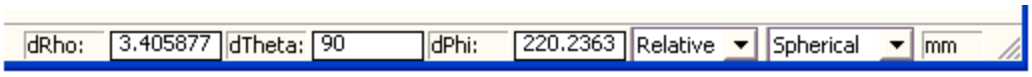
If you want to also modify the whether the coordinate system is Absolute or Relative, and to change how your express the coordinates, do the following:

1. From the menu bar, click **Modeler** > **Coordinate System** > **Edit**.

   The *Select Working CS* dialog box appears.

2. Select the CS you want to modify.
3. Click **Select**.

   This action places you in the graphical selection mode for redefining the CS (origin or X-axis) and enables the editable text boxes in the Status Bar at the bottom right area of the Ansys Electronics Desktop application.



4. You can click points to redefine the CS. Alternatively, you can specify the type of coordinate definition (Absolute or Relative) and tab between the coordinate entry text boxes in the Status Bar to numerically define new coordinates.
   - If you selected a Relative CS, follow the directions for creating a relative CS.
   - If you selected a Face CS, follow the directions for creating a face CS.
   - If you selected an Object CS, follow the directions for creating an object CS.
5. Select the type of coordinate system from rightmost drop-down menu in the Status Bar (Cartesian, Cylindrical, or Spherical).

   The values you specify are translated to Cartesian coordinates in the *Properties* of the revised coordinate system.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

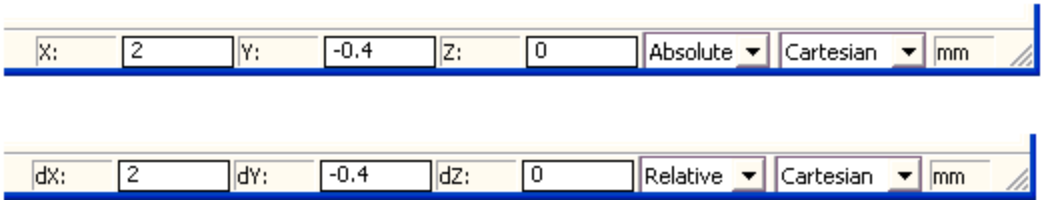
An Object CS can be edited in the same way as it was created. For example, if the CS was created in Offset mode, it will be edited in Offset mode only.

- You can edit the reference CS of an object CS through the docked *Properties* window or the *Properties* dialog box. Doing so will impact object CSs created in *Offset* or *Rotated* mode. It will not impact an object CS created in *Both* (offset and rotated) mode, as the CS fully depends on the object.
- The Origin will be editable when an Object CS was created in *Rotated* mode. Otherwise the Origin coordinates will be read-only text.
- The X Axis and Y Axis will be editable when an Object CS was created in *Offset* mode. Otherwise it will be axis coordinates will be read-only text.
- Either the X Axis or Y axis can be reversed through the docked *Properties* window or the *Properties* dialog box. When the X Axis is reversed, the Y Axis does not change and vice versa.
- When origin or axis properties are defined from snapping points, the point ID is listed in the properties instead of the coordinates. Some examples of snapping point IDs are: *Vertex_10, Face_7 center, Edge_9 midpoint, Edge_8 quadrant,* and *Edge_17 arc center*.

Expressing Cartesian, Cylindrical, or Spherical Coordinates

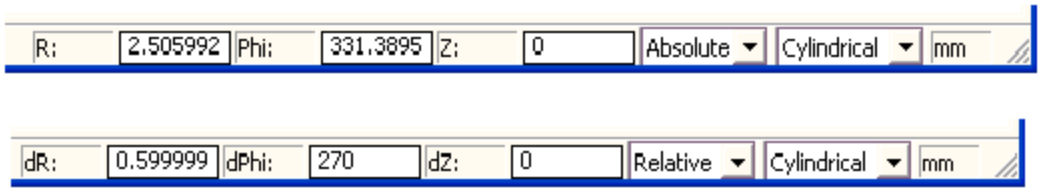
You have choices for expressing coordinates, specifically as Cartesian, Cylindrical, or Spherical by using the editable fields in the Status Bar. In each case, you can also specify whether to enter the coordinates as Absolute or Relative to a reference point. Click the first text box, or press **Tab** to begin entering values or variables. **Tab** to the next text box or **Ctrl+Tab** to the previous one. Press **Enter** when you are done. After you enter values or variables in the text boxes, they are evaluated as Cartesian for the coordinate system *Properties*, regardless of the way they were defined.

- **Cartesian:** Coordinates represent the point's distance from the origin in the x, y, and z directions. The values are specified in the **X, Y,** and **Z** text boxes.

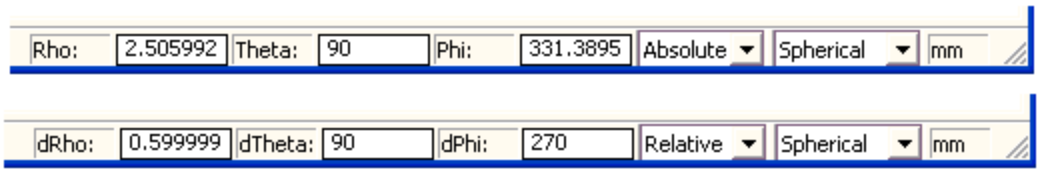


- **Cylindrical:** Specify the point's radius, measured from the origin, is the **R** text box, the angle from the x-axis in the **Theta** text box, and the z-distance from the origin in the **Z** text box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Spherical:** Specify the point's radius, measured from the origin, in the **Rho** text box, the angle from the x-axis in the **Theta** text box, and the angle from the origin in the z direction in the **Phi** text box.



Deleting Coordinate Systems

1. In the *Coordinate System* branch of the History Tree, click the name of the CS you want to delete.
2. Delete the CS using one of the following methods:
   - Press **Delete**.
   - From the menu bar, click **Edit** > ✕ **Delete**.
   - On the **Desktop** ribbon tab, click ✕ **Delete**.

   The CS will be deleted and, all objects drawn on it will be deleted. Furthermore, any CS that was dependent upon the deleted CS will be deleted, and any objects that were drawn on the dependent CS will also be deleted.

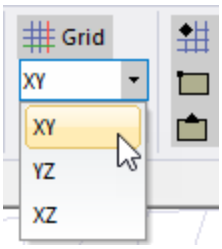
Setting the Drawing Plane

The *Drawing plane* options are available from a drop-down menu on the *Draw* ribbon tab and from the *Grid Plane* submenu of the *Modeler* menu. The XY, YZ, and XZ drawing planes correspond to the active coordinate system, not the global planes (unless *Global* is the currently active CS).

To set the drawing plane, do one of the following:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- On the **Draw** ribbon tab, select **XY, YZ,** or **XZ** from the **Drawing plane** drop-down menu:



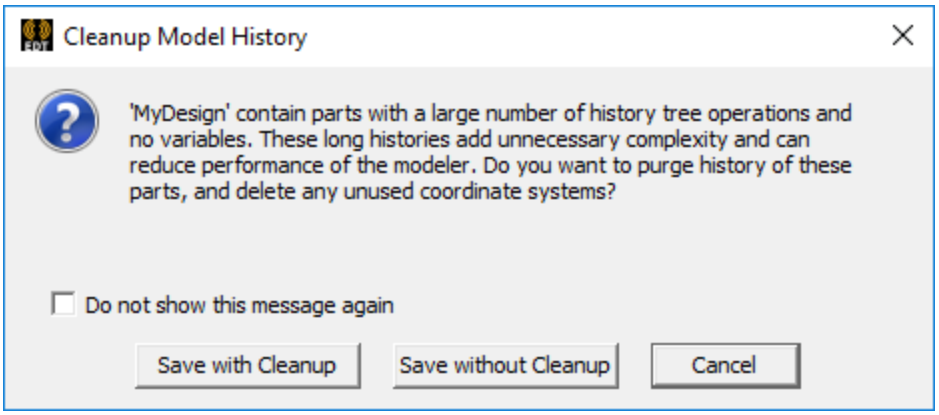
- From the menu bar, click **Modeler** > **Grid Plane** > and select one of the planes **XY, YZ,** or **XZ**.

Cleaning Up Model History

Ansys Electronics Desktop can clean up history tree operations in order to improve performance.

When saving your project, you will be prompted to clean up your model's history if some parts meet two conditions:

- No variables assigned to the part
- A history greater than a specified number of actions



The number of actions that triggers this prompt can be set in 3D Modeler Operations Options.

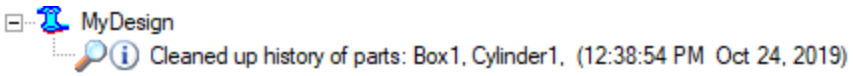
When promoted, you can choose **Do not show this message again** to hide the prompt for the active project.

To clean up the model history manually:

- Select **Modeler** > **Cleanup Model History**.

After the model has been cleaned up, you can view cleaned up parts in the **Messages** window:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

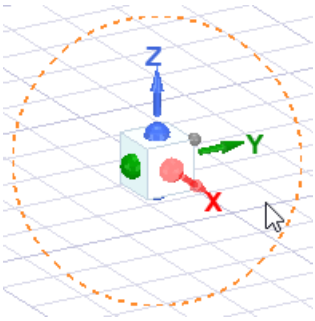
8 - Modifying the 3D Model View

You can modify the view of objects in the 3D **Modeler** window without changing their actual rotation or position within the coordinate system. You vary the viewing direction and the relative location of the "camera" (that is, the viewpoint) relative to the model geometry. The tools allow you to zoom, pan, or rotate the view orientation and to quickly switch to predefined or user-defined view orientations. Additionally, you can choose different object visualization options, show/hide objects, and customize the window background.

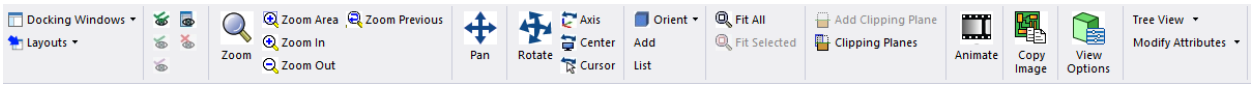
What do you want to do?

Change the View	Show or Hide Design Objects
Change Object Visualization	Change the Background

In addition to various menu bar, ribbon, and shortcut menu commands, you can manipulate the view orientation of 3D models and 3D plots using the Orientation Gadget:



Finally, you can also access many of the Modify View features via the **View Ribbon**. The following image has hot links for each area.



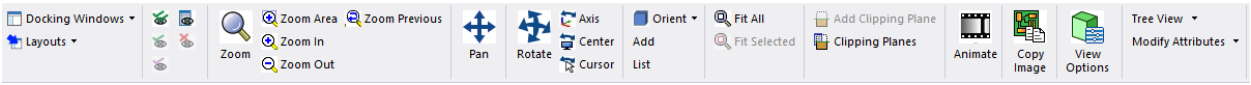
Change the View

- Using the Orientation Gadget
- Changing the Model View with Alt+Double-Click Areas in the Modeler Window
- Apply an Orientation to the Current View

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Set the View Options
- Rotate the View
- Pan the View
- Zoom In or Out
- Zoom to Selected Excitation
- Zoom In or Out Using a Mousewheel
- Fit Contents in the View Window
- Use Clip Planes

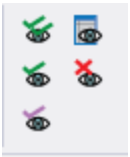
You can also access many of the Modify View features via the **View Ribbon**. The following image has hot links for each area.



Show or Hide Design Objects

- Show or Hide Objects
- Showing Only Selected Objects in All or Active Views

You can also access the Show or Hide features by using the following icons on the **View, Draw,** or **Model** ribbon tabs:



Change Object Visualization

- Render Objects as Wireframes, Flat-Shaded, or Smooth-Shaded Solids
- Set the Enhanced Display of Material Color and Transparency
- Set the Surface Visualization
- Modify the View Orientation
- Modify the Lighting
- Set the Projection View

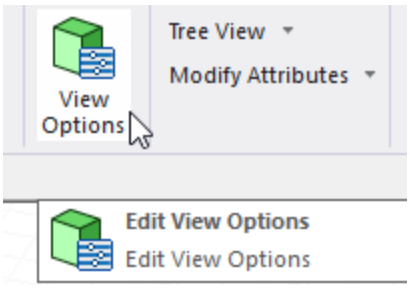
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Change the Background

- Setting the Background Color
- Modify the appearance of the Coordinate System Axes
- Modify the appearance of the Grid

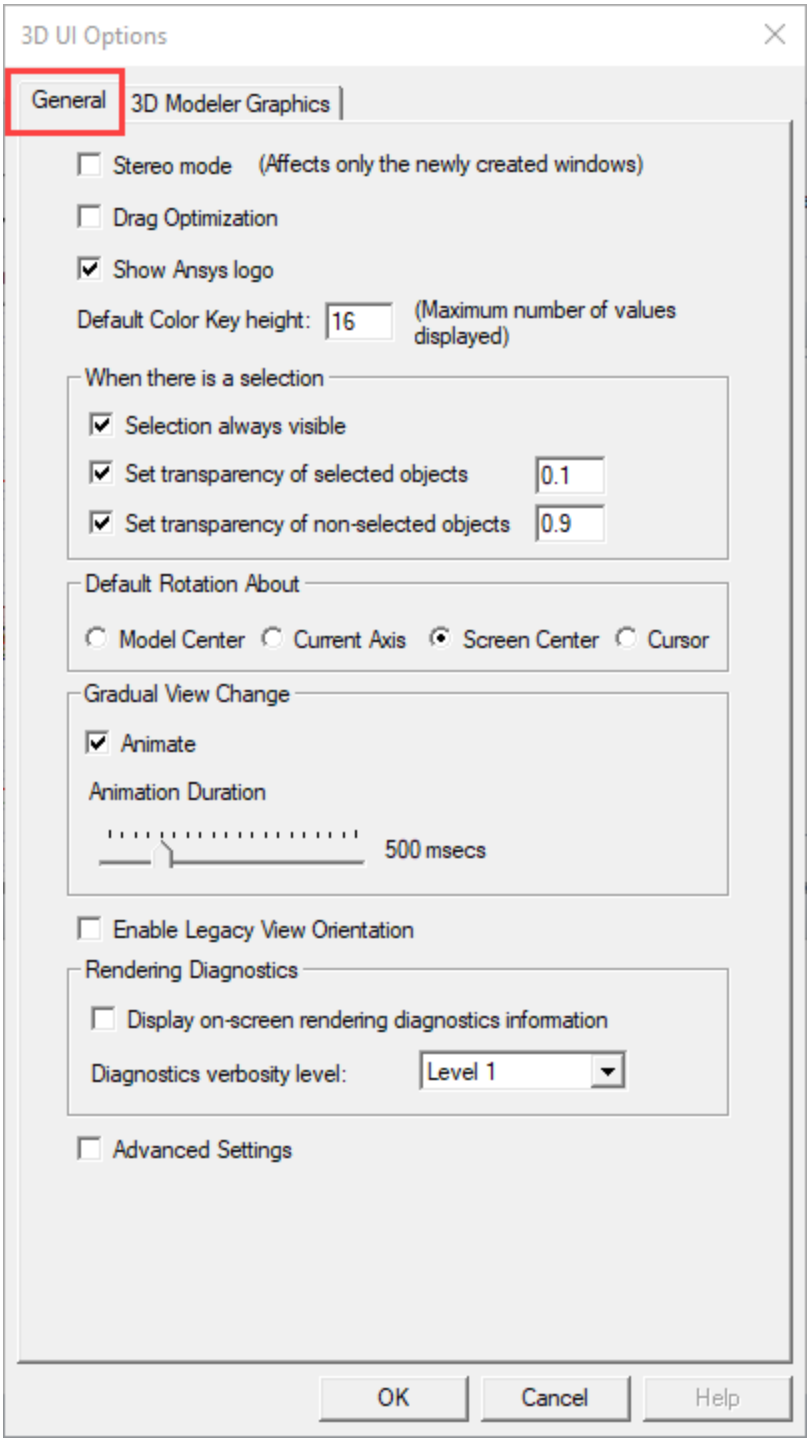
View Options: 3D UI Options

To access View Options, use the **View** > **Options** command, or click **View Options** on the **View** ribbon.



The **3D UI Options** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **General** tab lets you set defaults for the following view options:

- Stereo Mode (default, disabled) – This option is only applicable to systems with support for stereographic viewing for greater 3D realism.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Drag Optimization (default, disabled)
- Show Ansys logo (default, enabled)
- Default Color Key height (default = 16)

When there is a selection option:

- Selection always visible (default, enabled)
- Set transparency of selected objects
- Set transparency of non-selected objects.

Default Rotation About options:

- Model Center
- Current Axis
- Screen Center (default)
- Cursor

  When you select one of these as the default, the **View** > **Rotate** selection menu changes to show that the **Alt+drag** combination attaches to your selection.

Gradual View Change settings:

- Animate
- Animation Duration

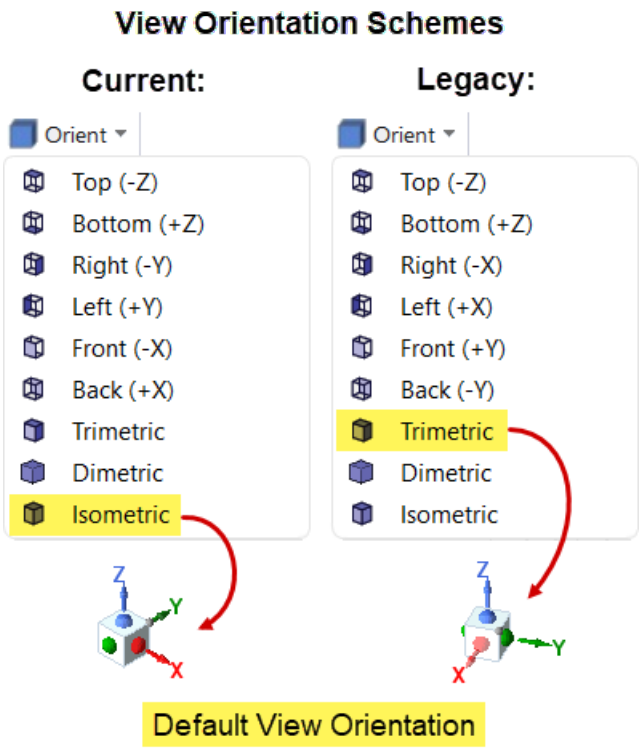
  For operations such as Fit Selection, Fit All, and Zoom, these settings determine whether Electronics Desktop animates the transition between images and how smoothly it does so.

Enable Legacy View Orientations option:

- When selected, the view orientations scheme and the axis directions for named views follow the legacy conventions (software version 2023 R2 or older):
- When cleared, the view orientations scheme and the axis directions for named views follow the current view orientation scheme

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following view is a comparison of the two options:



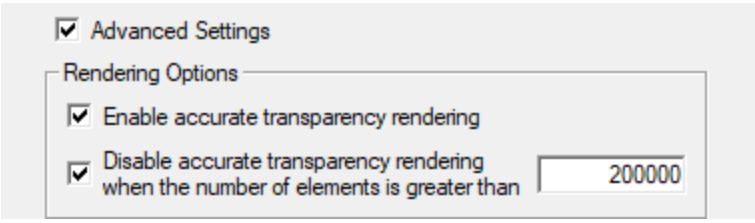
Rendering Diagnostics Options: .

You can select to display on-screen rendering information, and the diagnostics verbosity level from level 1 to level 4.

Advanced Settings: Rendering Options:

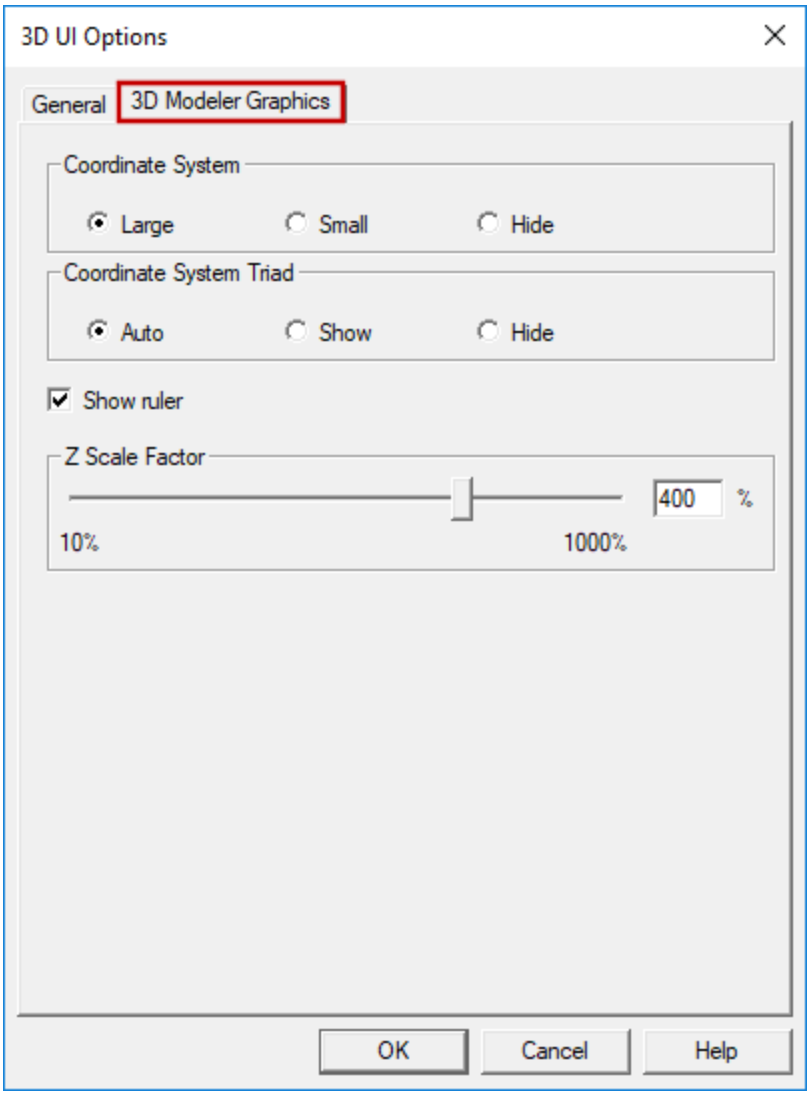
Because rendering transparency correctly can be compute intensive, especially for complex models with thousands of objects, it can bring down the frame rate. In such cases, you can select whether to **Enable accurate transparency rendering**.

You can also choose to **Disable accurate transparency rendering** when the number of elements is greater than a specified number. The default is 200000.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3D Modeler Graphics Tab



The **3D Modeler Graphics** tab lets you set the following options which are invoked for the next Project and design created.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

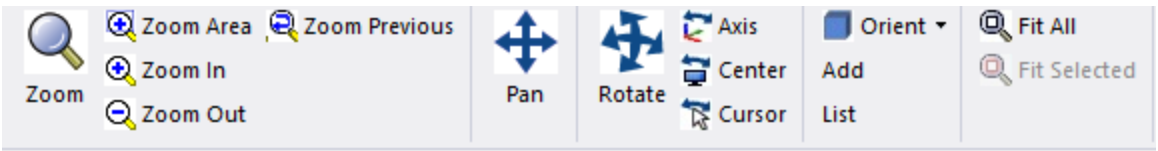
Settings	Description of Choices
Coordinate System	• **Large** (extends to the window edges) • **Small** (centered, not extending toward edges with smaller arrows) • **Hide** (do not display)
Coordinate System Triad (A secondary, smaller coordinate system appearing near the lower right corner of the Modeler window)	• **Auto** • **Show** • **Hide**
Show Ruler (To indicate the scale of the model, based on current units. The ruler appears at the lower center of the Modeler window)	Select to enable; clear to disable
Note: You can also toggle the ruler visibility by clicking **Ruler** on the **Draw** ribbon tab or by right-clicking in the *Modeler* window and choosing **View** > **Ruler** from the shortcut menu.	
Z Scale Factor	The default is 400%. You can use the slider or the text field to specify a different value.

Panning the View

To move (pan) the view:

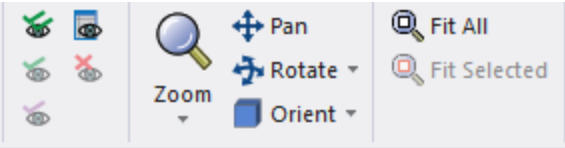
1. Click **View** > **Interaction** > **Pan** or click the **Pan** icon on the **View, Draw,** or **Model** ribbon tab.

   View ribbon tab:



   Draw or **Model** ribbon tabs:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click and drag the mouse in the direction you want to pan the view.

   The view will pan until you release the left mouse button.

3. To exit **Pan** mode, click **Pan** on the **View** menu again or press **Esc**.

> **Tip:**
>
> Alternatively, pan the view using one of the following methods:
>
> - Hold down **Ctrl** as you click and drag using the **middle mouse button**.
> - If you chose to select *Enable Legacy View Navigation* in the General > User Interface options, you can also use the legacy combination **Shift** + **drag** (with the left mouse button) to pan the view.
> - Press **Ctrl** + ***arrow keys*** to pan vertically or horizontally. The movement is 10 screen pixels per press of the arrow key.
> - In addition to the menu bar and ribbon tabs, you can access *Pan* from a shortcut menu. Right-click in the Modeler window and click **View** > **Pan**. Click the command again or press **Esc** to terminate panning.

Zooming in and Out

You can magnify (zoom in) or shrink (zoom out) the contents in the view window using the following methods:

- Hot Keys (that work without the mouse)
- Hot Keys (used in conjunction with clicking and dragging the mouse)
- **Zoom In** and **Zoom Out** commands, which produce 5% zoom increments
- **Zoom Previous** to undo recent zoom commands, as described below
- Zoom into a Rectangular Area
- Zoom in or out Using a Mouse Wheel
- Zoom to a Selected Boundary or Excitation

> **Note:** Some international keyboards may not support the following hotkeys. If the Shift key is required for a character, such as the plus sign (+), the hotkey may not function.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To zoom in using hotkeys:

- Press **Ctrl** + "**+**"

  If using the "=\+" key (not the "+" key on the numeric keypad) do **not** press the *Shift* key.

  The view zooms in 5 percent.

To zoom out using hotkeys:

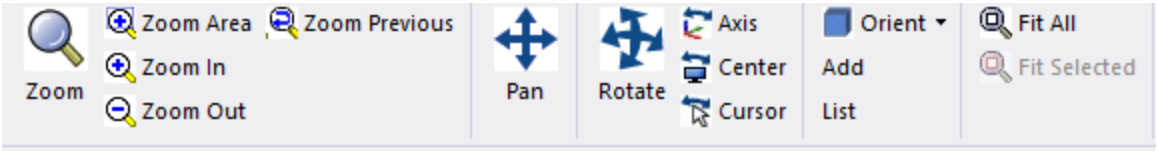
- Press **Ctrl** + "**-**"

  The view zooms out 5 percent.

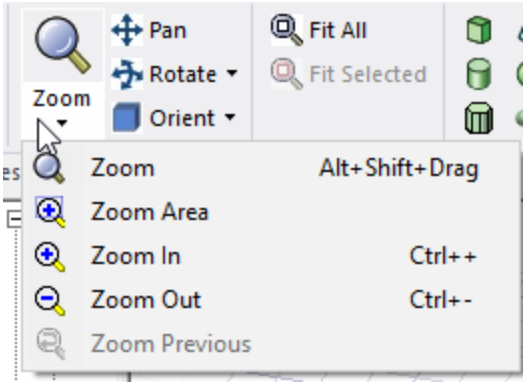
To zoom by dragging the mouse.

1. Click **View** > **Interaction** > **Zoom**, click **View** > **Zoom** on the shortcut menu, or click the **Zoom** icon on the **View, Draw,** or **Model** ribbon tabs.

   View ribbon tab with **Zoom** commands.



   Draw or **Model** ribbon tabs with **Zoom** icon, and drop-down menu for Zoom commands:



2. To zoom in, drag the mouse toward the top of the view window. The objects in view expand as you drag.

   To zoom out, drag the mouse toward the bottom of the view window. The objects in view decrease in size as you drag.

   When zooming on a view of model objects the absolute size of the model does not change.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When zooming on a 2D report, axis labels and ticks will adjust automatically during the zoom operation and will rescale to their final value after the zoom operation is complete.

3. To end **Zoom** mode, click **View** > **Interaction** > **Zoom** again, or click the Zoom icon, or press **Esc**.

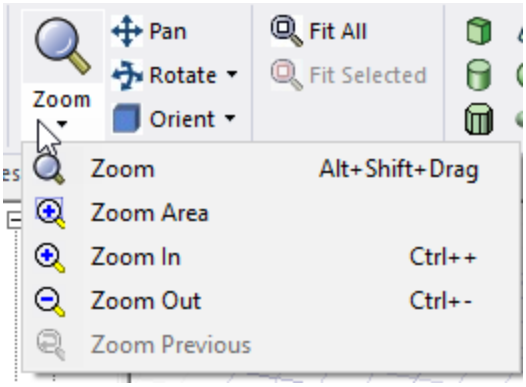
Zoom In and Zoom Out Commands

To use **Zoom In**, **Zoom Out** to zoom in or out in 5% increments:

1. Click **View** > **Interaction** > **Zoom**, click **View** > **Zoom In** or **Zoom Out** on the shortcut menu, or click the **Zoom** icon on the **View, Draw,** or **Model** ribbon tabs.

   View ribbon tab with **Zoom In** and **Zoom Out**.

   Draw or **Model** ribbon tabs with **Zoom** icon, and drop-down menu for **Zoom In** and **Zoom Out**:



2. Click **Zoom In** or **Zoom Out** for a 5% increment zoom in or out respectively.

   To zoom out, drag the mouse toward the bottom of the view window. The objects in view decrease in size as you drag.

   When zooming on a view of model objects the absolute size of the model does not change.

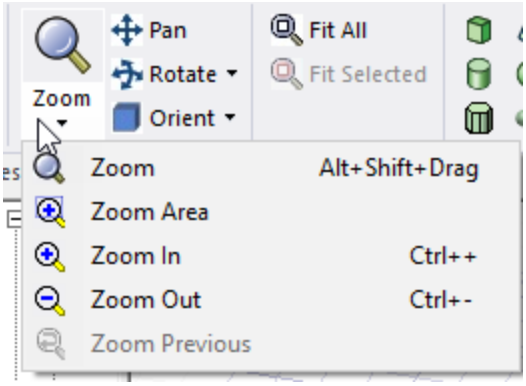
   When zooming on a 2D report, axis labels and ticks will adjust automatically during the zoom operation and will rescale to their final value after the zoom operation is complete.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. To end **Zoom** mode, click **View** > **Interaction** > **Zoom** again, or click the **Zoom** icon, or press **Esc**.

Zoom Previous Command

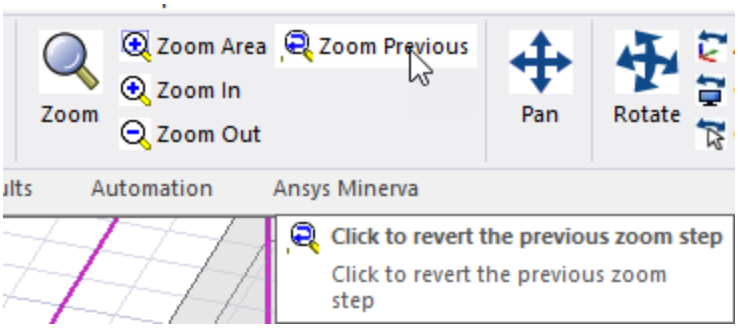
The **Zoom Previous** command is not enabled until at least one **Zoom** command has been used.



It operates on any existing sequence of **Zoom** commands. That is, it undoes the last previous **Zoom** command, and then, another exists, the one before that. To use **Zoom Previous**.

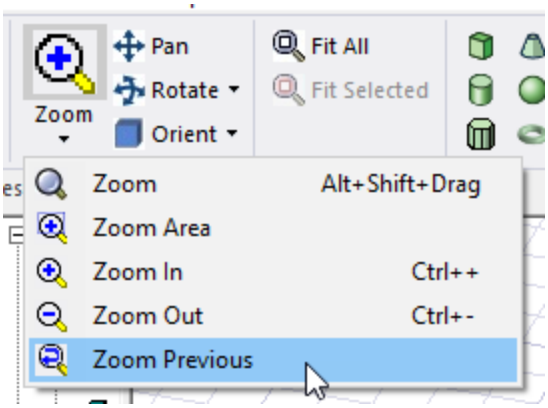
1. Click **View** > **Interaction** > **Zoom Previous**, click **View** > **Zoom Previous** or Zoom Out on the shortcut menu, or click the **Zoom** icon on the **View, Draw,** or **Model** ribbon tabs.

   View ribbon tab with .



   Draw or **Model** ribbon tabs with **Zoom** icon, and drop-down menu for **Zoom Previous**:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. **Zoom Previous** undoes the last executed Zoom command, and then, if others exist, can be used to undo them in reversed order of execution.

   When zooming on a view of model objects the absolute size of the model does not change.

   When zooming on a 2D report, axis labels and ticks will adjust automatically during the zoom operation and will rescale to their final value after the zoom operation is complete.

3. To end **Zoom** mode, click **View** > **Interaction** > **Zoom** again, or click the **Zoom** icon, or press **Esc**.

Tip:

Alternatively, zoom in or out using one of the following methods:

- Hold down **Shift** as you click and drag using the **middle mouse button**. Moving the cursor upward zooms in and moving it downward zooms out.
- If you chose to select *Enable Legacy View Navigation* in the General > User Interface options, you can also use the legacy mouse-button and hotkey for zooming. Hold down **Alt** + **Shift** as you click and drag (using the left mouse button).
- Roll the mouse wheel. Rolling the top of the wheel forward (away from you) moves the model view further away from you (zooming out). Conversely, rolling the top of the wheel rearward (towards you) zooms in.
- Right-click in the Modeler window and then click **View** > **Zoom** on the shortcut menu. Click and drag using the left mouse button to zoom in or out. Repeat the command or press **Esc** to cancel zooming.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

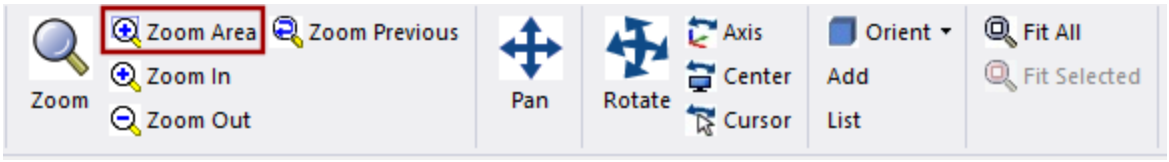
If zooming is slow, especially for complex models, for some graphics cards, you can improve performance by setting **NVIDIA Control Panel** > **3D Settings** > **Manage 3D Settings** > **Global Settings** > **Global Presets: Workstation App - Dynamic Streaming**

For more information about graphics cards, navigate to the appropriate Windows or Linux Ansys Installation Guide and search for **OpenGL**.

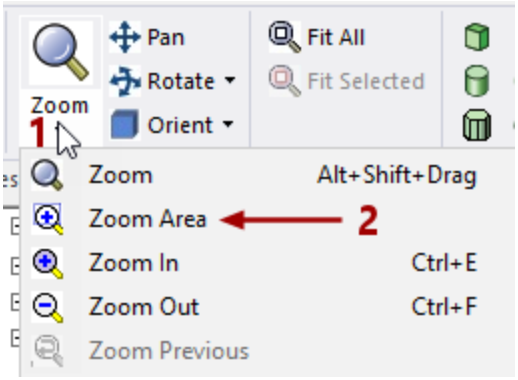
Zooming into a Rectangular Area

Zoom to expand a specified rectangular area to fill the view window, as follows:

1. Use one of the following methods to begin the *Zoom Area* mode:
   - Using the menu bar, click **View** > **Interaction** > **Zoom Area**.
   - From the **View** ribbon tab, click **Zoom Area**:



   - From the **Draw** or **Model** ribbon tab, click **Zoom** > **Zoom Area**:



   - Right-click in the Modeler window and choose **View** > **Zoom Area** from the shortcut menu.
2. Click and drag the mouse to draw a rectangle enclosing the area that you want to zoom into.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you release the mouse button, the rectangular area will come close to filling the display area of the window.

The absolute size of the zoomed geometry does not change, only the scaling of the model view is affected.

3. To end the **Zoom Area** mode, click the command again or press **Esc**.

   The rotate and zoom drop-down menu icons on the ribbon update dynamically to indicate the current rotate/zoom mode. These ribbon buttons are also "sticky." That is, the last active rotate or zoom mode becomes the button default until a different command is chosen or the application is restarted. Default commands are restored the next time you launch the Ansys Electronics Desktop program.

   For 2D or 3D reports, axis labels and tick marks are adjusted after the zoom operation is complete.

Zooming in or out Using a Mouse Wheel

You can zoom the active model view in and out by rotating the mouse wheel. This action works in the Modeler window, for 2D and 3D plots, and more. The cursor location is the center point whenever using the mouse wheel for zooming, and this point remains fixed. The grid and ruler, when visible, are adjusted dynamically as you zoom in and out.

Rotating the wheel forward, away from you, moves the model away from you (zooming out). Rotating the wheel backward, towards you, pulls the model closer (zooming in).

> **Note:**
>
> If Zooming is slow, especially for complex models, for some graphics cards, you can improve performance by setting **NVIDIA Control Panel** > **3D Settings** > **Manage 3D Settings Global Settings** > **Global Presets: Workstation App - Dynamic Streaming**
>
> For more information about graphics cards, navigate to the appropriate Windows or Linux Ansys Installation Guide and search for **OpenGL**.

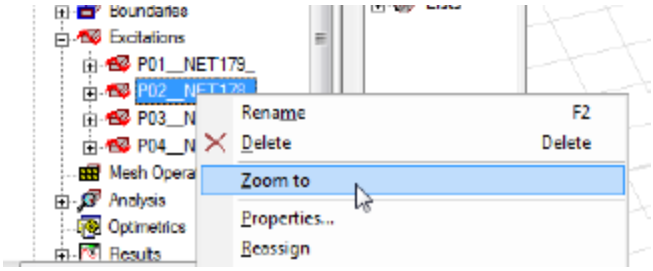
Zoom to a Selected Boundary or Excitation

Zoom to is an option that can enlarge the view of a boundary or excitation. This command can be very useful for locating small items with boundaries or excitations assigned or to inspect any problem areas.

You can use the **Zoom to** command as follows.

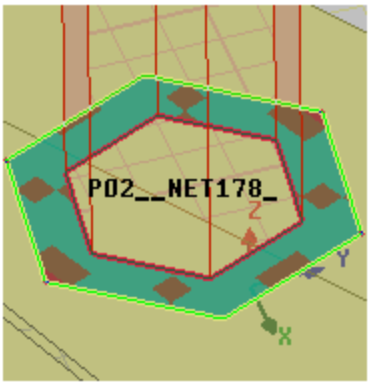
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. If needed, expand the *Boundary* or *Excitation* branch in the Project Manager to see the previously assigned conditions.
2. Right-click the boundary or excitation of interest to highlight it in the model view and to access the shortcut menu, as shown below:



3. Select **Zoom to**.

   Notice that the command instantly increases magnification, filling the screen with the object(s) or face(s) to which the boundary or excitation is assigned, as shown in the following figure:

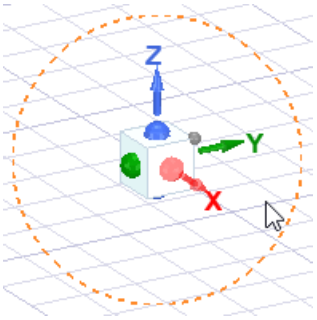


Modifying the View Orientation

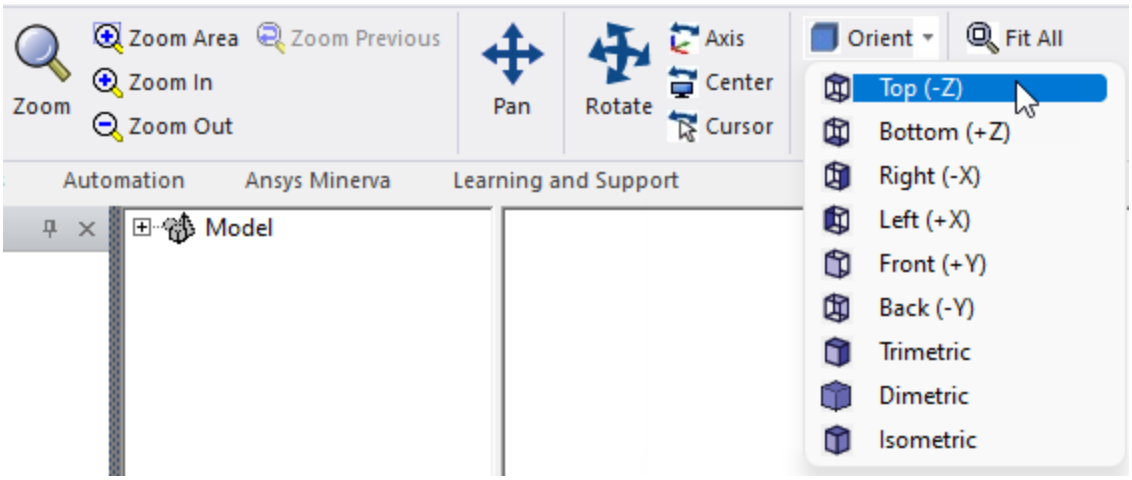
There are several places you can access commands to change the view orientation in a *Modeler* or *3D Plot* window:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Use the View Orientation Gadget:



- Select the **Orient** drop-down Menu on the **View** ribbon tab and choose one of the predefined view orientations:



The **Orient** drop-down menu can also be located on the **Draw** and **Model** ribbon tabs.

The sign and letter in parentheses that follows each standard view indicates the viewing direction, not the model side. For example, consider a simple box aligned with the global planes. The right side of the box is the +X side, but the viewing direction is -X (you look leftward to see the right side of an object).

- Right-click in the Modeler window and, from the shortcut menu that appears, point to **View** > **Apply Orientation**, and select the desired viewing direction from the submenu. The choices include those shown in the preceding image plus any custom views you have defined (see the next bullet).
- From the menu bar, click **View** > **Modify Attributes** > **Orientation List**.

  A dialog box with orientation settings appears. It includes a table with the names of nine basic orientations (Top, Bottom, Right, Left, Front, Back, Trimetric, Dimetric, and Isometric), any additional orientations that you have added, and a section for managing orientations (adding/removing orientations or setting the default view).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The table includes columns that show the viewing angles relative to the global axes and the equivalent vector components. These vector components are listed based on the global coordinate system and any applicable local coordinate systems.

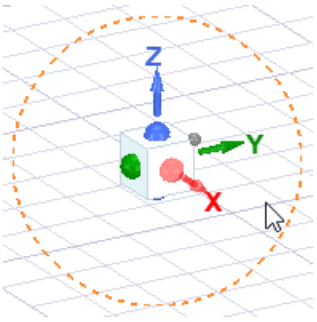
1. Select a default orientation or create a new orientation.
2. Click **Apply to View** for the selected view to appear in the window.
3. Optionally, click the **Reset View Orientation** button to restore the selected view to the original angles.
4. Click **Make Default** if you want the selected viewing direction to be the initial viewing direction when a **3D Modeler** window is opened, either in the current project or future projects.
5. Click **Close** to dismiss the dialog box.

   The orientation you set will be saved with the design. New orientations assigned to other designs after this point will not affect this orientation.

Apply an Orientation to the Current View

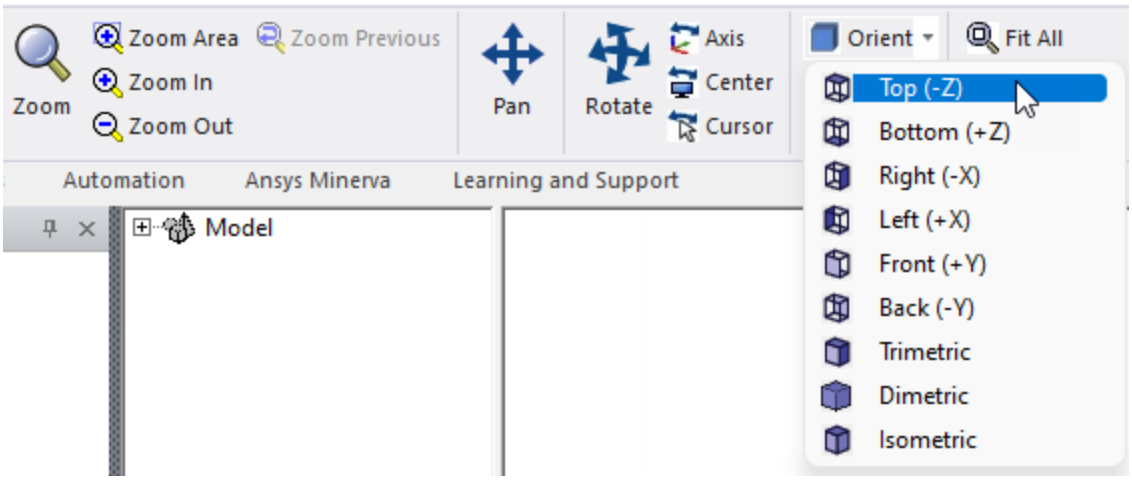
Apply view orientation settings using any one of several available methods:

- Use the Orientation Gadget:

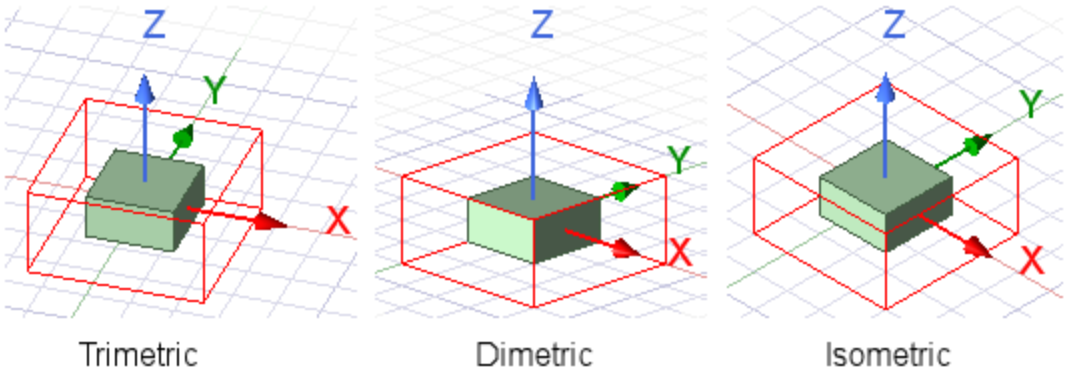


- Use the commands in the **Orient** drop-down menu located on the **View, Draw,** or **Model** ribbon tabs:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

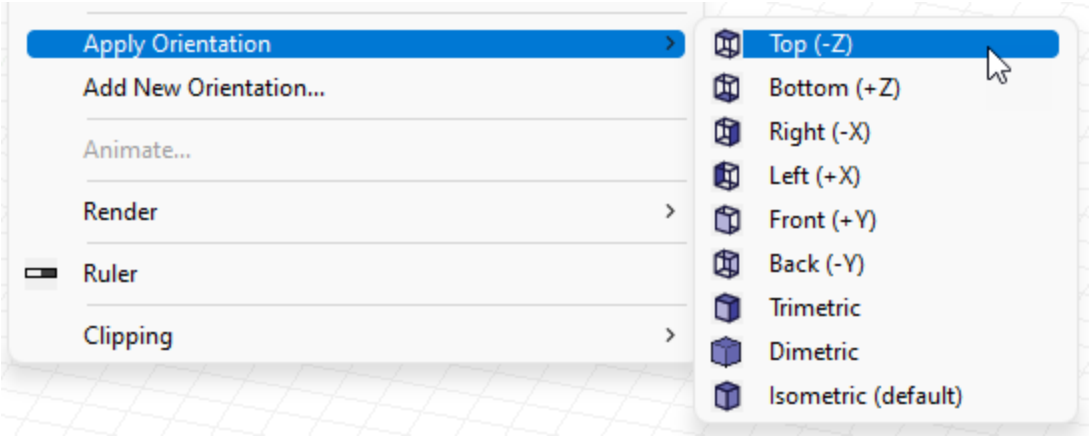


Possible orientations are Top, Bottom, Right, Left, Front, Back, Trimetric, Dimetric, and Isometric. These last three differ as shown below:



- Right-click in the Modeler window. In the shortcut menu that appears, go to the **View** > **Apply Orientation** submenu:

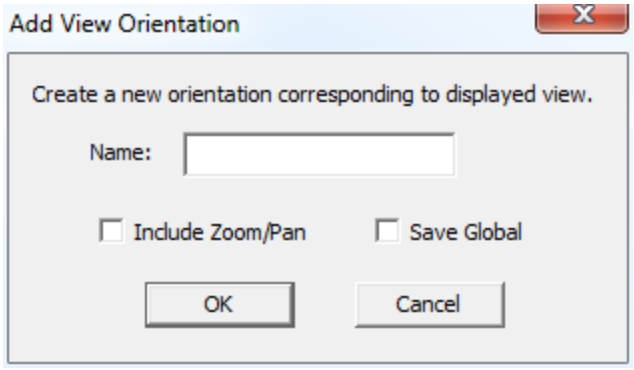
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In addition to the standard views, any custom views that you have defined will appear in this submenu.

After applying an orientation, you can click **View** > **Undo View : *orientation***.

You can also right-click in the Modeler window and choose **View** > **Add New Orientation** to create a custom viewpoint.

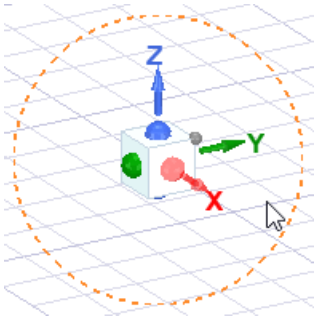


The **Name** that you assign will appear in the **View** > **Apply Orientation** submenu within the Modeler window's shortcut menu. You can specify whether to **Include Zoom/Pan** (in addition to the viewpoint orientation) and whether to **Save Global**, which makes the custom orientation available for all projects.

Using the Orientation Gadget

The **Orientation Gadget** is a tool for quickly manipulating the view orientation of 3D models and 3D plots graphically. It is a convenient alternative to using the **Orient** drop-down menu on the **View** ribbon tab. The Orientation Gadget is enabled by default and appears near the lower left corner of the display area in *Modeler* and *3D Plot* windows:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The orange dashed circle appears only when the cursor is near the gadget, and its purpose is discussed later in this topic.

There is a colored dot at the center of each face of the cube and red (X), green (Y), and blue (Z) axes, indicating the axis positive directions for the currently active coordinate system. Additionally, there is a smaller gray dot at the corner where the +X, +Y, and +Z faces meet.

The following list summarizes the results of clicking different parts of the gadget (and optionally dragging the mouse while clicking):

- **Click the large dot at the center of any of the six cube faces:**

  This action rotates the view orientation to place the side of the model that the clicked face represents towards you, parallel with the screen. For example, if you click the dot on the -Y face, the -Y side of the model faces you, which is the *Left* view (+Y viewing direction).

  > **Note:**
  >
  > If the current viewing direction corresponds to one of the global or active user-defined coordinate system axes (±X/Ux, ±Y/Uy, or ±Z/Uz), clicking the axis label or dot at the center of the Orientation Gadget (on the face that is towards you) reverses the view orientation. So, with a single click of the mouse, you can switch from front-to-back, right-to-left, and top-to-bottom (or the inverses of these view orientation changes).

- **Click on one of the gadget axes:**

  This action is equivalent to clicking the dot at the center of the same positive cube face. The axis orientation becomes perpendicular to the screen and towards you. Therefore, the resulting three views are looking at the +X (front), +Y (right), or +Z (top) side of the model (-X, -Y, or -Z viewing direction, respectively).

- **Click the small gray dot at the cube's +X, +Y, +Z corner:**

  This action produces the standard *Isometric* view orientation.

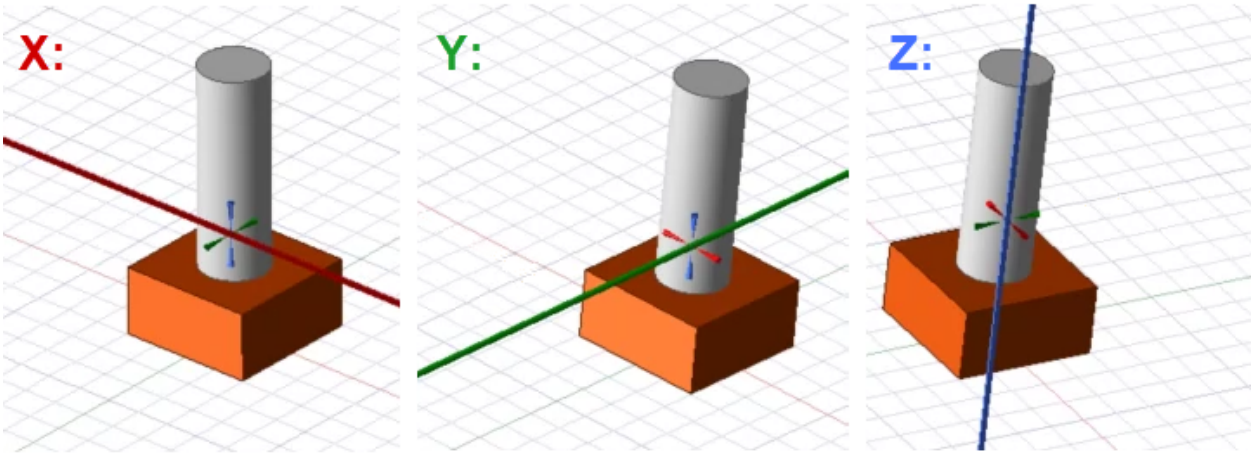
- **Click on or very close to the cube (but not on an axis or dot) and drag the mouse:**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This action freely rotates the view orientation. The rotation center depends on the **Default Rotation About** option selected in the *3D UI Options* dialog box. The behavior is equivalent to clicking and dragging in the display area using the middle button / wheel but is performed with the left mouse button. Additionally, you can constrain rotation to be about any of the three active coordinate system axes by pressing one of the following keys while clicking and dragging the mouse:

- **X + click and drag** to rotate the view orientation about the active **X** axis.
- **Y + click and drag** to rotate the view orientation about the active **Y** axis.
- **Z + click and drag** to rotate the view orientation about the active **Z** axis.

During constrained rotation, a heavy red (X), green (Y), or blue (Z) axis line appears in the Modeler window to provide a clear visual indication of the axis of rotation. These axes pass through the center of rotation, and this center is marked by crosshairs. When dragging the view orientation gadget to rotate the view, the center of rotation is the centerpoint of the drawing canvas. The following image shows the appearance of the rotation axes and center markers:



Additionally, a pair of gold, circular arrows appear on the view orientation gadget indicating the axis of constrained rotation:

While the same constrained rotation functionality for 3D models is also available for 3D reports, the red, green, and blue axes do not appear in 3D report windows during axis-constrained rotation. However, the pair of circular, gold arrows appear on the view

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

orientation gadget in both the Modeler and 3D Report windows to indicate the axis of constrained rotation.

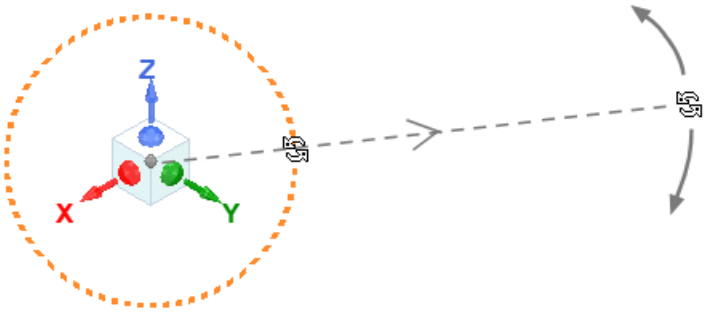
For more information and tips about the usage of this feature, see Rotating or Spinning the View.

- **Orange dashed circle:**

  This circle is used to rotate the current view about an axis normal to the screen. That is, the view rotation is constrained to the plane of the screen regardless of the current viewing direction. The line weight of the dashed orange circle becomes heavier when the cursor is touching it, indicating a correct clicking point. There are three actions, as follows:

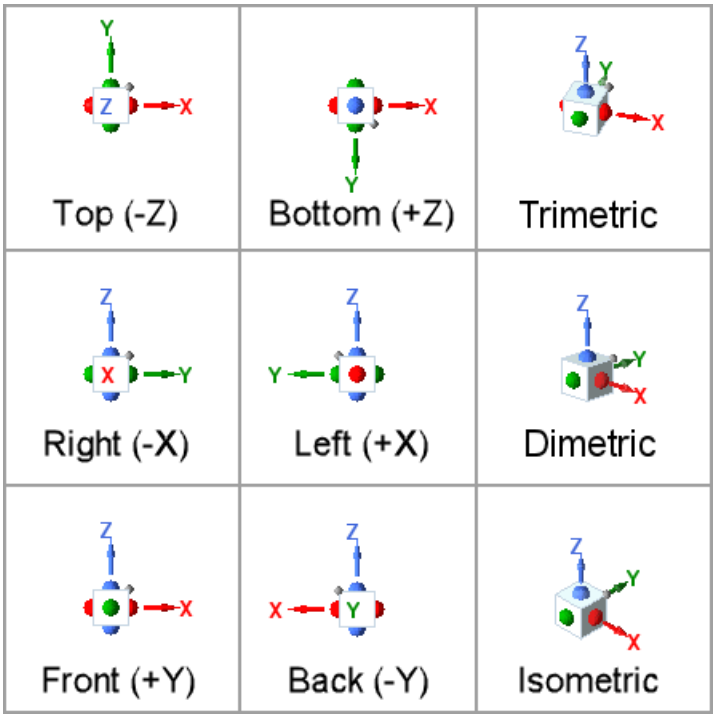
  - **Click anywhere on the circle:** The model rotates **90 degrees clockwise**.
  - **Ctrl+click anywhere on the circle:** The model rotates **90 degrees counterclockwise**. This action and the preceding one are particularly useful for changing the in-plane positive axis directions for standard top, bottom, left, right, front, or back views.
  - **Click and drag:** Freely rotates the model in either direction. Release the mouse button when the desired orientation is reached. While the mouse button remains pressed, the cursor changes to a rotation icon:

    You do not have to keep the cursor on the circle while dragging. You can often achieve more precise rotations with the cursor further away from the orientation gadget. Moving the cursor in a radial direction away from the center of the orientation gadget produces minimal rotation. Then, move it tangentially for precise rotation. The greater the distance from the gadget, the finer the rotational control.



The appearance of the Orientation Gadget for each of the standard view orientation is as shown in the following image:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



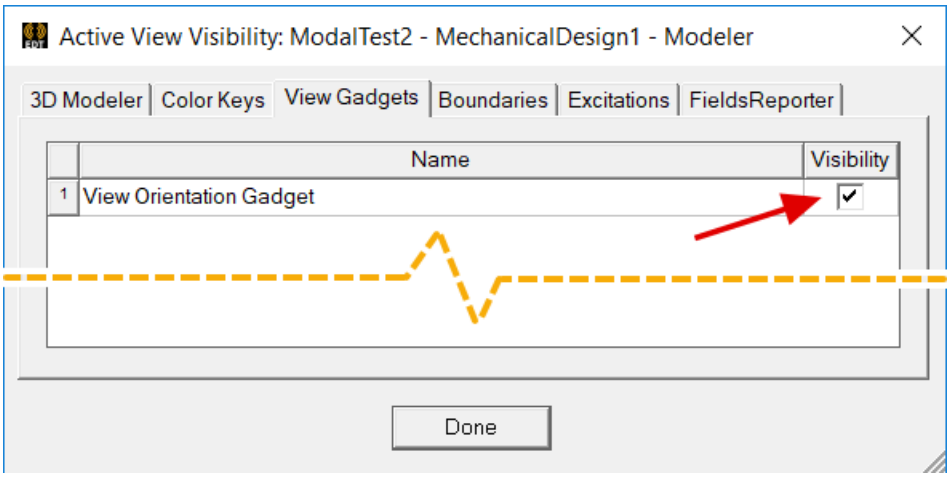
> **Note:**
>
> The orientation, colored dots, and axes of the Orientation Gadget correspond to the active coordinate system, whether **Global** or **user-defined**. All standard views are defined in the same manner as they are for the global CS, but Ux, Uy, and Uz directions are used instead of global X, Y, and Z.

Controlling Visibility of the Orientation Gadget

You can enable or disable the Orientation Gadget visibility as follows:

1. Access the *Active View Visibility* dialog box using one of the following two methods:
   - From the menu bar, click **View > Visibility > Active View Visibility**.
   - On the **View, Draw,** or **Model** ribbon tab, click **Hide/Show overlaid visualization in the active view**.
2. Select the **View Gadgets** tab.
3. Select or clear the checkbox in the **Visibility** column to the right of **View Orientation Gadget** to control the gadget visibility.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Click **Done** to close the dialog box.

Rotating or Spinning the View

You can rotate the view relative to the Model Center, the Screen Center, the Current Axis (that is, the active coordinate system origin), or the Cursor location. You can activate the rotation modes via the menu bar, ribbon tabs, or shortcut menu. When a *Rotation* command is active, you can click and drag using the **left mouse button** to manipulate the model viewpoint.

Rotation Using Only the Mouse: You can use the **middle mouse button** to rotate the view *without* selecting a *Rotation* command. From the menu bar, click **View** > **Options** to access the *General* tab of the *3D UI Options* dialog box. Here, you can choose the desired **Default Rotation About** option. The view rotates about the **Screen Center** by default. The other available options are rotation about the **Model Center**, **Current Axis** (active coordinate system origin), or **Cursor** location. This setting also controls the rotation center for the **Spin** command.

Rotation using Only the Keyboard: You can use the key combination **Alt** + ***arrow keys*** to rotate the model view about a vertical or horizontal axis. Similar to using the middle mouse button, the center of rotation, through which the vertical or horizontal rotation axis runs, is determined by the **Default Rotation About** selection in the *3D UI Options* dialog box.

For each active *Rotation* command, or when using the middle mouse button, the behavior differs depending on where you click and drag, as follows:

- **Click and drag the cursor near any border of the Modeler window's display area:**

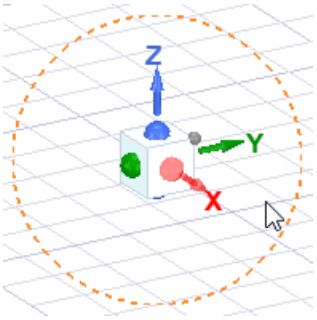
  The view is constrained to rotate about an axis that is perpendicular to the screen. The location of the axis, which is the rotation centerline, depends on the active rotation mode (Current Axis, Model Center, Screen Center, or Cursor) or, when no command is active, the *Default Rotation About* option previously described.

- **Click and drag the cursor within the interior of the display area (not close to a border):**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this case, the view rotates freely about any axis, depending on the direction you drag and the relative positions of the cursor and center of rotation. The rotation center depends on the active rotation mode (Current Axis, Model Center, Screen Center, or Cursor) or, when no command is active, the *Default Rotation About* option previously described.

- **Click and drag on or near the View Orientation Gadget to freely rotate the view:**



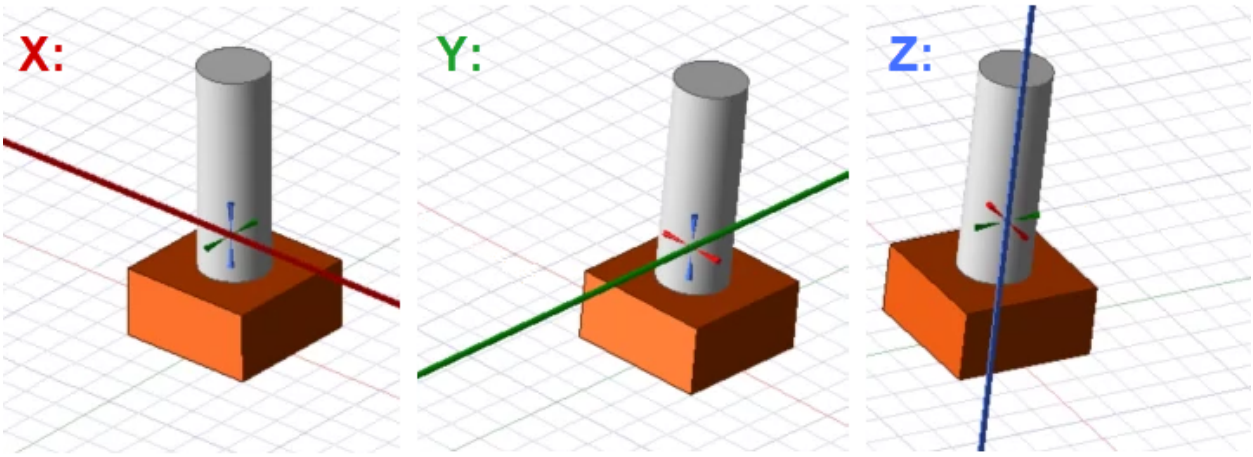
- **Constrained rotation about the X, Y, or Z axis**:

  Use the following hotkeys to constrain rotation of the model viewpoint to be about a coordinate system axis (global or user-defined). These hotkeys work whether you perform the rotation using the **middle mouse button**, any of the **Rotate** commands available on the ribbon (under the *View, Draw,* or *Model* tabs), or any of the **Rotate** commands in the menu bar (**View** > **Interactions** > **...**).

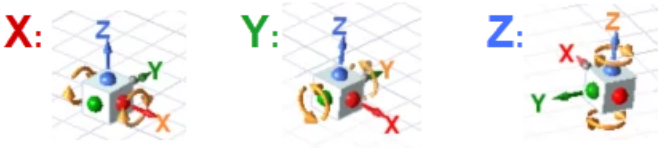
  - **X + click and drag** to rotate the view orientation about the active **X** axis.
  - **Y + click and drag** to rotate the view orientation about the active **Y** axis.
  - **Z + click and drag** to rotate the view orientation about the active **Z** axis.

  During constrained rotation, a heavy red (X), green (Y), or blue (Z) axis line appears in the Modeler window to provide a clear visual indication of the axis of rotation. These axes pass through the center of rotation, and this center is marked by crosshairs. When using the middle mouse button, the center of rotation is the centerpoint of the drawing canvas. For menu and ribbon commands, you can choose alternative rotation centerpoints (model center, cursor location, or origin), which are covered further down in this help topic. The following image shows the appearance of the rotation axes and center markers:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Additionally, a pair of gold, circular arrows appear on the view orientation gadget, indicating the axis of constrained rotation:



While the same constrained rotation functionality for 3D models is also available for 3D reports, the red, green, and blue axes do not appear in 3D report windows during axis-constrained rotation. However, the pair of circular, gold arrows appear on the view orientation gadget in both the Modeler and 3D Report windows to indicate the axis of constrained rotation.

This feature provides a very convenient means of achieving the ideal model viewpoint for setup operations and for presentation purposes, giving you much more control than free rotation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Tip:**
>
> - For best results, drag the cursor in a roughly circular path around the rotation centerpoint. The model orientation will follow the cursor movement, with the angle of rotation matching the angle of the cursor's arc about the same centerpoint.
> - Hold down the X, Y, or Z key throughout the constrained rotation operation. The axial rotation constraint is canceled as soon as you release the key, though the rotation mode remains active. Depending on how you initiated rotation, cancel it by either releasing the middle mouse button, pressing **Esc**, or clicking the command again.
> - During an active constrained rotation operation, you can release the current hotkey and press one of the other hotkeys to choose a different rotation axis, without first canceling the operation.

- **Rotating the view about an axis normal to the screen:**

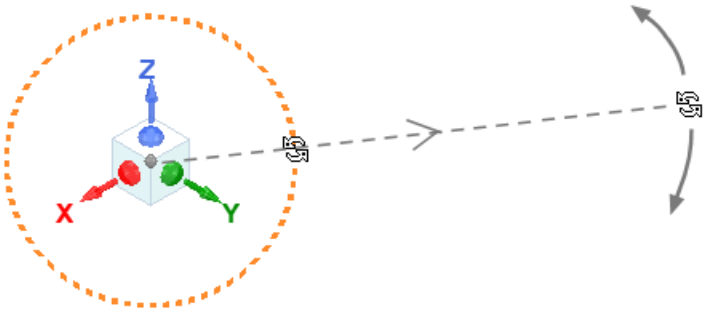
  This orange dashed circle that appears around the view orientation gadget when the cursor is nearby is used to rotate the current view about the screen-normal axis. That is, the view rotation is constrained to the plane of the screen regardless of the current viewing direction. The line weight of the dashed orange circle becomes heavier when the cursor is touching it, indicating a correct clicking point. There are three actions, as follows:

  - **Click anywhere on the circle:** The model rotates **90 degrees clockwise**.
  - **Ctrl + click anywhere on the circle:** The model rotates **90 degrees counterclockwise**. This action and the preceding one are particularly useful for changing the in-plane positive axis directions for standard top, bottom, left, right, front, or back views.
  - **Click and drag:** Freely rotates the model in either direction. Release the mouse button when the desired orientation is reached. While the mouse button remains pressed, the cursor changes to a rotation icon:

    You do not have to keep the cursor on the circle while dragging. You can often achieve more precise rotations with the cursor further away from the orientation gadget. Moving the cursor in a radial direction away from the center of the orientation gadget produces minimal rotation. Then, move it tangentially for precise

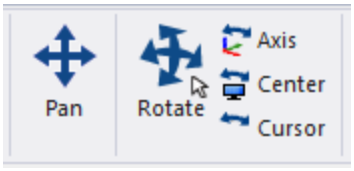
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

rotation. The greater the distance from the gadget, the finer the rotational control.



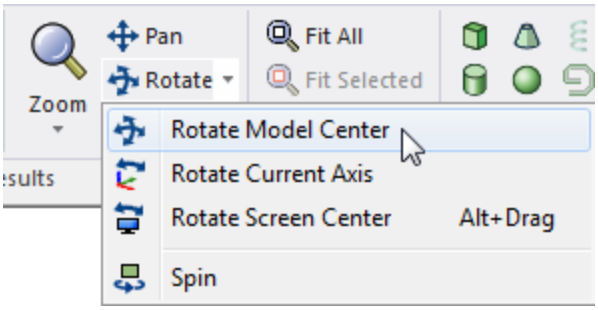
To rotate the view about the **Model Center**:

1. Choose the *Model Center* rotation mode using one of the following four methods:
   - From the **View** ribbon tab, choose **Rotate**:



> **Note:**
>
> The *Center* icon on the *View* ribbon tab is for rotation about the *screen center*, not the model center.

   - From the **Draw** or **Model** ribbon tab, choose **Rotate** > **Rotate Model Center**.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Tip:**
>
> After selecting a rotation mode, the selected option becomes the default operation for the remainder of the application session or until a different rotation option is selected. To repeat the last rotation command, you do not have to access the drop-down menu; just click **Rotate**:

- From the menu bar, click **View** > **Interaction** > **Rotate Model Center**.
- Right-click in the Modeler window and choose **View** > **Rotate** > **Rotate Model Center** from the shortcut menu.

2. Click and drag the mouse in the direction you want to rotate the view.
3. To exit the rotation mode, click the command again (per step 1) or press **Esc**.

Current Axis option – To rotate the view about the origin of the currently active coordinate system axes:

1. Choose the *Current Axis* rotation mode using one of the following four methods:
   - From the **View** ribbon tab, click **Axis**.
   - From the **Draw** or **Model** ribbon tab, choose **Rotate** > **Rotate Current Axis**.
   - From the menu bar, click **View** > **Interaction** > **Rotate Current Axis**.
   - Right-click in the Modeler window and choose **View** > **Rotate** > **Rotate Current Axis** from the shortcut menu.
2. Click and drag the mouse in the direction you want to rotate the view.
3. To exit the rotation mode, click the command again (per step 1) or press **Esc**.

To rotate the view around the **Screen Center**:

1. Choose the *Screen Center* rotation mode using one of the following four methods:
   - From the **View** ribbon tab, click **Center**.
   - From the **Draw** or **Model** ribbon tab, choose **Rotate** > **Rotate Screen Center**.
   - From the menu bar, click **View** > **Interaction** > **Rotate Screen Center**.
   - Right-click in the Modeler window and choose **View** > **Rotate** > **Rotate Screen Center** from the shortcut menu.
2. Click and drag the mouse in the direction you want to rotate the view.
3. To exit the rotation mode, click the command again (per step 1) or press **Esc**.

To rotate the view around the **Cursor** location:

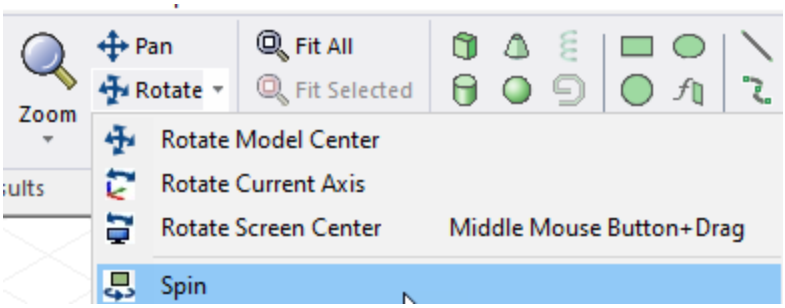
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Choose the *Cursor* rotation mode using one of the following three methods:
   - From the **View** ribbon tab, click **Cursor**.
   - From the menu bar, click **View** > **Interaction** > **Rotate Cursor**.
   - Right-click in the Modeler window and choose **View** > **Rotate** > **Rotate Cursor** from the shortcut menu.
2. Click and drag the mouse in the direction you want to rotate the view.
3. To exit the rotation mode, click the command again (per step 1) or press **Esc**.

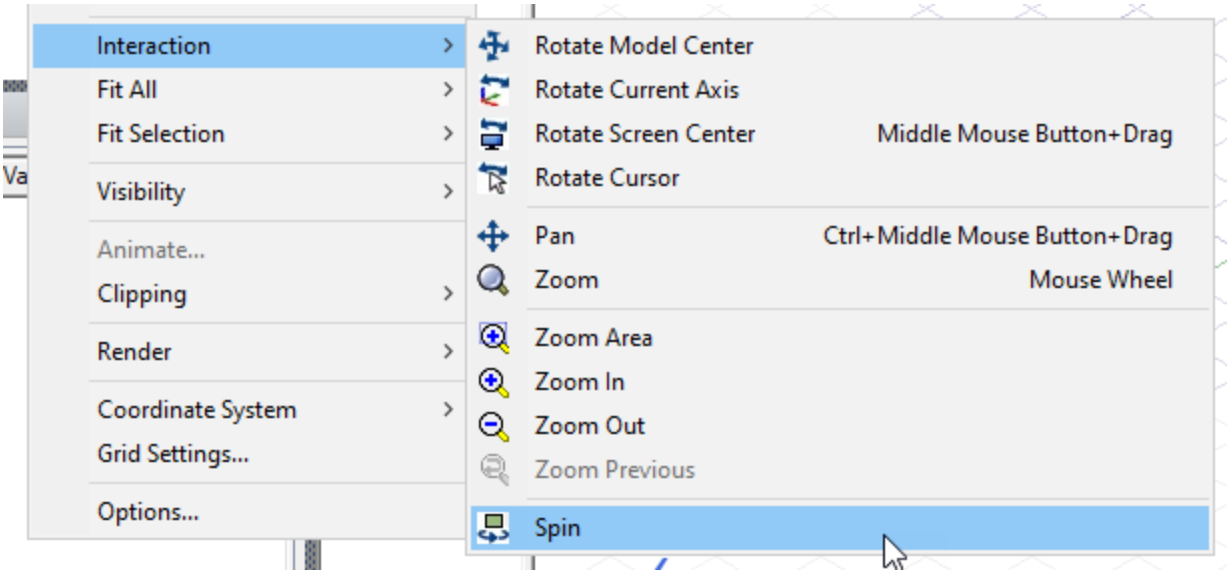
Spin Mode: This tool is useful for capturing animations (for presentation purposes) that show a 360 degree view of the model geometry, mesh, or simulation results.

To **Spin** the model view (that is, to keep it rotating after you release the mouse button):

1. Choose the *Spin* mode using one of the following two methods:
   - From the **Draw** or **Model** ribbon tab, choose **Rotate** > **Spin**.



   - From the menu bar, click **View** > **Interaction** > **Spin**.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

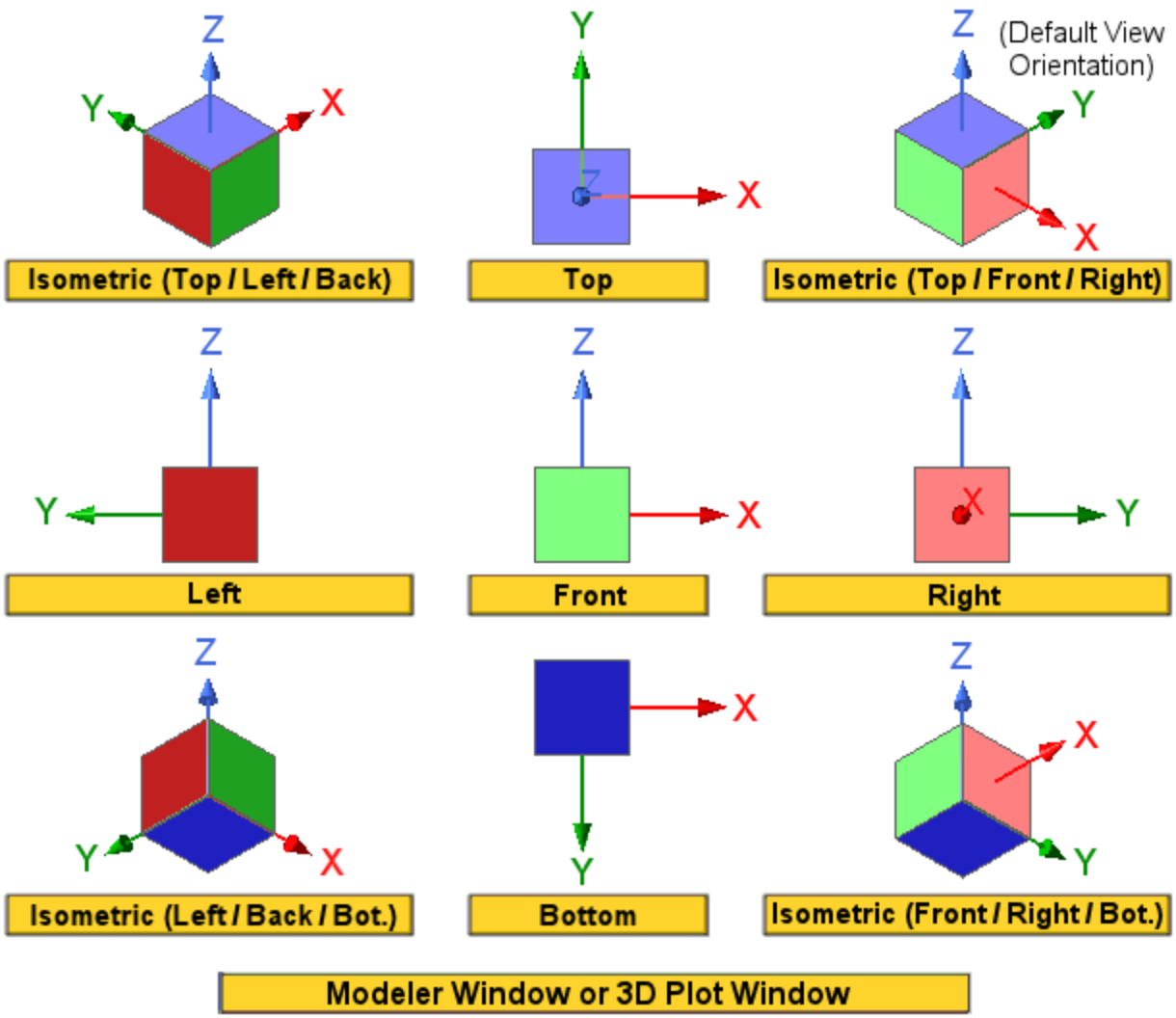
2. Click and drag the mouse in the direction you want to spin the view. Release the mouse button while still dragging.
    - The model will continue to spin in the specified direction. The spinning speed will be proportional to how fast you were dragging the cursor when you released the mouse button.
    - The center of rotation for the *Spin* mode is controlled by the *Default Rotation About* option selected in the 3D UI Options dialog box.
    - The *X, Y,* and *Z* keys also work as modifiers for the *Spin mode* (for constrained spinning about one of the active coordinate system axes).
3. To exit the *Spin* mode, click the command again (per step 1) or press **Esc**.

> **Note:**
>
> If rotation response is slow, especially for complex models:
>
> - For some graphics cards, you can improve the performance by setting **NVIDIA Control Panel** > **3D Settings** > **Manage 3D Settings** > **Global Settings** > **Global Presets: Workstation App - Dynamic Streaming**.
>
>   For more information about graphics cards, navigate to the appropriate Windows or Linux Ansys Installation Guide and search for **OpenGL**.

Changing the Model View with Alt+Double-Click Areas

In the following figure, the colored boxes show the nine view orientations you can obtain by using **Alt + double-click** in the corresponding areas of the *Modeler* window. Each face is a unique color and is named according to the standard view orientation names. The **Alt + double-click** view navigation method is also applicable to report windows containing *3D Plots* (such as a 3D Polar Gain plot).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The *Back, Trimetric,* and *Dimetric* standard view orientations are not available via *Alt* + *double-click* zones. For these three views, use other available ribbon commands or shortcut menus. For user-defined views, use the *Update View Orientation* dialog box or the Modeler window's shortcut menu (*Right-click* > *View* > *Apply Orientation*).

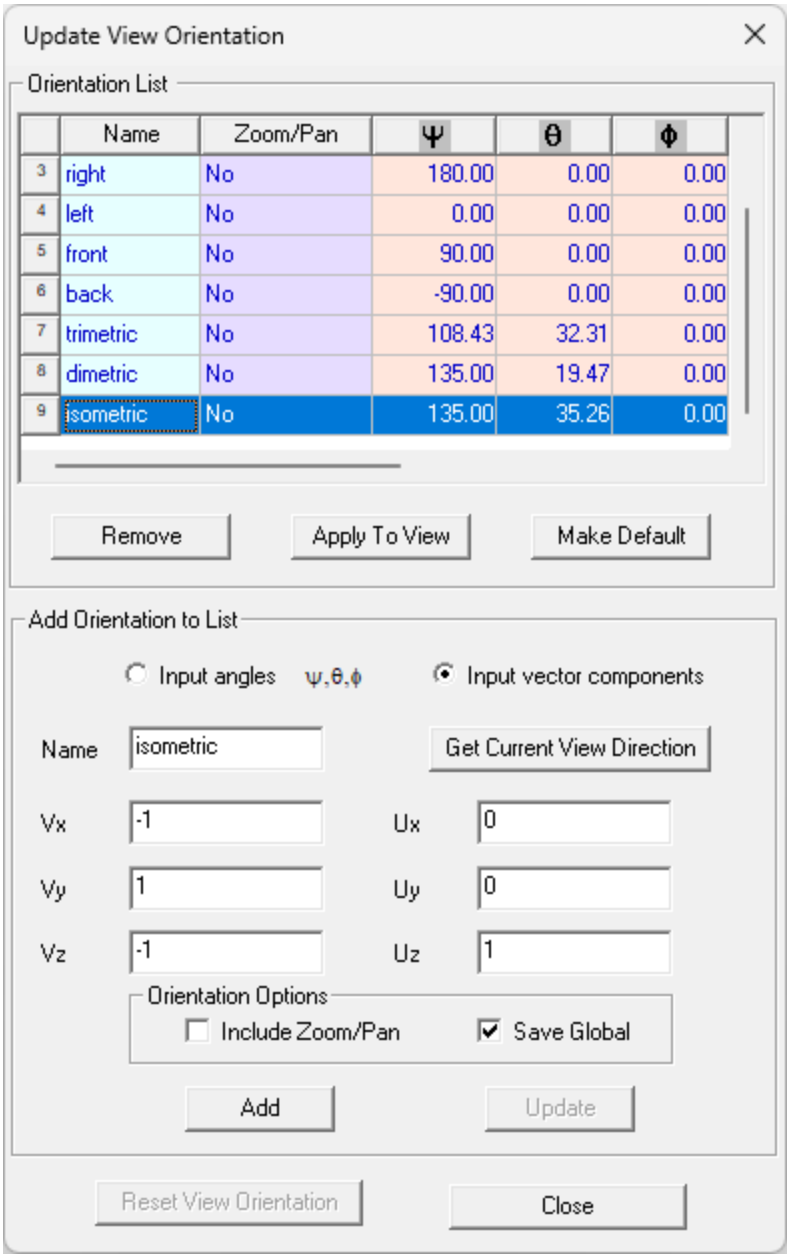
Applying a Default View Orientation

To apply a default viewing direction to the active view window:

1. Access the *Update View Orientation* dialog box using one of the following two methods:
   - From the **View** ribbon tab, click **List**.

     No icon is associated with this command.
   - From the menu bar, click **View** > **Modify Attributes** > **Orientation List**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click one of the orientation names listed in the orientation list.
3. Click one of the orientation names listed in the viewing directions list.
4. Click **Apply**.

   The viewing direction will be applied to the active view window.

   You can use the **Reset View Orientation** button to restore the view to the original angle.
5. Click **Close**.

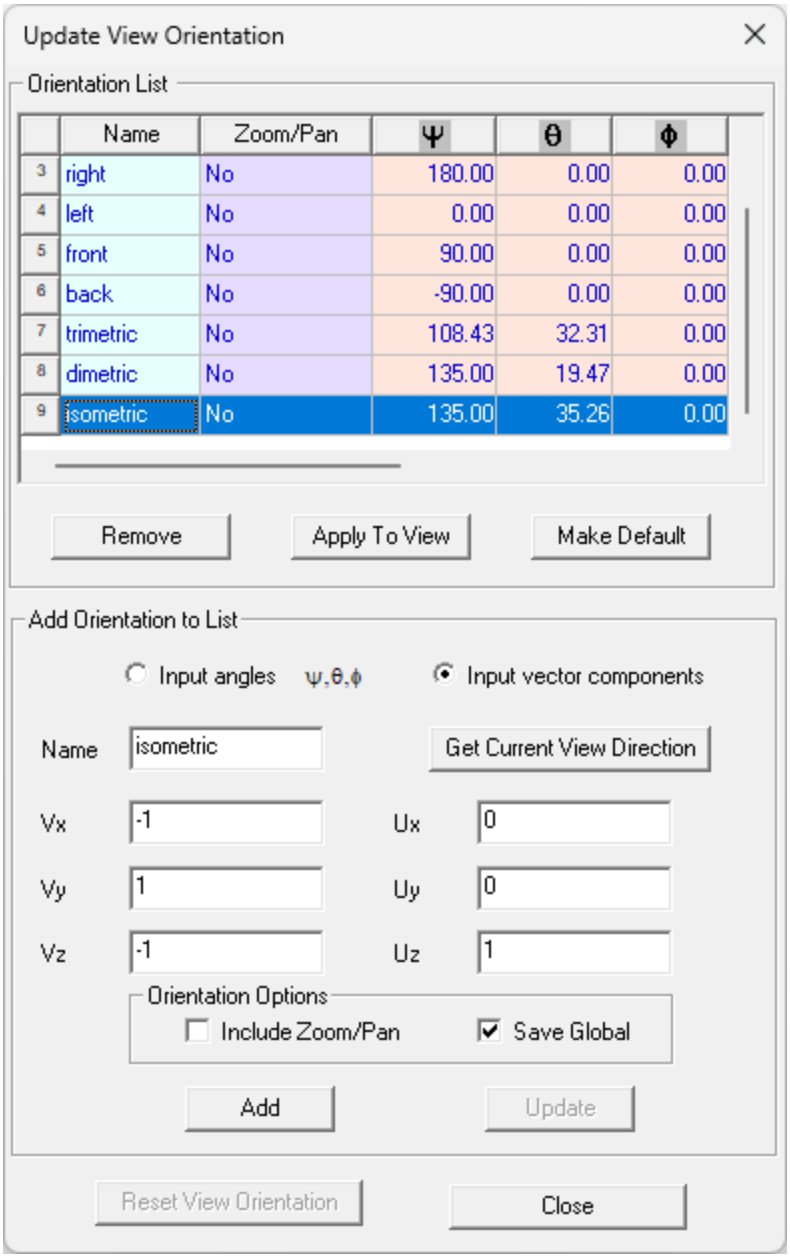
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Applying a New View Orientation

To create and apply a new viewing direction to the active view window:

1. Access the *Update View Orientation* dialog box using one of the following two methods:
   - From the **View** ribbon tab, click **List**.

     No icon is associated with this command.
   - From the menu bar, click **View** > **Modify Attributes** > **Orientation List**.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. To create a viewing direction that is based on a default viewing direction, click the existing orientation name in the viewing directions list.

   To create a viewing direction based on the current view in the **Modeler** window, click **Get Current View Direction**.

   - To modify the selected orientation's vector components, select **Input vector components** under **Add Orientation to List** and then modify the values in the **Vx, Vy**, or **Vz** text boxes, and the **Up vector** boxes for **Ux, Uy,** and **Uz**.
   - To modify the selected orientation's input angles, select **Input angles** under **Add Orientation to List** and then modify the values in the **psi, phi** and **theta** text boxes.
3. Type a name for the new orientation in the **Name** text box.
4. Click **Add/Edit**.

   The new orientation is added to the list of viewing directions.
5. Click **Make Default** if you want the new viewing direction to be the initial viewing direction when a **3D Modeler** window is opened in the current project or future projects.
6. Click **Close**.

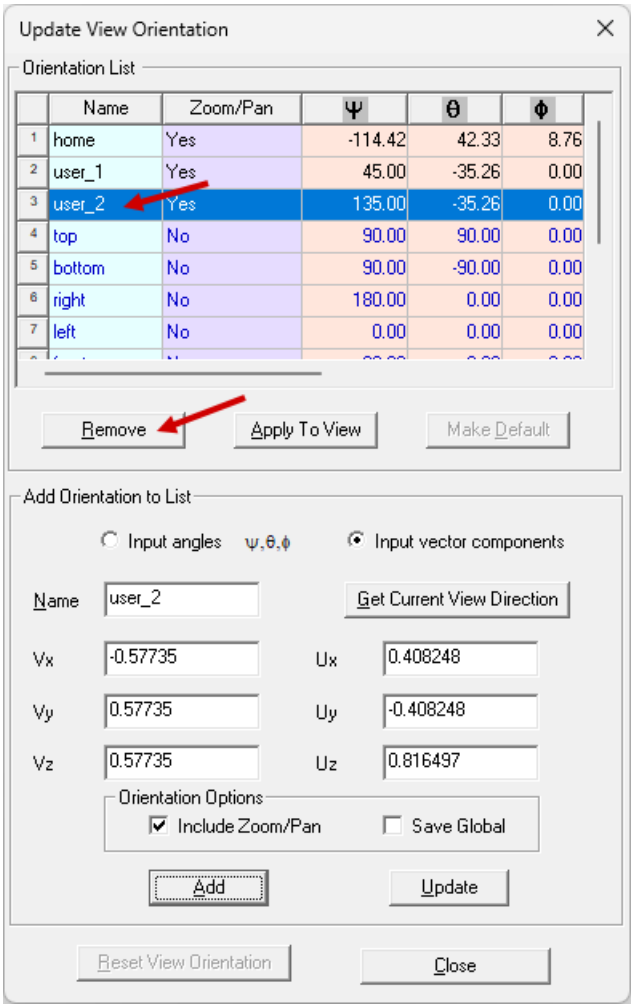
Removing an Orientation

To remove a viewing direction from the list in the orientation settings dialog box:

1. Access the *Update View Orientation* dialog box using one of the following two methods:
   - From the **View** ribbon tab, click **List**.

     No icon is associated with this command.
   - From the menu bar, click **View** > **Modify Attributes** > **Orientation List**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click the viewing orientation you want to delete from the list of names.
3. Click **Remove**.

   The viewing direction is removed from the list.

> **Note:**
>
> This operation cannot be undone.

Fitting Objects in the View Window

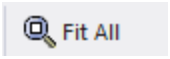
What do you want to do?

- Fit all objects or traces in a view window
- Fit selected objects in a view window

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fitting All Objects in a View Window

To fit all the views: click **View** > **Fit All** > **All Views** or use the equivalent icons in the **Draw** or **Model** ribbon tabs.



All view windows displaying the active design change to include all model objects.

To fit only the active view: click **View** > **Fit All** > **Active View**.

The view in the active Modeler window changes to include all model objects.

> **Tip:**
>
> Alternatively, fit all objects in the active view window using one of the following methods:
>
> - Press **Ctrl+D**
> - Right-click in the Modeler window and then click **View** > **Fit All** on the shortcut menu.

When **Fit All** is used in a **Report** view, the window is automatically rescaled to fit all traces in the window and the axis label and ticks are rescaled.

Fitting Selected Objects in a View Window

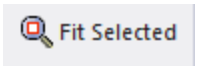
To fit one or more selected objects, 3D Components, or User Defined Models in the Modeler window or traces selected in the Reporter:

1. When you are working on a model view, select the objects you want to fit in the view. When you are working on a report, select the traces you want to fit.
   - To fit the selection in the active view window: Click **View** > **Fit Selection** > **Active View**.
   - To fit the selection in every open view window of the active design: Click **View** > **Fit Selection** > **All Views.**

     > **Tip:**
     >
     > Alternatively, fit the selection in the active view window by clicking **View** > **Fit Selection** on the shortcut menu.

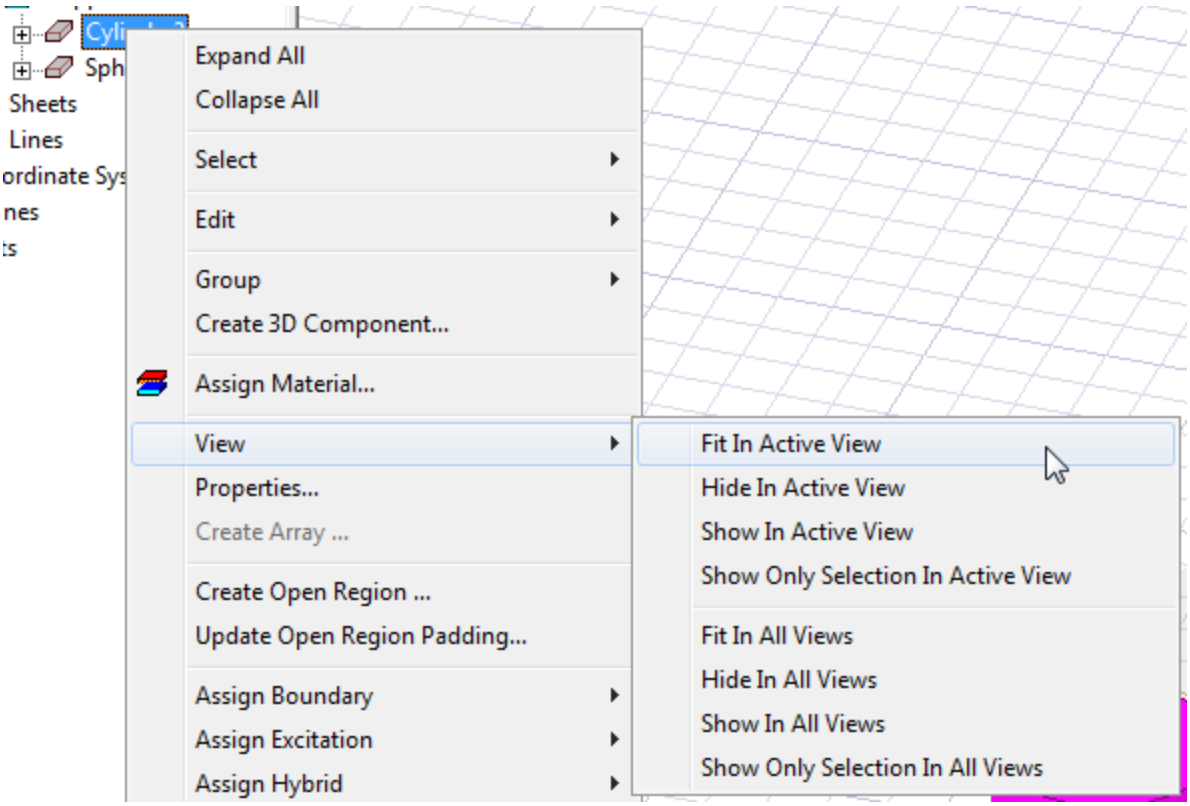
   You can also use the Fit Selected icon in the **Draw** and **Model** ribbon tabs:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To fit one or more objects or components selected in the History Tree:

1. Select the objects or components of interest and right-click to display the shortcut menu.
2. Select **View** > **Fit in Active View** or **Fit in All Views**:



.The view adjusts to fit the select objects.

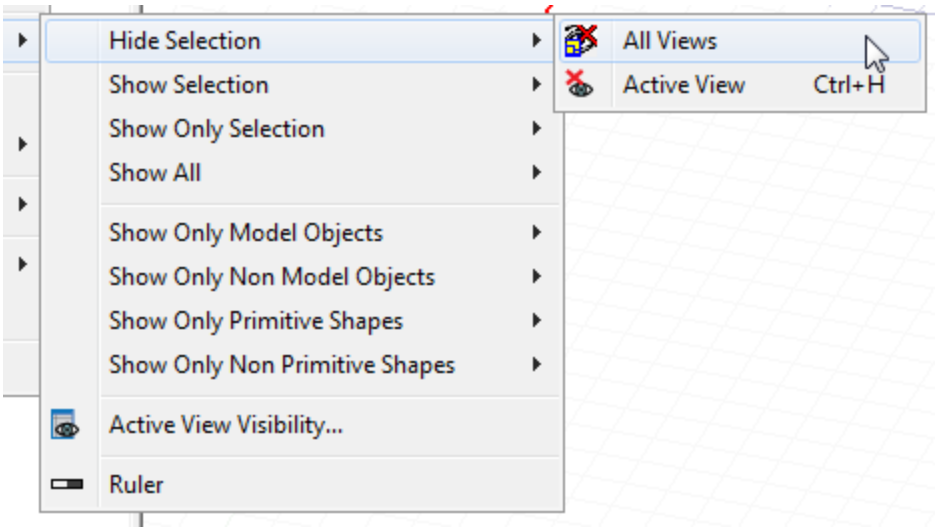
Hiding Objects from View

To hide selected objects.

1. Select the object you want to hide from view. This enables the menu commands and **View, Draw,** and **Model** ribbon tabs icons for hiding objects.
2. Click **View** > **Visibility** > **Hide Selection** and one of the following commands:
   - **All Views** to hide the selected object in every open view window.
   - **Active View** to hide the selected object in the active view window. You can also use **Ctrl+H** to hide a selected object.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also right-click in the **Modeler** window for a short-cut menu and use the **View** commands for **Hide Selection** or **Show Only Selection**.



You can also use the **Hide selected objects** icon in the **View, Draw,** or **Model** ribbon tabs to hide selected objects in the active view.

The objects you selected are hidden.

If there are many objects, you may find it easier to hide objects using the Active View Visibility dialog box. You can also choose to Show only selected objects in all or active views, effectively hiding all unselected objects.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

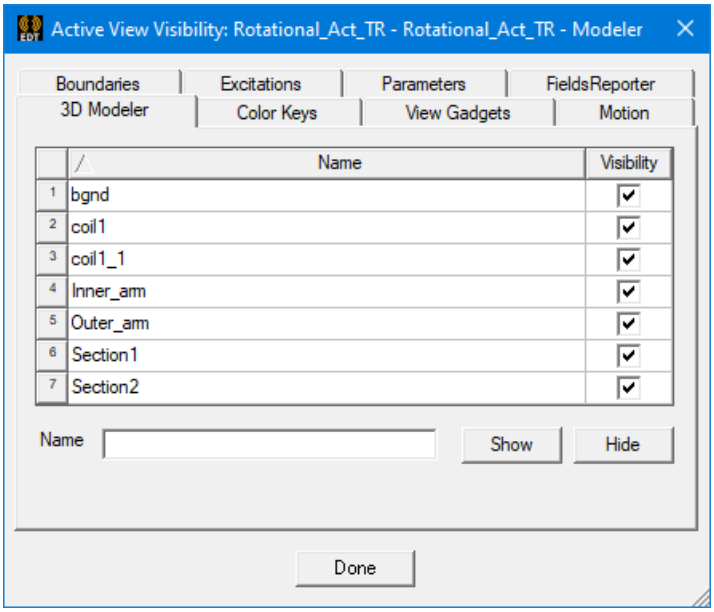
> **Note:**
>
> Hiding boundaries also turns off a check for boundary overlaps during boundary assignment. In the case of very large models with many boundaries, hiding boundaries can prevent delays during boundary assignment. Full model validation will subsequently check for boundary overlaps.

Object visibility is saved with the project.

Showing Objects

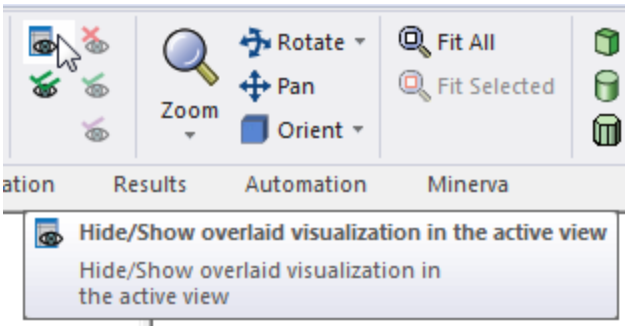
You can show or hide objects in several ways, using the **Active View Visibility** dialog box for individual objects, or **Visibility** menu selections for All Objects, Only Selection, Model Objects, Non-model Objects, Primitive Shapes, or Non-primitive shapes.

The **Active View Visibility** dialog box contains tabs for 3D Modeler, Color Keys, Array Setup, Boundaries, Excitations, and Fields Reporter. Select the tab for the objects you want to show or hide, and select or deselect **Visibility**.

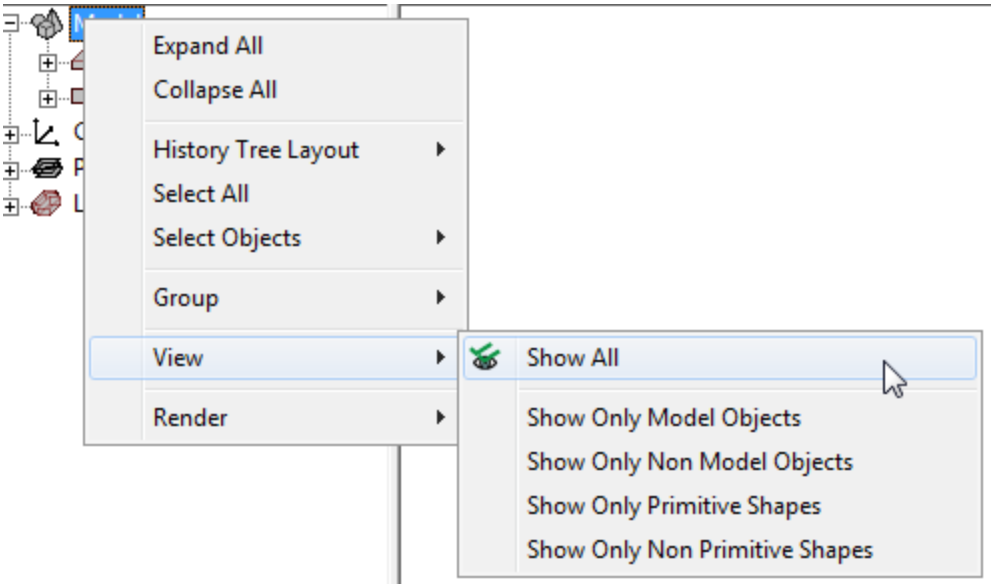


You can access the dialog box from the menu bar (**View** > **Visibility** > **Active View Visibility...**) or by clicking the Hide/Show visualization icons in the **View, Draw,** or **Model** Ribbon tabs.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Visibility** menus are available on the **View** menu and as shortcut menus in the *Modeler* window and in the History Tree when you right-click the **Model** icon:



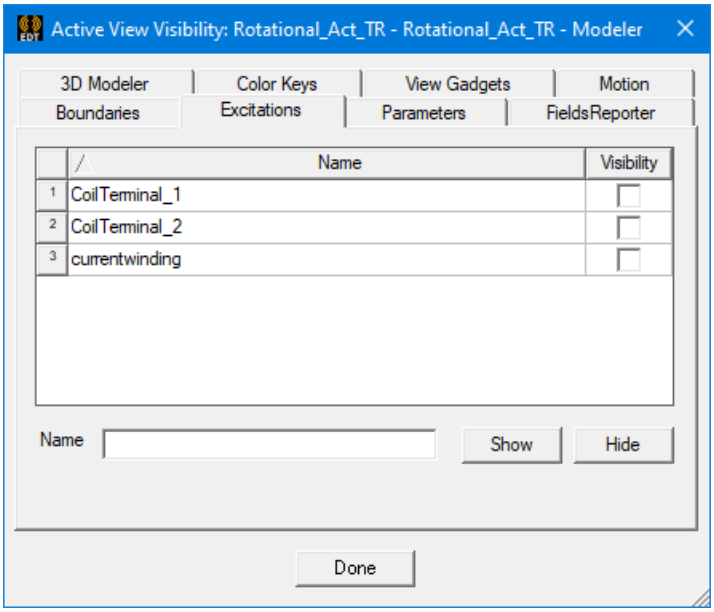
To show *one or more objects* that are currently hidden by using the *Active View Visibility* dialog box:

1. Click **View** > **Visibility** > **Active View Visibility...**, or click the visibility icons on the **View, Draw,** or **Model** ribbon tabs.

   The **Active View Visibility** dialog box appears.

2. Select the tab for the objects you want to show or hide. The dialog box contains tabs for 3D Modeler objects: Color Key objects, Boundaries, Excitations, and Fields Reporter objects.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Under the tab you need, select the **Visibility** option for the objects you want to show in the active view window.
    - For designs with large numbers of objects, you can resize the dialog box for easier selection.
    - By default, objects are listed in alphabetical order. You can invert the order by clicking the Name bar above the Name fields. A triangle in the bar indicates the direction of the listing.
    - You can also use the **Name** field to type in an object name and apply the visibility via the **Show** and **Hide** buttons.

    The objects you select and designate as Visible (by selecting the property or using Show) reappear.

> **Note:**
>
> Hiding boundaries also turns off a check for boundary overlaps during boundary assignment. In the case of very large models with many boundaries, hiding boundaries can prevent delays during boundary assignment. Full model validation will subsequently check for boundary overlaps.

To show *all objects* that are currently hidden by using the **Visibility** menu:

1. Click **View** > **Visibility** > **Show All** or use the shortcut menu in the Modeler window or History Tree and one of the following commands:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **All Views** to show all objects in every open view window
- **Active Views** to show all objects in the active view window.

The selected objects reappear.

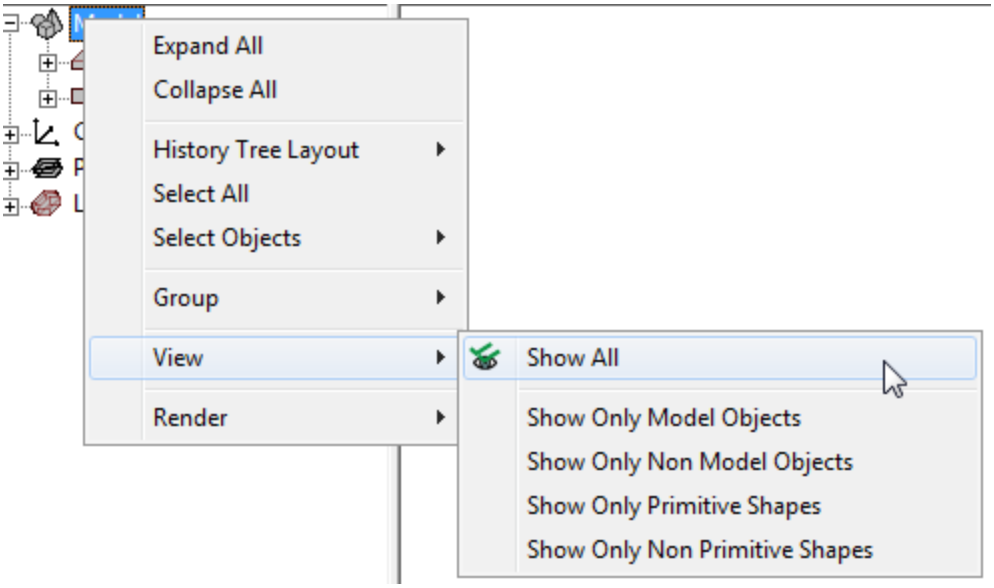
To show *selected objects* that are currently hidden:

1. Select the object. Hidden items are selected once the node corresponding to them is clicked in the history tree.
2. Click **View** > **Visibility** > **Show Selection** and one of the following:
   - **All Views** to show selected objects in every open view window
   - **Active Views** to show selected objects in the active view window.

   You can also use the **View, Draw,** or **Model** ribbon tab icons to **Show selected objects in active views**. Or, open the **Active View Visibility** dialog box.

To show one or more *3D components* that are currently hidden:

1. In the History Tree, select one or more 3D components. This activates the **Hide/Show** icons on the **View,Draw,** and **Model** ribbon tabs and the **View** menu selections.
2. You can right-click to display the shortcut menu and select **View** > **Show All**.



Object visibility is saved with the project.

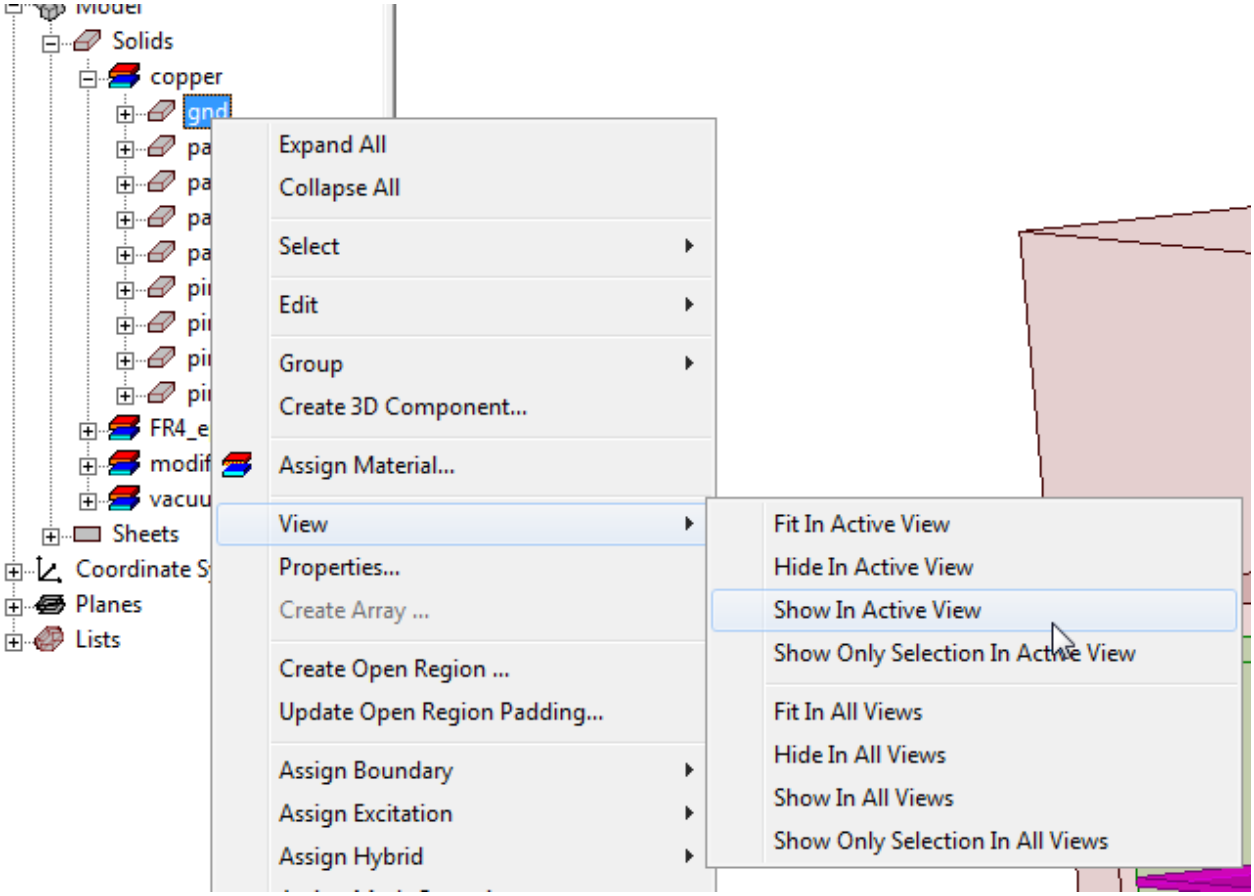
Showing Only Selected Objects in All or Active Views

To show only *selected objects*:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

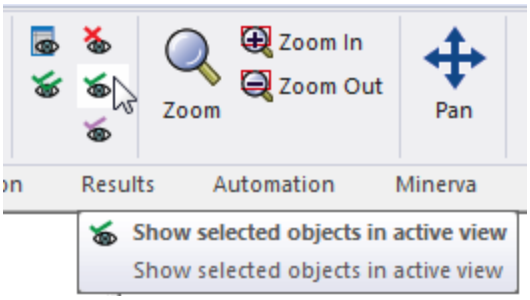
1. Select the object from the Project Manager or History Tree. Hidden items are selected once the node corresponding to them is clicked in the History Tree.
2. Click **View** > **Visibility** > **Show Selection** and one of the following:
   - **All Views** to show selected objects in every open view window
   - **Active Views** to show selected objects in the active view window

   You can also right-click an object in the History Tree and use the shortcut menu:
3. You can right-click to display the shortcut menu and select **View** > **Show in Active View, Show in All Views,** or **Show Only Selection In Active View**.



You can also use the **View/Draw/Model** ribbon icons to **Show selected objects in all views** and **Show selected objects in active views**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The selected objects reappear.

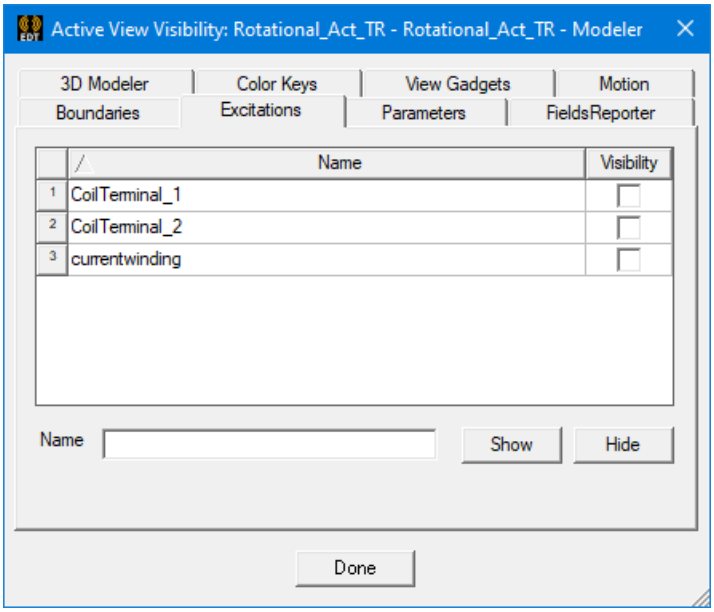
Active View Visibility Dialog Box

If there are many objects, it may be easier to show or hide objects using the **Active View Visibility** dialog box.

1. Click **View** > **Visibility** > **Active View Visibility**, or click the equivalent **Hide/Show** icon on the **View, Draw,** or **Model** ribbon tabs.

The **Active View Visibility** dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select the tab for the objects you want to show or hide. The dialog contains tabs for 3D Modeler objects, Color Key objects, Boundaries, Excitations, and Fields Reporter objects.
    - For designs with large numbers of objects, you can resize the dialog for easier selection.
    - By default, objects are listed in alphabetical order. You can invert the order by clicking the Name bar above the Name fields. A triangle in the bar indicates the direction of the listing.
    - You can also use the **Name** field to type in an object name and apply the visibility via the **Show** and **Hide** buttons.
3. Under the tab, clear the **Visibility** option for the objects you want to hide in the active view window.

    The objects you designate are hidden.

> **Note:**
>
> Hiding boundaries also turns off a check for boundary overlaps during boundary assignment. In the case of very large models with many boundaries, hiding boundaries can prevent delays during boundary assignment. Full model validation will subsequently check for boundary overlaps.

Object visibility is saved with the project.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rendering Objects as Wireframes or Solids

To render (display) *all* objects in the view window as wireframe outlines, flat-shaded solids, or smooth-shaded solids:

1. Click **View**> **Render** and click one of the following:
    - **Wireframe**:

      The objects in the view window are displayed as skeletal structures, enabling you to see all sides of the objects at one time.

      You can also use the **F6** key to display a wireframe view.

    - **Smooth Shaded**:

      The objects in the view window are displayed as shaded objects with smooth edges.

      You can also use the **F7** key to display a smooth shaded view.

To render a *single* object in the view window as a *wireframe outline*:

1. Select the object you want to render as a wireframe.
2. In the **Attribute** tab of the docked **Properties** window, select **Display Wireframe**.

Setting the Default View Rendering Mode

To set a default rendering mode for all objects created in the active design and in future designs:

1. Click **Tools** > **Options** > **Modeler Options**.
2. Click the **Display** tab.
3. Select one of the following options from the **Default View Render Mode** drop-down menu:
    - **Wireframe**.

      The objects in the view window will be displayed as skeletal structures, enabling you to see all sides of the objects at one time.

    - **Smooth Shaded**.

      The objects in the view window will be displayed as shaded objects with smooth edges.

4. Click **OK**.

   The rendering mode will be applied to all new objects you create.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Enhanced Display of Material Color and Transparency

You can specify the use of material color and transparency display by using the **View** > **Render** > **Enhanced Display** command. Any changes reset the default.

To set the Enhanced Display for the active modeler window:

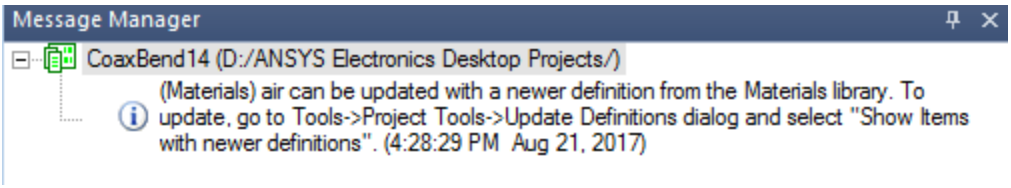
1. Click **View** > **Render** > **Enhanced Display** or press **F8**.

Enhanced Display overrides part color and transparency properties with material settings for all the parts visible in the active view. By default it is off (unchecked on the **View> Render** menu) and part attribute color and transparent values are applied.

It is possible for the view to be in the “Enhanced Display” mode but some of the parts in the model could have “Use Material Appearance” unchecked. In this case you can edit the **Color** or **Transparent** values, or select **Use Material Appearance** in the docked **Properties** window, but they are not applied to the active view visualization. This resembles how the **Wire Frame** setting is handled currently.

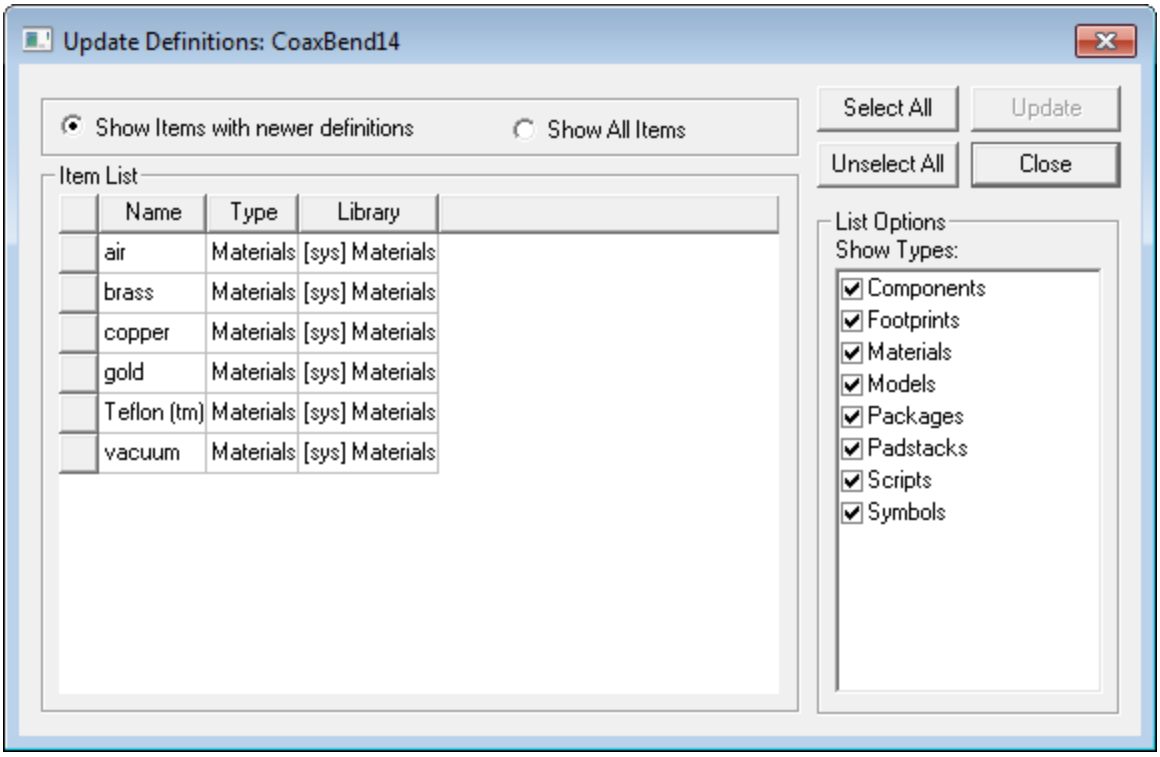
Sheet objects in the 3D modeler do not have a material assignment, but they could have a boundary assignment with material assigned to the boundary. In this case sheet object visualization resembles solid object visualization, where the material appearance from the assignment is used.

When you are working with a legacy project and use **View** > **Render** > **Enhanced Display**, if there are updates to the materials definitions for appearance, you will receive an info message informing you that the material definition was updated.



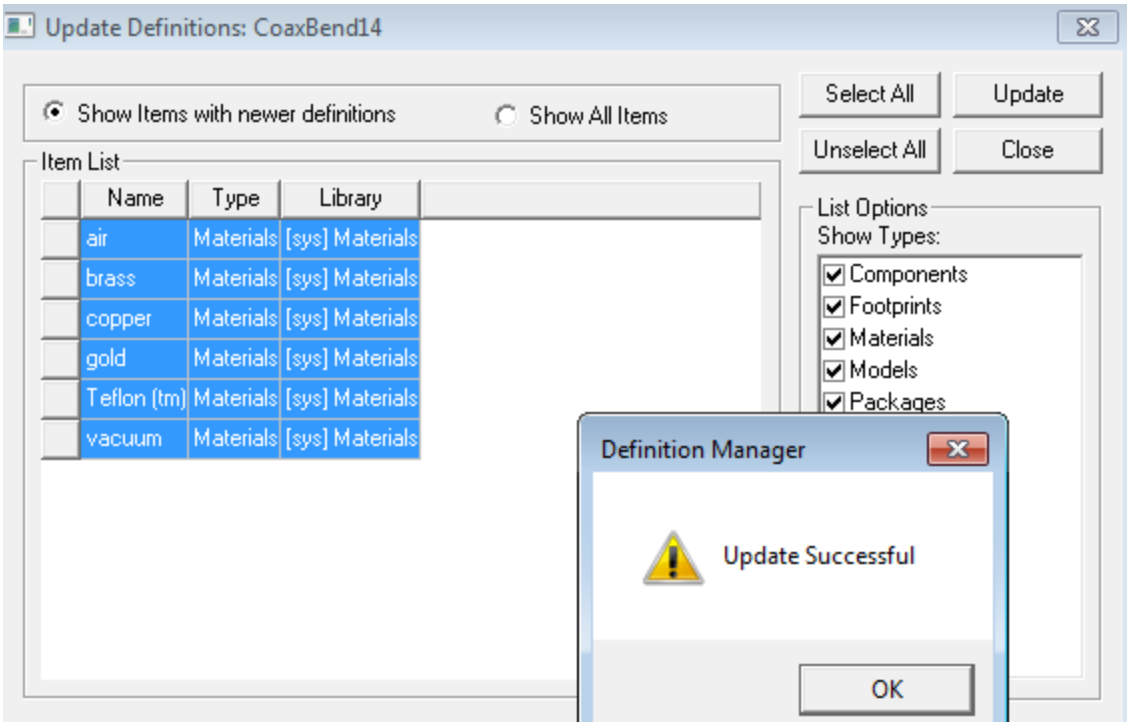
If you want to update, click **Tools** > **Project Tools** > **Update Definitions** to open the **Update Definitions** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Here you can multi-select rows of materials to update them. In case of updates available from multiple libraries, you can choose the library location. Selecting material rows enables the **Update** button. Selecting **Update** then produces a message on a successful update.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Setting the Curved Surface Visualization

You can specify the faceting for rendering true curves by using the **View** > **Render** > **Curved Object Visualization** command. There are two options for control--Maximum surface deviation and Maximum normal deviation. This resembles the Mesh surface approximation settings. Reduce either or both of the allowed deviations to improve the image quality. Improved image quality comes at the cost of increased CPU consumption. Changes apply to the current model until they are changed again. Any changes reset the default. The default gives satisfactory results (cpu/memory consumption vs. graphical display) for various model complexities.

When you change Curved Object Visualization settings and apply them to a design, those settings are saved with design unless you change it again. That means when you open the design again, it will apply saved visualization settings and NOT the default settings. Because this affects the CPU and memory required to open the project, typically, you should not save a project with other than the default settings.

To set the Curved Object Visualization for the active modeler window:

1. Click **View** > **Render** > **Curved Object Visualization**.

   This command displays the **Curved Object Visualization**dialog for the active modeler window. The dialog contains areas for setting the Maximum Deviation, and the Maximum Normal deviation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Set the Maximum Deviation by first selecting from the radio buttons for **Ignore**, set as **Relative Deviation** or set as **Absolute Deviation**. Selecting the later two radio buttons enables the value field.

   When set as **Relative Deviation**, the actual surface deviation depends on the model size. For example, sphere with a radius of 10 has same number of facets as a sphere with a radius of 1. This means that CPU cost does not increase based on the model dimension.

   When set as **Absolute Deviation**, the maximum surface deviation for both the spheres will be approximately same since a bigger sphere has more facets than a smaller one. This means that the most CPU cost applies to the larger objects.

3. If you selected the radio buttons for Relative or Absolute Deviation for Maximum Deviation, enter a value in the field.
4. To change the **Maximum Normal Deviation**, enter a value in the text field. Units are degrees.

   > **Note:**
   >
   > Wire bodies cannot be rendered with a Maximum Normal Deviation value less than 1 degree. When using a setting less than 1 degree all wire bodies will be rendered with a setting of 1 degree and all closed bodies will be rendered with the dialog box setting.

5. The **Save As Default** button lets you Save any values you change to the drop-down menus for the fields.
6. The **Restore Default**s button lets you return to the original values. Any values you provided through **Save As Default** remain on the drop-down menus for the fields for surface and normal deviations.
7. Click **Apply** to apply the current values to the active modeler window, and **Close** or **Cancel** to close the dialog box without changing settings.

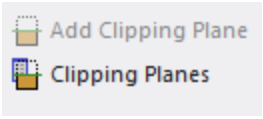
Using Clip Planes

The **View** > **Clipping** > **Clip Plane** command lets you define a clip plane that you can use to interactively make any desired cut-away view of a model. If you use **Edit** > **Copy Image** or **Modeler** > **Export**> **[image format]** with the clip plane active, the image shows the clipped plane. When parts of the model are hidden by a clip plane, model selection works as though only the visible parts are present.

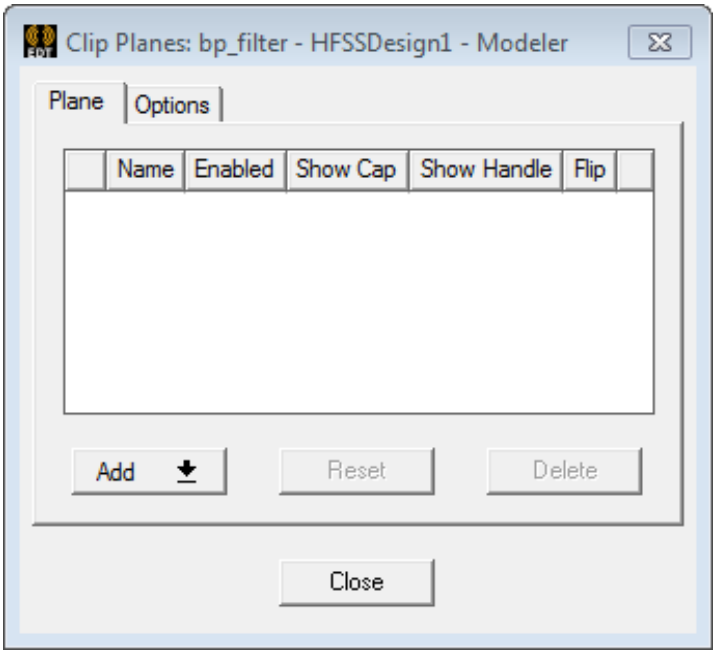
To add a clip plane:

1. Click **View** > **Clipping** > **Clip Plane** or click the **Clipping Planes** icon on the **View** ribbon tab.

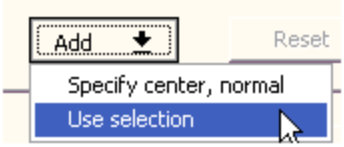
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This displays the **Clip Plane** dialog box with the **Plane** tab selected.

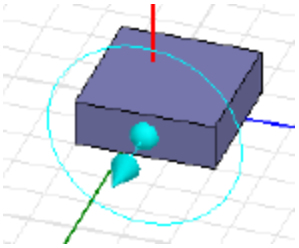


2. The **Add** button contains a drop-down menu with choices for **Specify center, normal**, and **Use selection**. If you want to use a selection, you must first select a face or an existing cut plane.

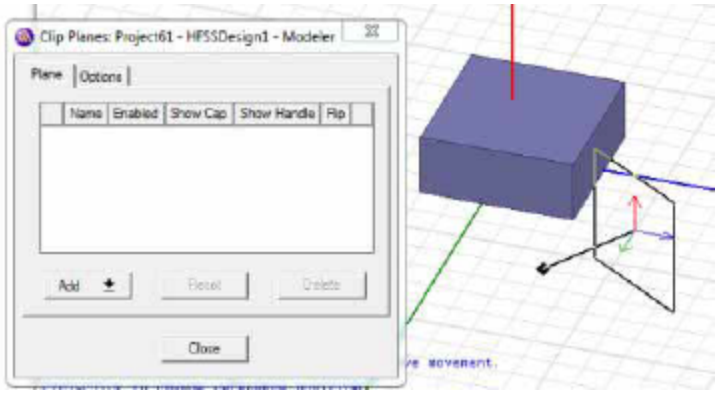


- If you first select a face or cut plane, and then click **Add** > **Use Selection**, the clip plane is added on that face.

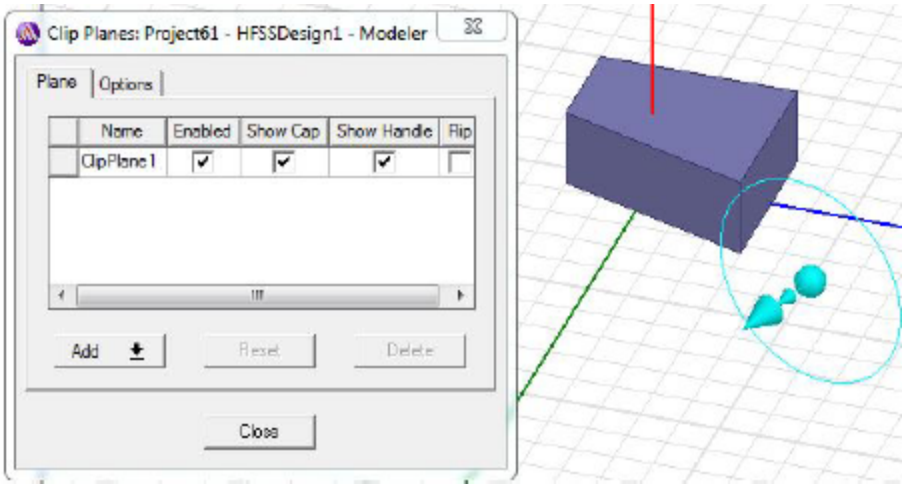
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- If you select **Add** > **Specify center, normal**, this launches a Measure dialog box and enters a mode for you to click to define the start location (shown as a triad).
  a. When you move the cursor, a rectangle represents the clip plane, and a vector the current direction.



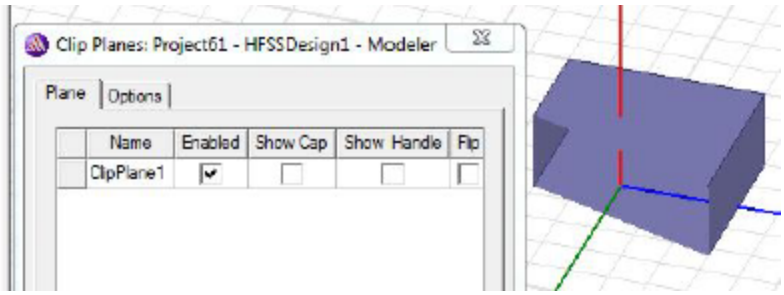
  b. Click again to set the reference position.



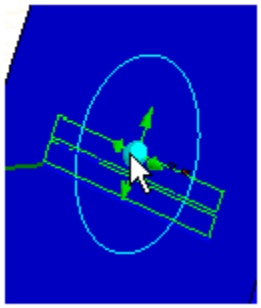
  After the second click, the clip plane is active. The handle is visible as a circle with a sphere at the center, and an arrow pointing the normal for the plane.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

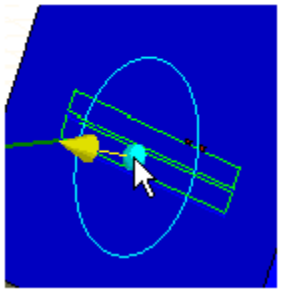
The **Clip Planes** dialog box shows the clip plane name, that it is enabled, the cap (which is the plane surface), and the handle. **Flip selection** lets you reverse the direction of the clip plane. If you deselect **Show cap** and/or **show handle**, they disappear from the display.



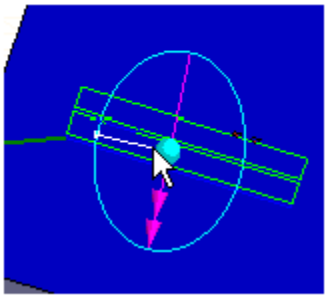
3. With **Show Handle** enabled, you can use the handle to manipulate the location and orientation of the clip plane. The handle changes appearance and function relative to the position of the cursor. Dragging the cursor makes use of the current function.



Move Handle independently of clip plane. This lets you move the handle away from the model.



Move Handle while dragging clip plane



Rotate clip plane by dragging the cursor. The white line is parallel to the clip plane rotation. The double arrow line is the axis.

Move the cursor around the center of the handle to change the axis.

4. The **Options** tab of the **Clip Planes** dialog box contains four options:
   - Force opaque for the unclipped portion.
   - Disable clip plane when drawing a new clip plane.
   - Plane handle color

     The button shows the current color. Click the button to display a color selection dialog box. Select a default or custom color and click OK.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Plane handle radius.

  This slider lets you resize the radius of the handle to the most convenient size. The radius resizes dynamically. When you close and reopen the modeler window, the last selected size persists.

You can save your choices as new defaults.

Modifying the Lighting

You have the option to emit the following types of light on a design:

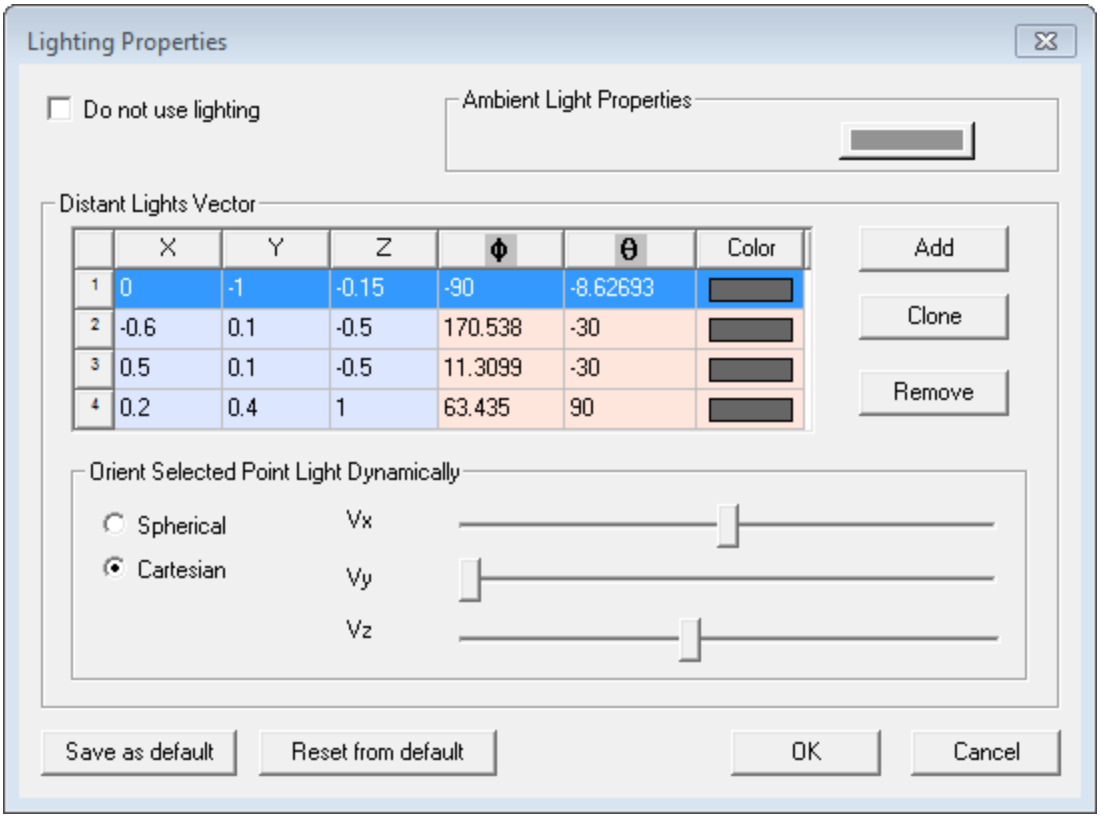
- *Ambient* lighting surrounds the model evenly with light. All objects are lit evenly in every direction by a color of light that you specify.
- *Distant* lighting directs a ray of light at the model in a direction you specify. By default, two distant light vectors are in effect for every new view window.

To modify the lighting:

1. Click **View** > **Modify Attributes** > **Lighting** or, on the **View** tab of the ribbon, click on **Modify Attributes** and select **Lighting** from the drop-down menu.

   The **Lighting Properties** dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. To turn off ambient and distant lighting, select **Do Not Use Lighting**.
3. To surround the model with light, click the **Ambient Light Properties** color button and then select a color for the surrounding light from the *Color* palette.
4. To modify the distant light on a model, do one of the following:
   - Add a new distant light by clicking **Add**.
   - Copy an existing distant light that you intend to modify by first selecting it in the **Distant Light Vectors** table and then clicking **Clone**.
   - Select a default distant light to modify by selecting it in the **Distant Light Vectors** table.
5. For the selected distant light vector, specify the vector direction:
   a. To modify the direction by specifying Cartesian coordinates, do one of the following:
      - Enter the new Cartesian coordinates in the **X**, **Y**, and **Z** boxes.
      - Use the **Vx**, **Vy**, and **Vz** sliders to specify the Cartesian coordinates dynamically.
   b. To modify the direction by specifying the spherical coordinates, do one of the following:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Enter the new spherical coordinates in the Φ (phi) and θ (theta) boxes.
- Use the Φ and θ sliders to specify the spherical coordinates dynamically.

6. To revert to the default ambient and distant light settings, click **Reset**.
7. Click **Save As Default** if you want the new lighting settings to be the defaults for all **3D Modeler** windows, either in the current project or future projects.
8. Click **OK** to dismiss the dialog box.

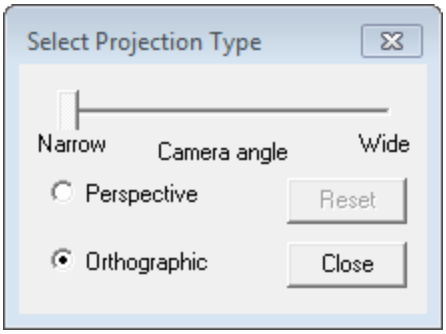
The lighting settings will be saved with the design. New lighting applied to other designs after this point, including new default settings, will not affect these lighting settings.

Setting the Projection View

To modify the projection of model objects (the camera angle) in the view window:

1. Click **View**> **Modify Attributes**> **Projection** or, on the **View** tab of the ribbon, click **Modify Attributes** and select **Projection** from the drop-down menu.

   The *Select Projection Type* dialog box appears:



2. Select **Perspective** to enable the slider to change the angle of the view.
   - Move the slider to the right to increase the proximity, or widen, the view. Move the slider to the left to decrease the proximity, or flatten, the view.

   Objects that are closer appear larger relative than objects that are farther away.

3. Select **Orthographic** to view the model without distortion.

   The slider is disabled because a distortion scale is no longer applicable.

4. Click **Reset** to return the model to its original view.
5. Click **Close** to accept the projection setting and dismiss the window.

   The *Select Projection Type* dialog box closes. The last view you specified in the projection window remains visible in the view window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

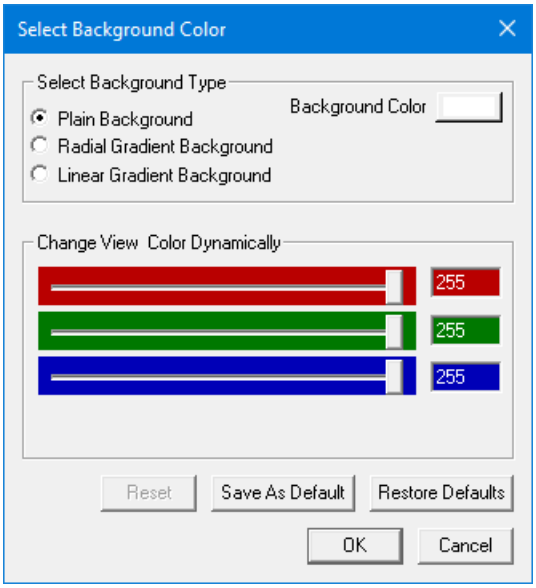
The projection view you set will be saved with the design. New projection views assigned to other designs after this point will not affect this projection setting.

Setting the Background Color

To set the color of the background in the view window:

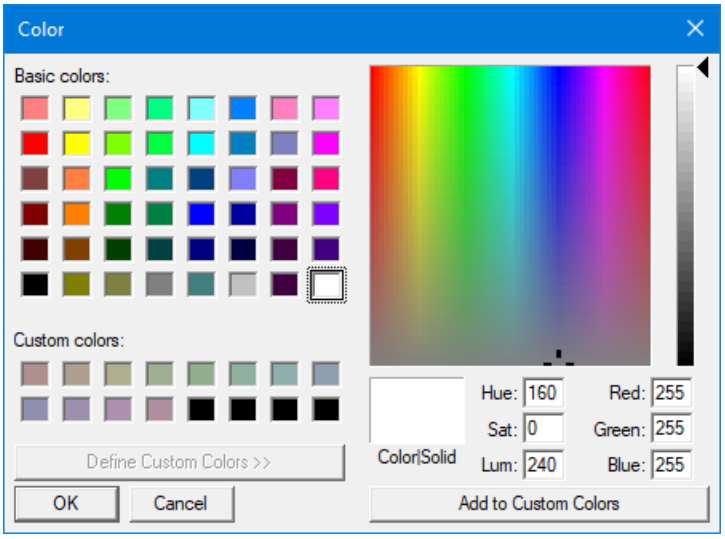
1. Click **View** > **Modify Attributes** > **Background color** or, on the **View** tab of the ribbon, click **Modify Attributes** and select **Background Color...** from the drop-down menu.

   The **Select Background Color** window appears.



2. To assign a solid background color, do the following:
   a. Select **Plain Background**.
   b. Modify the background color in one of the following ways:
      - Click the **Background Color** button and then select a color from the **Color** palette.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Use the RGB sliders under **Change View Color Dynamically** to specify the color's red, green, and blue values.

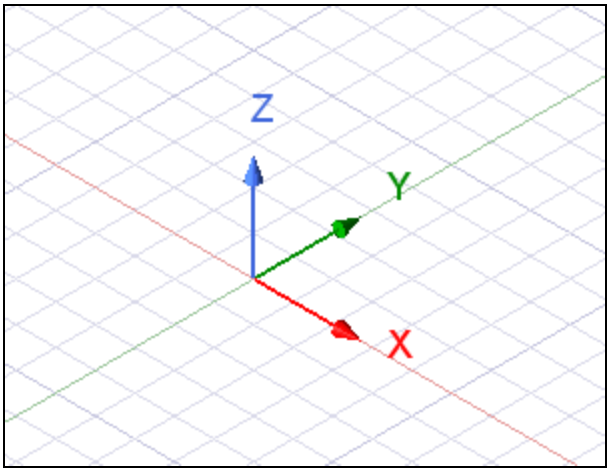
3. To assign a background color that gradually changes from one color to another, do the following:
   a. Select **Gradient Background**.
   b. Specify the background color at the top and bottom of the view window in one of the following ways:
      - Under **Select Background Type**, click the **Top Color** button and select a color from the **Color** palette. Then click **Bottom Color** and select a color from the **Color** palette.
      - Under **Change View Color Dynamically**, click **Top Color** or **Bottom Color** and use the RGB sliders to specify the color's red, green, and blue values.
4. Click **Reset** to revert to the default background colors.
5. Click **Save As Default** if you want the new background color to be the background color for all *3D Modeler* windows in either the current project or future projects.
6. Click **OK**.

The background color you set will be saved with the design. New background color settings assigned to other designs after this point, including new default settings, will not affect this design.

Modifying the Coordinate System Axes View

The coordinate system axes displays the x, y, z orientation from the origin point for the current working coordinate system. The axes use an XYZ = RGB coloring scheme.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



By option, you can also use a triad which is a secondary depiction of the coordinate system that appears at the lower right of the Modeler window. It shows the orientation of the currently selected working coordinate system.

What do you want to do?

- Show or hide the coordinate system axes
- Show the coordinate system axes for selected objects.
- Enlarge or shrink the size of the coordinate system axes
- Show or hide the triad axes

Showing or Hiding the Axes

1. Click **View** > **Coordinate System**, then click one of the following:
   - **Hide** to hide the x-, y-, and z-axes in the active view window.
   - **Show** to display the x-, y-, and z-axes in the active view window.

Show the Axes for Selected Objects

1. Click **Tools** > **Options** > **General Options**.

   This command displays the *Options* dialog box.
2. Expand the **3D Modeler** branch of the options tree.
3. Expand the **Display** sub-branch.
4. Select the **General** subgroup.
5. Select **Show orientation of selected objects**.
6. Click **OK** to close the dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enlarging or Shrinking the Axes

1. Click **View**> **Coordinate System**, then click one of the following:
    - **Large** to display the x-, y-, and z-axes as extending to the edges of the active view window.
    - **Small** to display the x-, y-, and z-axes in a smaller size in relative to the edges of the active view window.

Showing or Hiding the Triad Axes

The triad is a secondary depiction of the coordinate system that appears at the lower right of the Modeler window. It shows the orientation of the currently selected working coordinate system. It can be shown or hidden separately from the selected coordinate system.

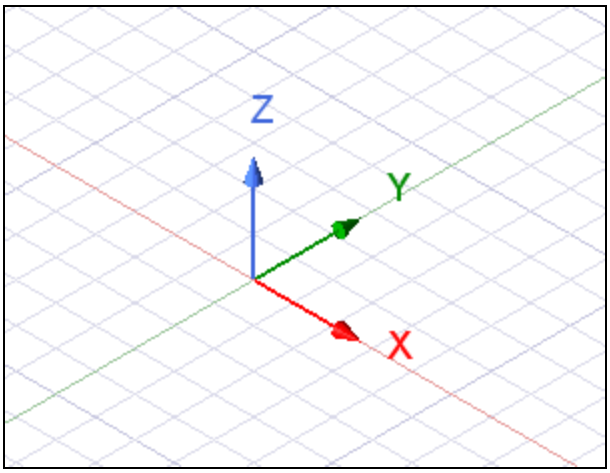
To show the triad:

1. Click **View**> **Coordinate System**> **Triad**, then click one of the following:
    - **Hide** to hide the triad x-, y-, and z-axes at the lower right of the active view window.
    - **Show** to display the triad x-, y-, and z-axes in the lower right active view window.
    - **Auto** to generally hide the triad axes.

Choosing Grid Settings

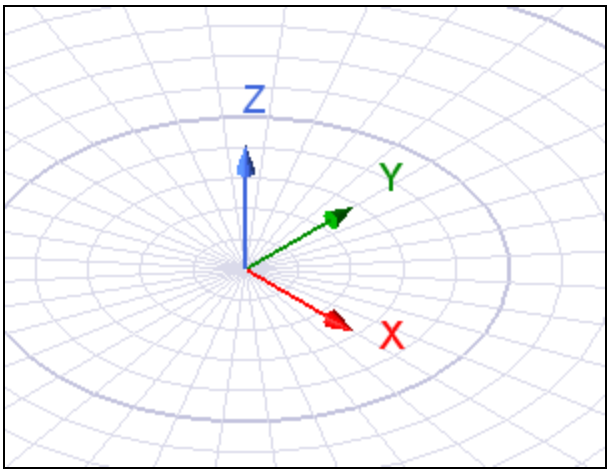
The grid displayed in the **Modeler** window is a drawing aid that helps to visualize the location of objects.

For Cartesian grids, the location of points on the grid are defined by intersections of planes that are perpendicular to and along the x-, y-, and z-axes. The division (the distance between neighboring parallel planes perpendicular to the same axis) can be set.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For polar grids, the location of points on the grid are defined by intersections of planes that are perpendicular to the local radius and angle coordinates. The division (the distance between neighboring parallel planes perpendicular to the same radius and angle) can be set.



Grid spacing is set according to the current project's drawing units. You can control the following aspects of the grid:

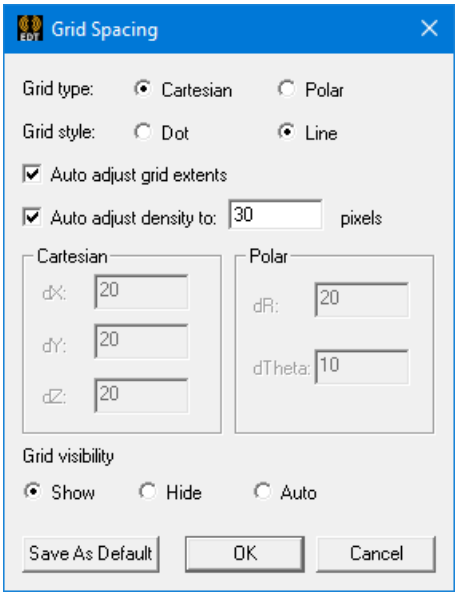
- Type (rectangular or circular)
- Style (dots or lines)
- Grid Extent (minimal for existing objects or as a plane)
- Density
- Spacing
- Visibility
- Snap settings
- Grid plane

Setting the Grid Type

1. Click **View** > **Grid Settings**.

   The **Grid Spacing** dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select a grid type for the active Modeler window: **Cartesian** for a rectangular grid or **Polar** for a circular grid.

   The grid in the active view window is centered at the origin of the working coordinate system.

For Cartesian grids, you define a coordinate by specifying its distance from the origin along each axis in the **X**, **Y**, and **Z** text boxes or its relative distance from the previously selected point in the **dX**, **dY**, and **dZ** text boxes.

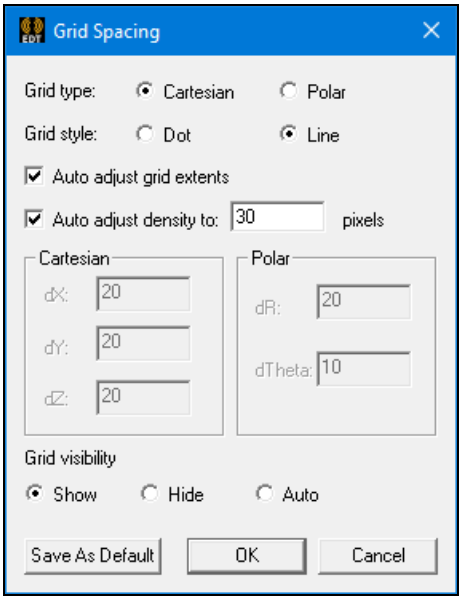
For polar grids, you define a coordinate by specifying its radius from the origin in the **R** text box and its angle from the x-axis in the **Theta** text box or its relative distance from the previously selected point in the **dR** and **dTheta** text boxes.

Setting the Grid Style

1. Click **View** > **Grid Settings**.

   The **Grid Spacing** dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select one of the following grid styles for the active view window:

   Dot Displays each grid point as a dot.

   Line Displays lines between grid points.

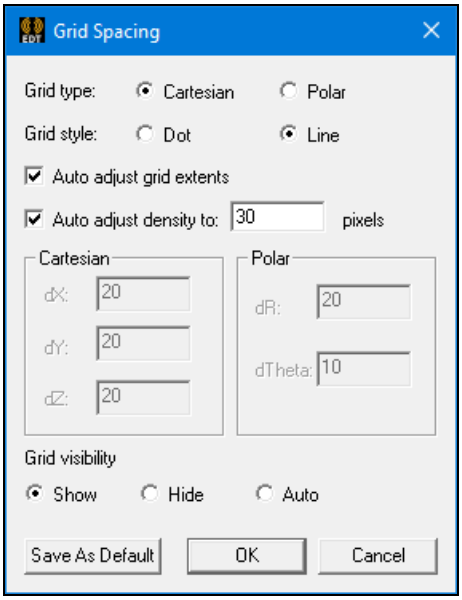
3. To show a Minimal grid around existing objects, check **Auto adjust grid extends**. To display a grid as a plane, deselect **Auto adjust grid extends**.

Setting the Grid Density and Spacing

1. Click **View** > **Grid Settings**.

   The **Grid Settings** dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. If you want to change the density of the grid in the active Modeler window as you zoom in or out on objects, do the following:
    a. Select **Auto adjust density to**.
    b. Specify a distance between grid points by typing a value in the **pixels** box.

    The default is set to 30 pixels, which is generally the best setting for displaying objects.

3. If you do not want the grid density to change when you zoom in or out, but instead want to specify a constant grid spacing, do the following:
    a. Clear the **Auto adjust density to** option.
    b. Specify the grid's spacing in the active design's units.

    If you selected a Cartesian grid, type the values of **dX**, **dY**, and **dZ**. These values represent the difference between one grid point and the next in the x, y, and z directions, respectively.

    If you selected a polar grid, type the values for **dR** and **dTheta**. **dR** represents the difference between each radius. **dTheta** is the difference between angles.

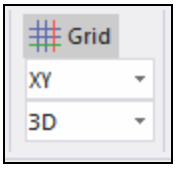
    The distance between grid points will increase and decrease proportionately as you zoom in and out in the active Modeler window.

4. Click **OK**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Grid's Visibility

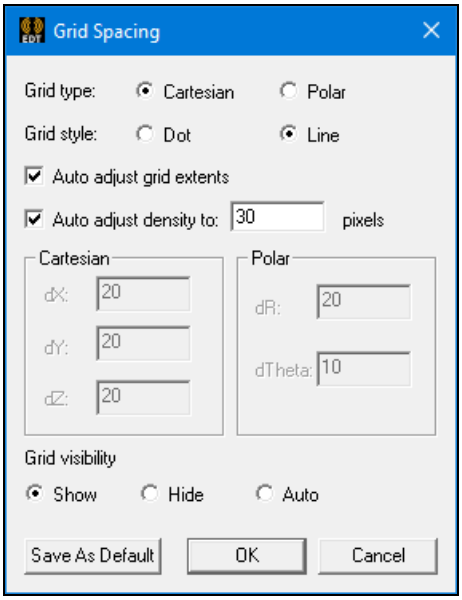
To hide the grid in the Modeler window, on the **Draw** ribbon tab, click the Grid toolbar icon. Click it again to show the grid.



Alternatively:

1. Click **View** > **Grid Settings**.

   The **Grid Spacing** dialog box appears.



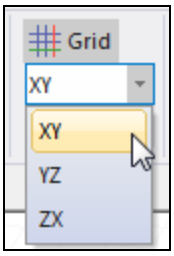
2. Select **Show** to make the grid always visible in the active **Modeler** window. Select **Hide** to make the grid always invisible in the active **Modeler** window. Select **Auto** so the grid appears only while you are drawing an object in the **Modeler** window.

Setting the Grid Plane

To specify the plane on which you want to display the grid in the active view window, do one of the following:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click **Modeler** > **Grid Plane** and then select a grid plane: **XY**, **YZ**, or **XZ**.
- Click a grid plane on the drop-down menu under Grid on the **Draw** ribbon tab:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9 - Maxwell User-Defined Primitives

User-Defined Primitives (UDPs) are compiled libraries of commonly used geometric models in electromechanical energy devices and power systems that can be added to the desktop interface and shared between users with common modeling needs. UDPs allow you to add customized geometric models to the Maxwell Desktop.

- In Maxwell, many commonly used UDPs have been defined, especially for RMxprt UDPs. For more information, see RMxprt UDPs and Helix UDPs.
- You can also define your own UDPs by writing source code and then generating .dll files for UDPs. For more information, see Source Code Files.

Maxwell User-Defined Primitives

User-Defined Primitives (UDPs) are compiled libraries of commonly used geometric models in electromechanical energy devices and power systems that can be added to the desktop interface and shared between users with common modeling needs. UDPs allow you to add customized geometric models to the Maxwell Desktop.

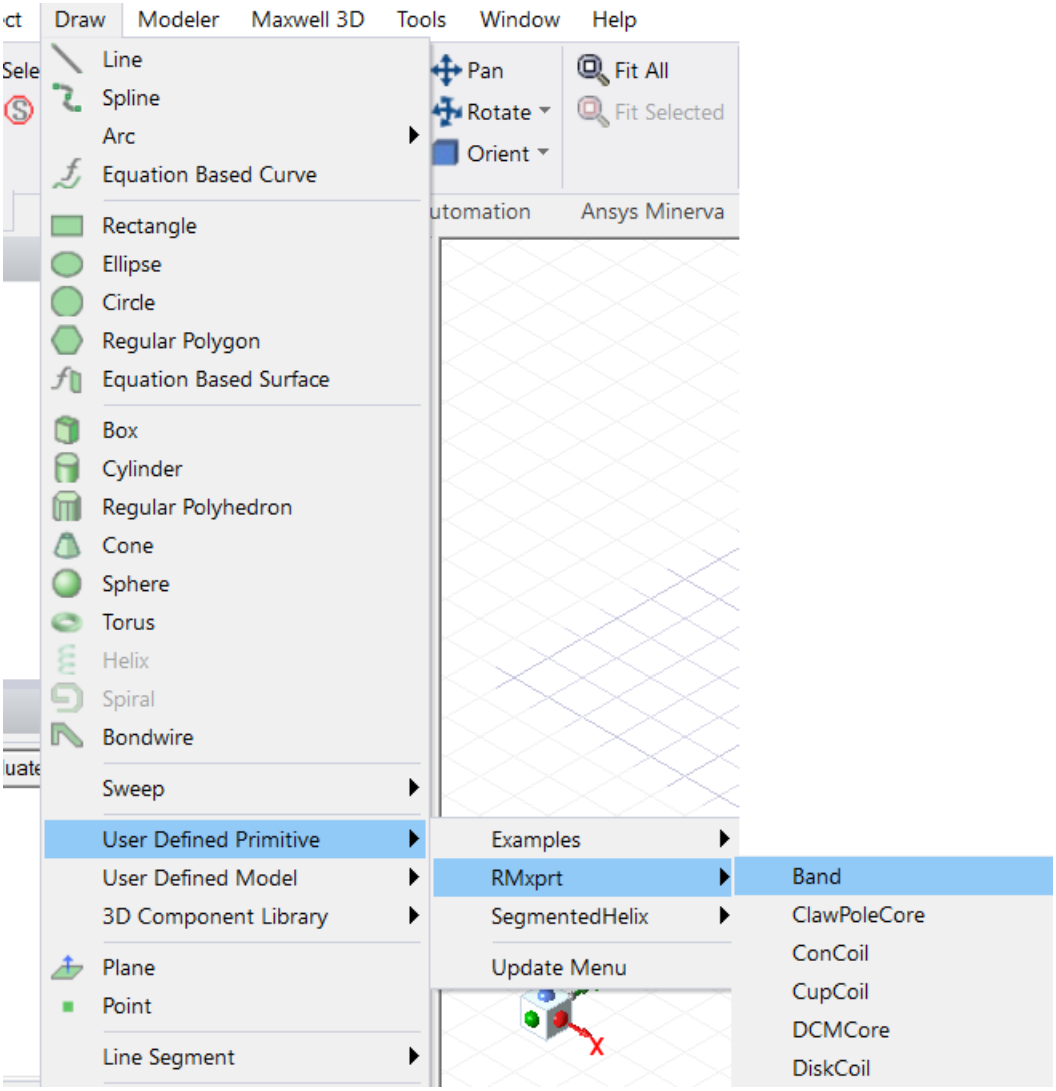
- In Maxwell, many commonly used UDPs have been defined, especially for RMxprt UDPs. For more information, see RMxprt UDPs and Helix UDPs.
- You can also define your own UDPs by writing source code and then generating .dll files for UDPs. For more information, see Source Code Files.

Launching a UDP from Maxwell Desktop

In the Maxwell desktop, click **Draw** > **User Defined Primitive** to choose a UDP. For example, a band UDP is launched as follows:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Draw** > **User Defined Primitive** > **RMxprt** > **Band**, as shown in the following figure**:**

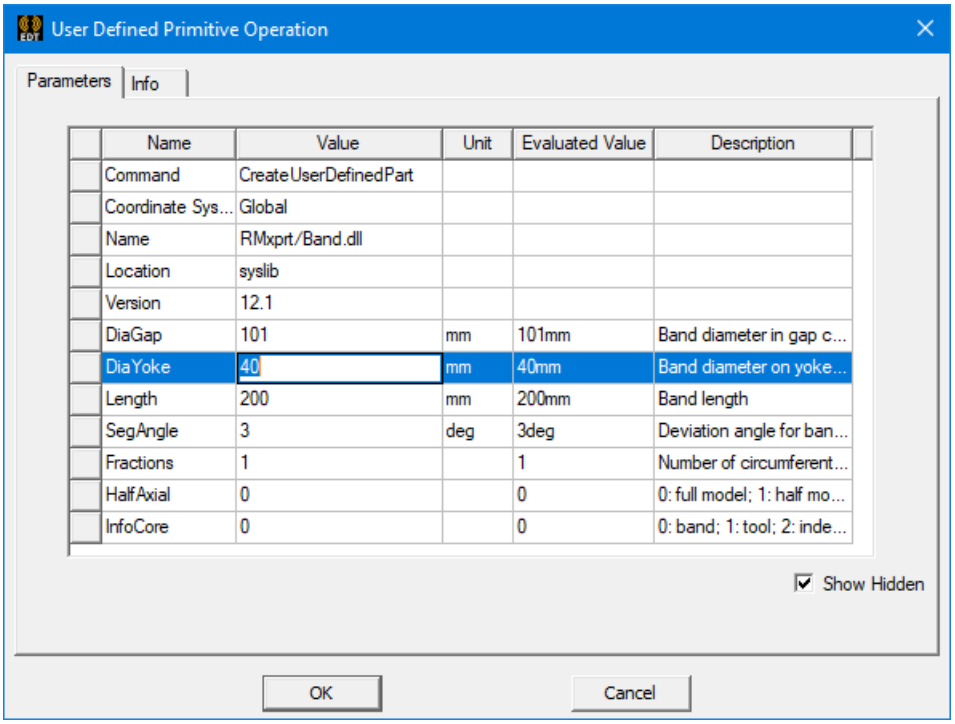


A **User Defined Primitive Operation** window opens.

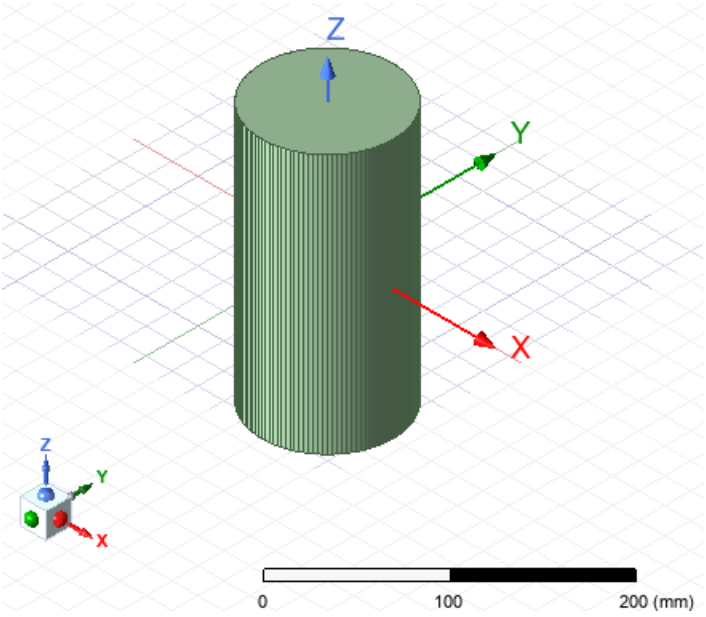
On the **Parameters** tab, you can view and edit the parameters. An **Info** tab contains information about the user defined primitive, its purpose, the company/author who created it, the date created and the version number.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Assign the values in the **Parameters** tab, and then click **OK** to finish setting parameters.



A Band UDP model appears in the coordinate system window:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RMxprt UDPs

RMxprt UDPs are 2D and 3D geometries created for finite element analysis of electrical machines.

RMxprt UDPs consist of the following types:

- UDPs for Band and Regions
- UDPs for Slot Cores and Coils
- UDPs for Pole Cores and Coils
- UDPs for Other Cores and Coils
- UDPs for Press Plates, Fingers, and Stems

UDPs for Bands and Regions

RMxprt includes two types of UDPs for defining bands and regions:

- Band UDP
- Core regions

Band UDP

The Band UDP is used to create

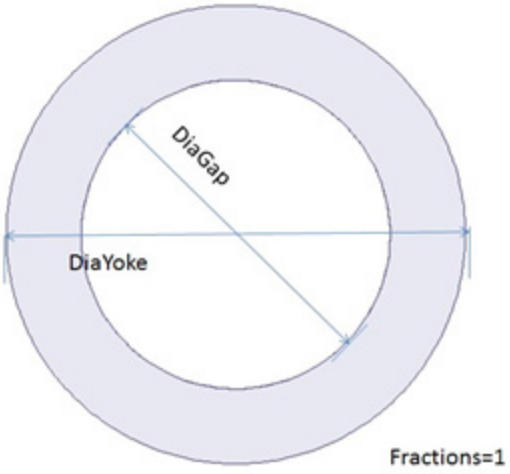
- a region to include all moving objects, which is called a “band”
- an outer region to include all moving and static objects
- a region used as a subtract tool to derive a partial model
- independent and dependent boundaries

Property	Description
DiaGap	Band diameter in gap center, DiaGap < DiaYoke for outer band
DiaYoke	Band diameter on yoke side, DiaYoke < DiaGap for inner band
Length	Band Length
SegAngle	Angle per segment of band (0.1 ~ 5 degrees)
Fractions	Number of circumferential fractions, 1 for circular region
HalfAxial	0: full model; 1: half model in axial direction.
InfoCore	0: band; 1: tool; 2: independent; 3: dependent; 100: region

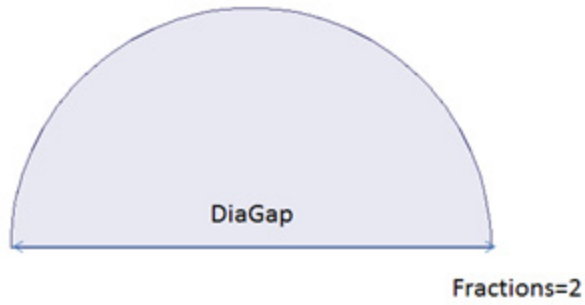
These parameters are used in the following 2D inner band and 2D outer band examples:

Inner band:

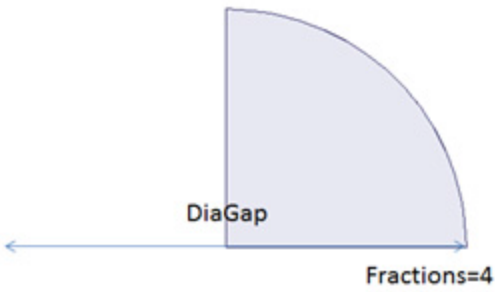
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Outer band:



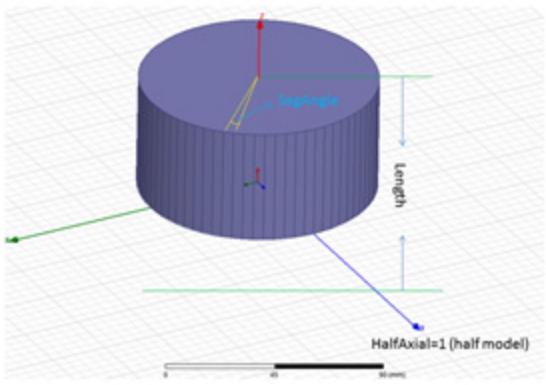
Inner band half:



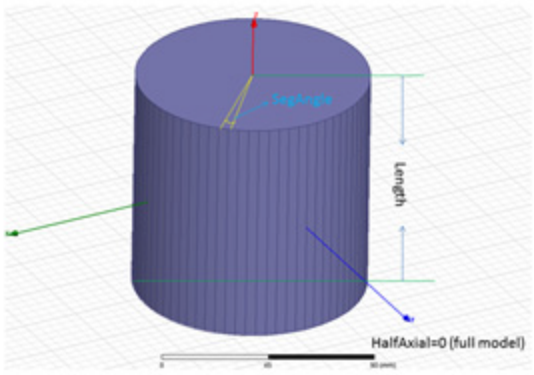
Inner band semi:

Parameters such as **SegAngle**, **Length** and **HalfAxial** are shown in the following 3D figures:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Inner band (full model):



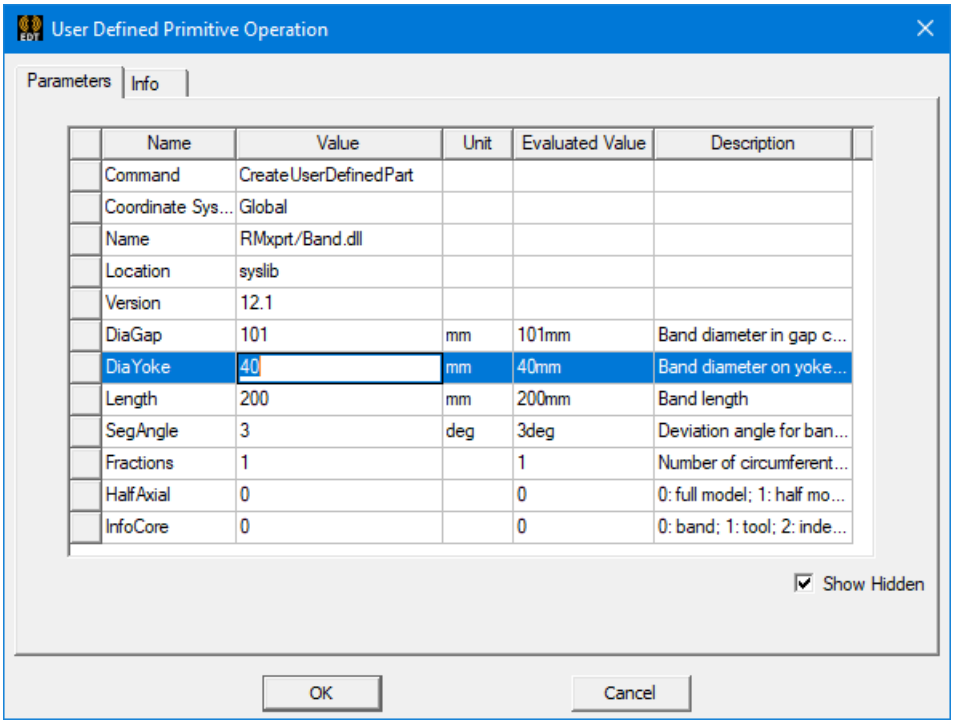
Creating a Band

Maxwell provides you with a .dll file to define the parameters of a band.

1. Click **Draw** > **User Defined Primitive** > **RMxprt** > **Band**.

   The **User Defined Primitive Operation** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Parameters** tab permits you to see and edit the parameters. An **Info** tab contains information about the user defined primitive, its purpose, the company/author who created it, the date created and the version number.

2. Specify the values for the parameters.
3. Click **OK**.

Creating an Outer Region

An outer region can be assigned by setting the **InfoCore** property to 100 either when creating the band or after creating it. The method for assigning during band creation is discussed in Creating a Band. The method for assigning after the band is created is as follows:

1. For assigning an outer region after the band is created, click **CreateUserDefinedPart** in the history tree under the specific band, as shown in "CreateUserDefinedPart in the History Tree" on the next page. You can change the **InfoCore** property to **100** under **Command** tab.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. For an outer region, set **DiaGap** < **DiaYoke**. For a shaft, set **DiaGap** > **DiaYoke**.

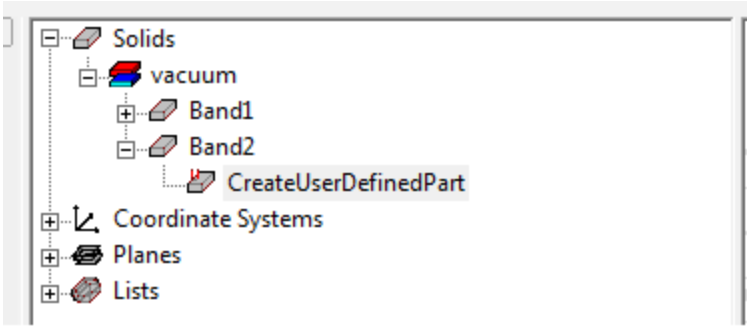


Figure 9-1 CreateUserDefinedPart in the History Tree

Creating a Tool

When you are defining parameters, the **InfoCore** property can be assigned to **1**, which means the band is used as a tool. This property can be assigned either when creating the band or after creating it. The method for assigning during band creation is discussed in Creating a Band. For assigning after the band is created, click **CreateUserDefinedPart** in the history tree under the specific band, as shown in "CreateUserDefinedPart in the History Tree" above. You can then change the **InfoCore** property to **1** under **Command** tab.

> **Note:** To be assigned as a tool, the band must be an outer band, which means the band's DiaGap value must be less than DiaYoke value.

A tool is mainly used for subtraction in the model.

For example, if a half model of NonSalientPoleCore UDP is needed, a tool is necessary to be used to subtract half of NonSalientPoleCore.

First, create a tool.

1. Click **Draw** > **User Defined Primitive** > **RMxprt** > **Band**.
2. Change **DiaGap** and **DiaYoke** values to set the **DiaGap** value to be less than the **DiaYoke** value.
3. Set **InfoCore** value to 1 and change the **Color** of the tool to another one in order to distinguish the tool from the NonSalientPoleCore, which is created later.

   Set the values of parameters to create a tool. Note that the DiaYoke parameter has to be larger than the DiaGap parameter.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

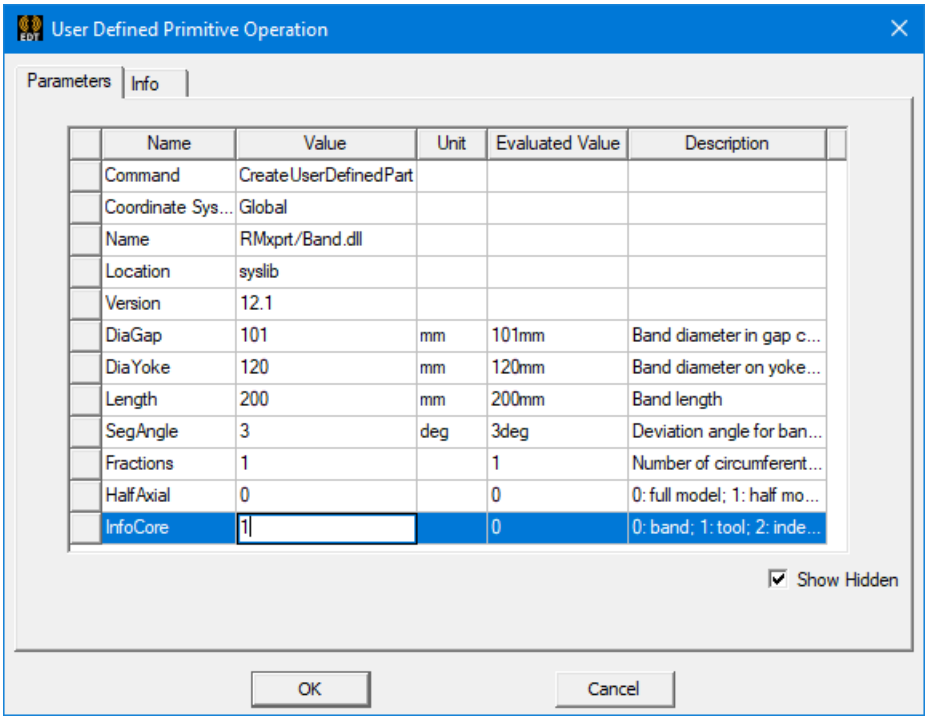
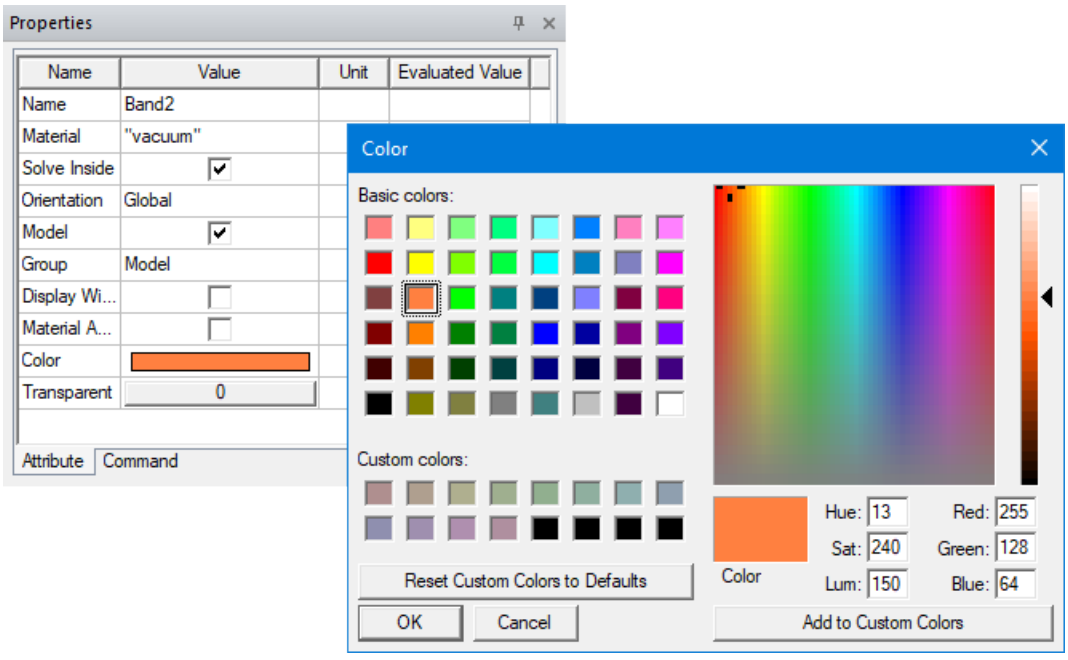


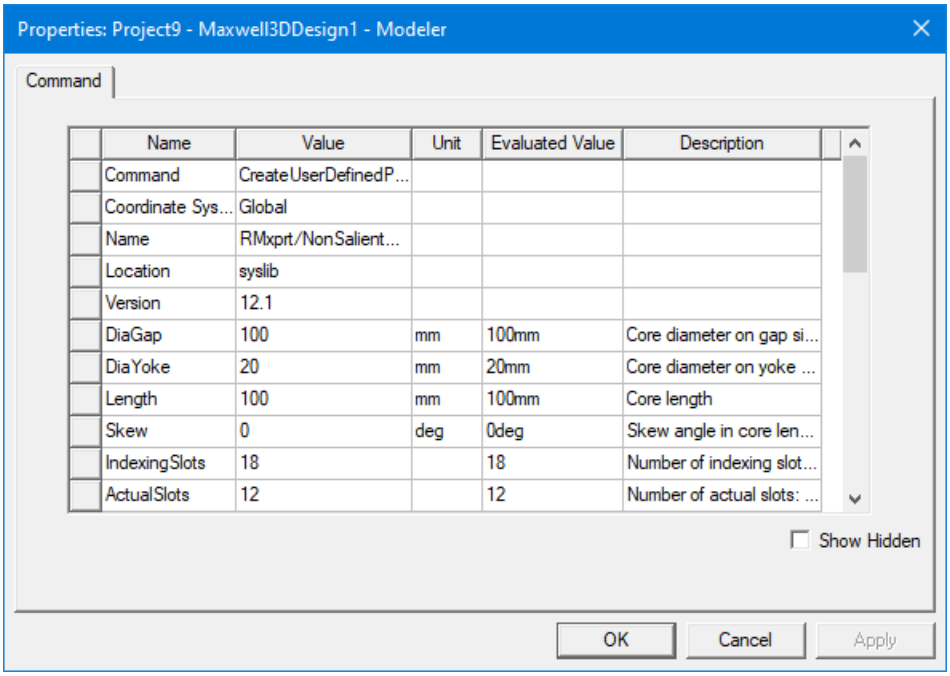
Figure 9-2 Change the color of the tool:



Then, create a non-salient pole core.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **Draw** > **User Defined Primitive** > **RMxprt** > **NonSalientPoleCore**. Apply the parameters shown below.



5. Click **OK** to finish setting the parameter.

   The coordinate system window looks similar to the following figure:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

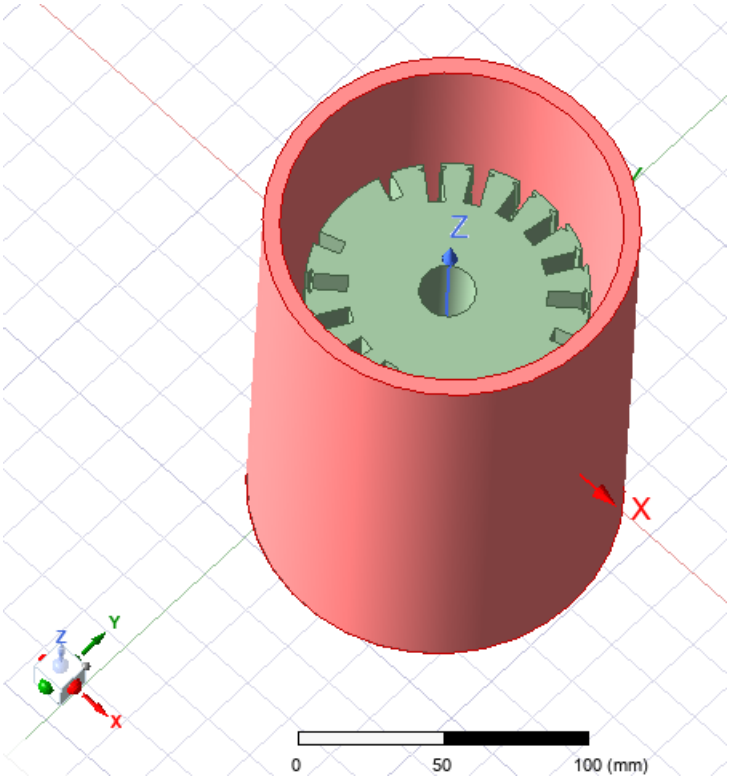


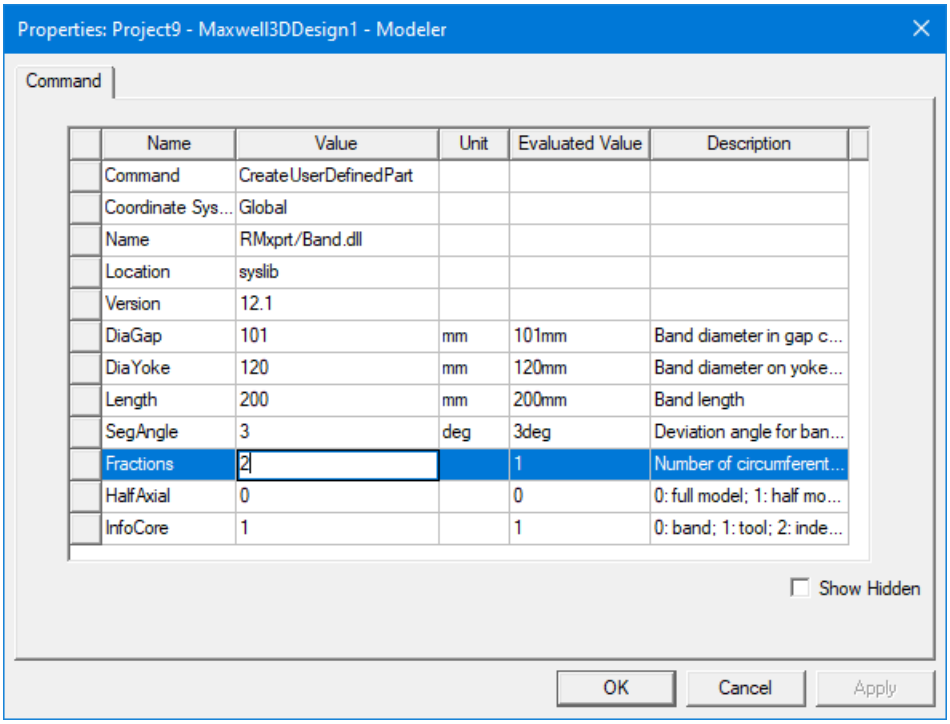
Figure 9-3 Non-Salient Pole Core (gray) with a Tool

Change the **Fractions** of the tool to 2:

1. Click **CreateUserDefinedPart** in the history tree under the tool (named Band1 in the figure).
2. Change the value of **Fractions** to 2.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **OK** to finish setting the parameter.



The last step is to subtract the non-salient pole core using the tool.

1. Choose NonSalientPoleCore1 and Band1 (tool) simultaneously by pressing **Ctrl** on the keyboard.
2. Click **Modeler** > **Boolean** > **Subtract** to do the subtraction.

   A Subtract window appears.

3. Choose NonSalientPoleCore1 as the **Blank Parts** and Band1 as the **Tool Parts**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **OK** to finish subtraction.

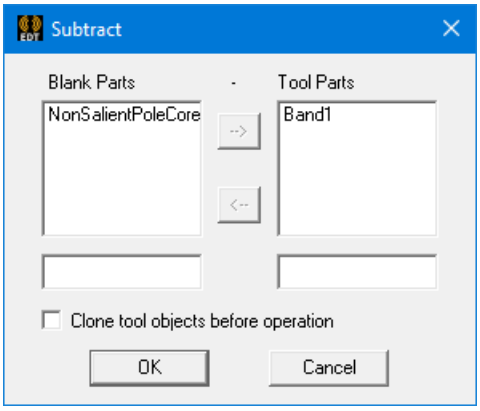


Figure 9-4 Set Blank Parts and Tool Parts

The final Non-Salient Pole Core is shown in the coordinate system window in the following figure:

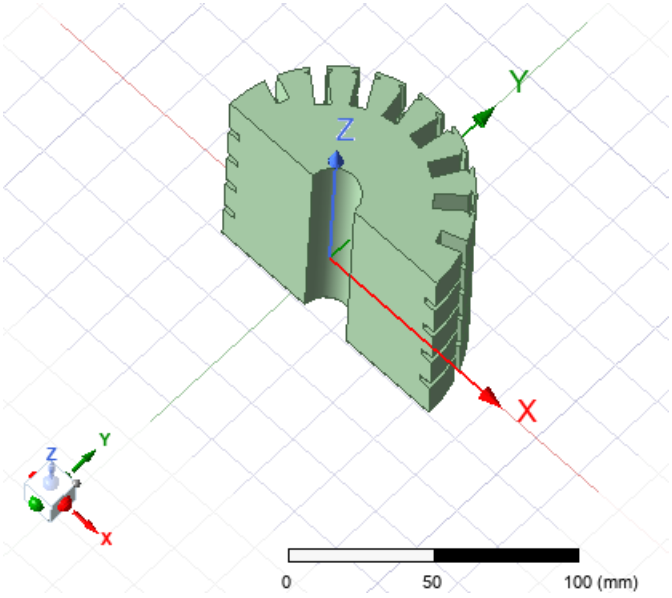


Figure 9-5 A Non-Salient Pole Core after Subtraction

Creating 3D Independent and Dependent Boundaries

There are two ways to create 3D independent and dependent boundaries:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Create a new band and change the entire band to an independent or dependent boundary
- Select one of the planar faces of an existing band in the coordinate window and assign the face to an independent or dependent boundary

The process of changing an entire band to an independent or dependent boundary is similar to creating a tool. When creating a new band or after creation of a band, you can assign the **InfoCore** property to 2 (for independent) or 3 (for dependent). After clicking **OK**, the band is changed to a face.

The process of assigning independent and dependent boundaries on an existing face follows:

1. Click **CreateUserDefinedPart** in the history tree under the band for which you are assigning boundaries. Set the **Fractions** property of the band to an integer larger than 1 to make the band have a planar face.
2. Click **Edit** > **Selection Mode** > **Faces** or right-click anywhere in the coordinate window to choose **Selection Mode** > **Faces**.
3. Select a planar face of the band by right-clicking the face. Then select **Assign Boundary** > **Matching** > **Independent** (or **Assign Boundary** > **Matching** > **Dependent** after the Independent boundary has been set up). When assigning an independent boundary, an **Independent Boundary** window appears. Otherwise a **Dependent Boundary** window appears.
4. Build up a tiny coordinate system of the face by setting the U vector and V vector to indicate the direction of the face. The U vector needs to be set using an arrow on the coordinate, and then the V vector is set automatically on the face (you can choose **Reverse Direction** of default V vector). Note that the U vectors and V vectors of a related independent and dependent must have the same orientation.

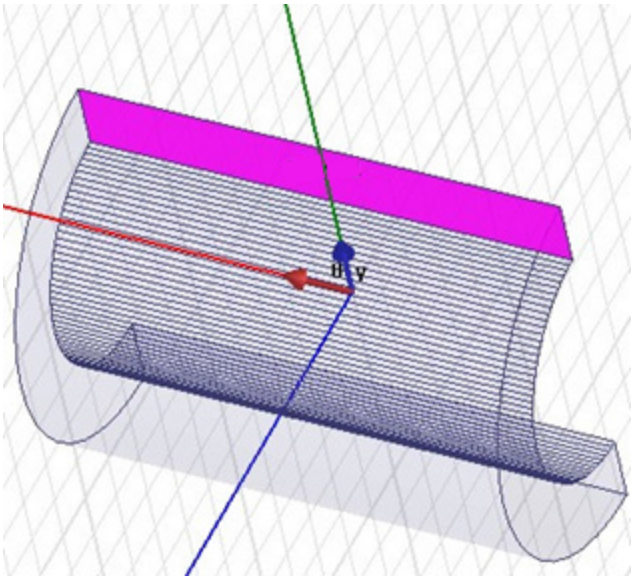


Figure 9-6 Set Up U Vector and V Vector

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. When creating a dependent boundary, you also need to set its independent boundary and its magnetic field strength (either Hm or –Hm).
6. Click **OK** to finish independent or dependent boundary creation.

Creating Core Regions

Most core UDPs, which are covered in UDPs for Slot Cores and Coils, UDPs for Pole Cores and Coils, and UDPs for Other Cores and Coils, can be changed to core regions. The ways to change them into regions are similar. This section gives a general introduction of how to change those core UDPs into core regions.

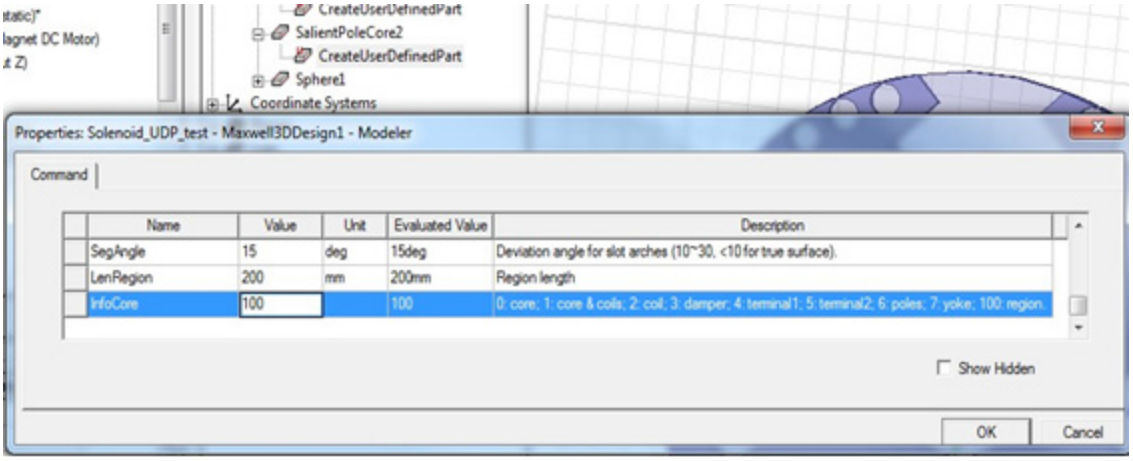
Inner and outer cores are defined as follows:

- An inner core is a core for which the **DiaGap** value is greater than the **DiaYoke** value.
- An outer core is a core for which the **DiaGap** value is less than the **DiaYoke** value.

Creating an Inner Core Region

Because the properties and parameters of an inner core region should be the same as the corresponding inner core, an inner core region is usually created by replicating an inner core and changing the new object's **InfoCore** property to **100**. The detailed steps are as follows:

1. Select the inner core in the history tree. Then right-click to choose **Edit** > **Copy**.
2. Right-click the same object in the history tree to choose **Edit** > **Paste**.
3. Click **CreateUserDefinedPart**, and set **InfoCore** property to **100** in the pop-up window.



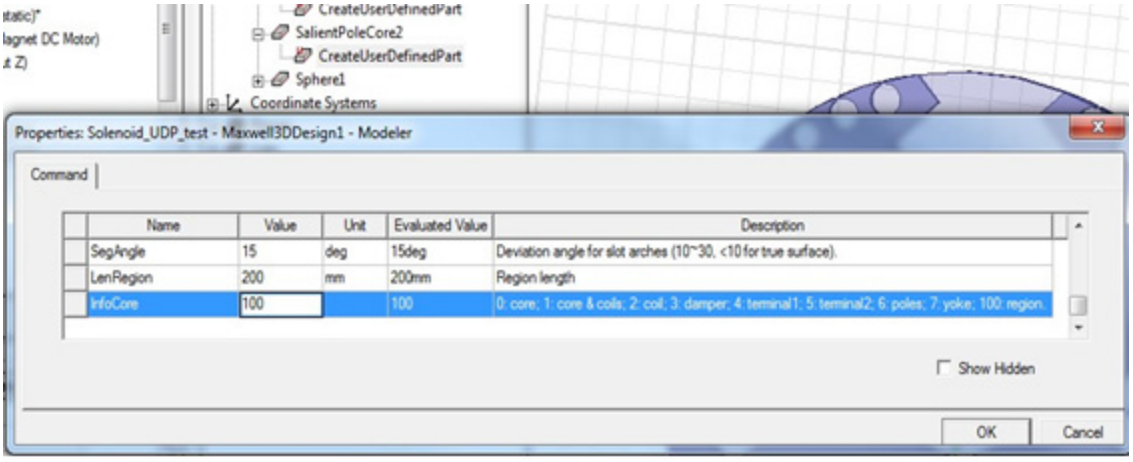
4. Click **OK.**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating an Outer Core Region

Because the properties and parameters of an outer core region should be the same as the corresponding outer core, an outer core region is usually created by replicating an outer core and changing the new object's **InfoCore** property to **100**. The detailed steps are as follows:

1. Select the outer core in the history tree. Then right-click to choose **Edit** > **Copy**.
2. Right-click the same object in the history tree to choose **Edit** > **Paste**.
3. Click **CreateUserDefinedPart**, and set **InfoCore** property to **100** in the pop-up window.



4. Click **OK.**

UDPs for Slot Cores and Coils

UDPs for defining slot cores and coils include:

- SlotCore UDP
- VentSlotCore UDP
- RacetrackSlotCore UDP
- NonSalientPoleCore UDP
- LapCoil UDP
- ConCoil UDP
- WaveCoil UDP
- SquirrelCage UDP
- DoubleCage UDP
- CupCoil UDP
- HairpinCoil UDP

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SlotCore UDP

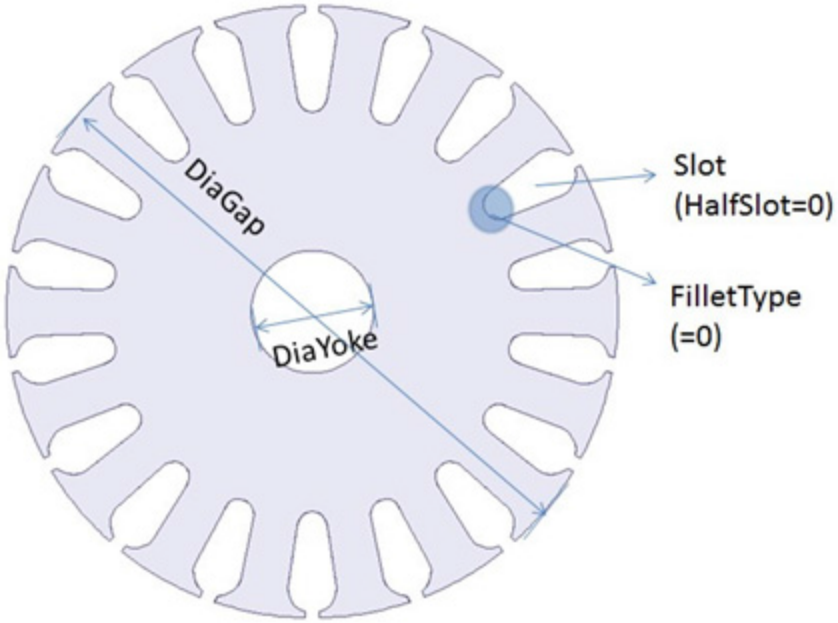
The SlotCore UDP is used to create a stator or rotor core with a distributed AC winding, or a DC commutating winding, or a squirrel-cage winding for the following machine types:

- Single-phase or multi-phase induction machines
- Wound-field or PM synchronous machines
- Brush-commutating machines, such as wound-field or PM DC machines, as well as universal motors
- Brushless DC machines, and others

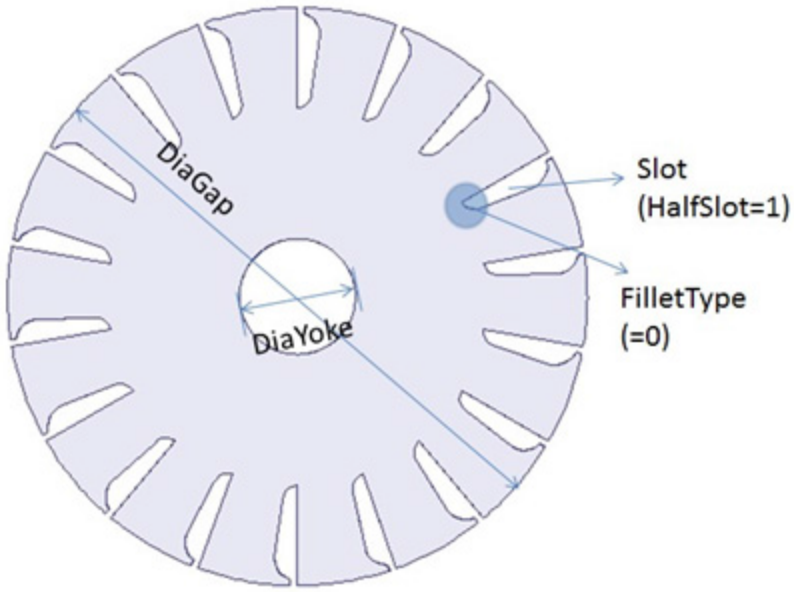
Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaYoke for outer cores.
DiaYoke	Core diameter on yoke side, DiaYoke < DiaGap for inner cores.
Length	Core length
Skew	Skew angle in core length range
Slots	Number of slots
SlotType	Slot Type: 1 to 6
Hs0	Slot opening height
Hs01	Slot closed bridge height
Hs1	Wedge height
Hs2	Slot body height
Bs0	Slot opening width
Bs1	Slot wedge maximum width
Bs2	Slot body bottom width, 0 for parallel teeth
Rs	Slot body bottom fillet
FilletType	0: a quarter circle; 1: a tangent connection; 2&3: arc bottom; 4&5: V bottom.
HalfSlot	0 for symmetric slot; 1 for half slot
SegAngle	Deviation angle for slot arches (10~30, <10 for true surface)
LenRegion	Region length
InfoCore	0: Core; 100: region

The following 2D SlotCore examples demonstrate how these parameters are used. The first example shows the diameters of SlotCore when **HalfSlot** is 0:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

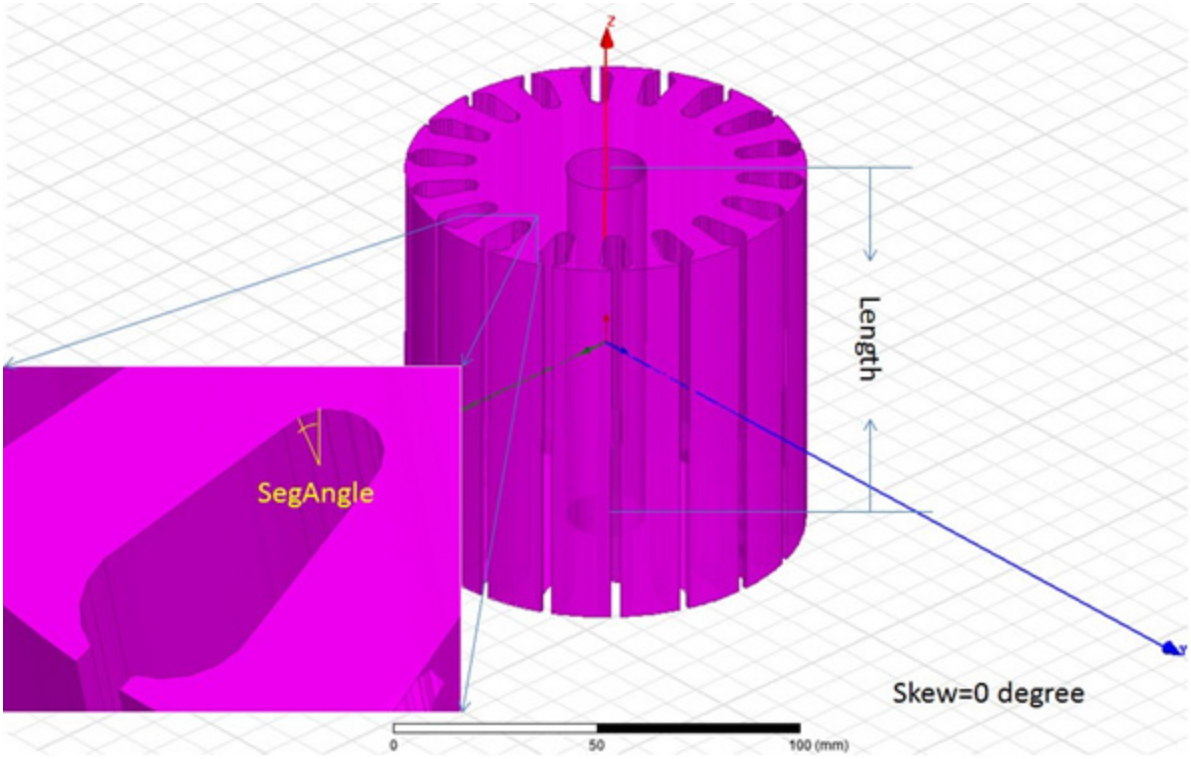


Diameters of SlotCore when **HalfSlot** is 1:

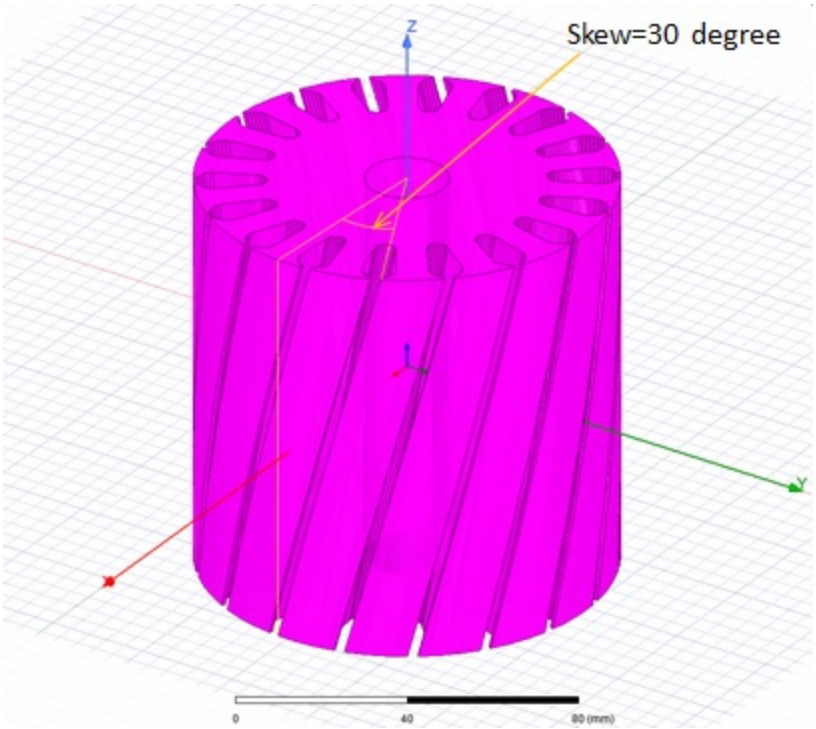


3D parameters of a slot core are shown in the following examples. The first figure highlights the **SegAngle** and **Length** of SlotCore:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



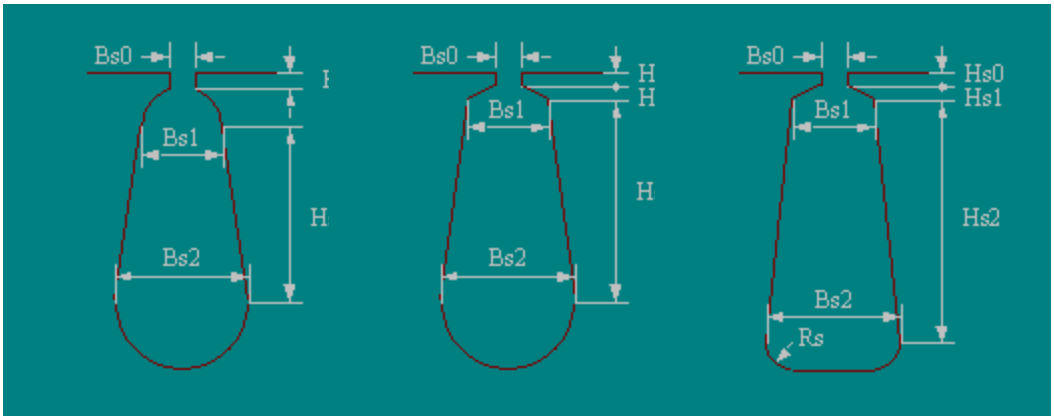
The **Skew** of SlotCore is highlighted in the next figure:



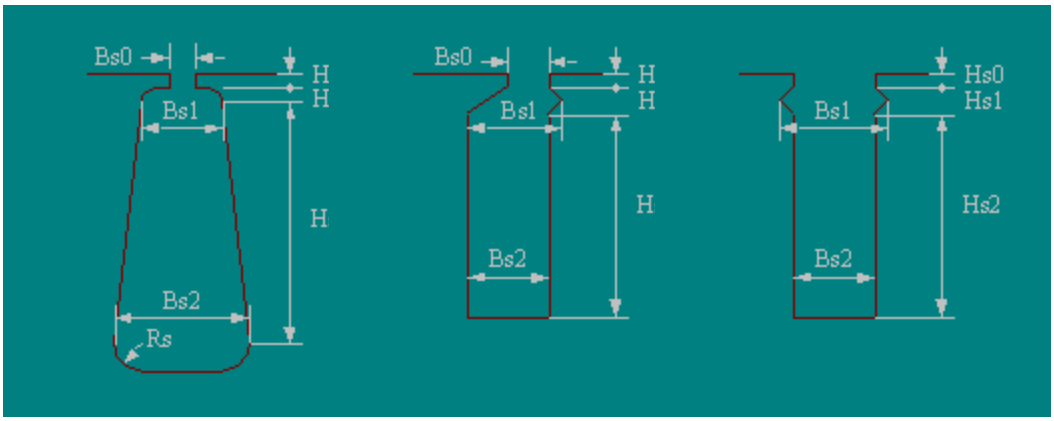
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Six common slot types and their related parameters are shown in the following figures:

Slot Type 1 **Slot Type 2** **Slot Type 3**



Slot Type 4 **Slot Type 5** **Slot Type 6**



Slot cores or coils with a **SlotType** parameter can choose one of these slots.

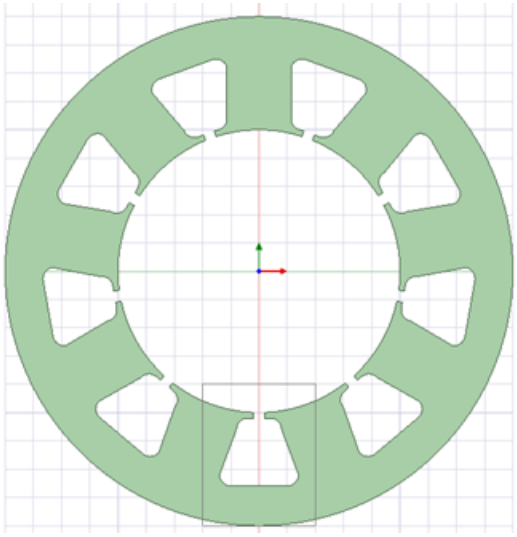
Creating an Inner or Outer Slot Core

Set the value of **DiaYoke** and **DiaGap** as **DiaYoke** < **DiaGap** to create inner cores or **DiaGap** < **DiaYoke** for outer cores. You can set the values either when creating the core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Set the **InfoCore** value to 0.

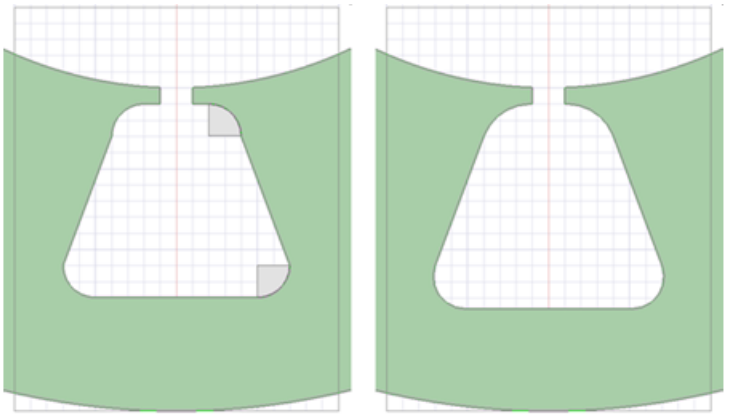
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Slots with Different Fillet and Bottom Types



Slot Core with Type-4 Slots

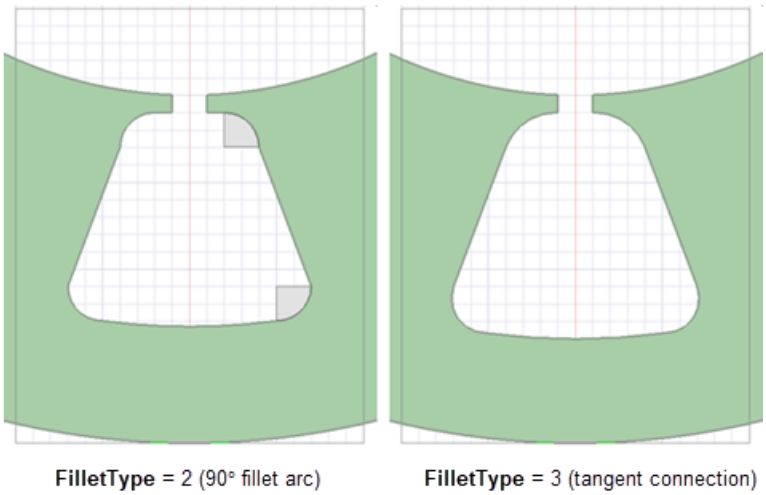
Various values of **FilletType** are used for different fillet and bottom types. **FilletType** values 0 and 1 are for half-circle slot bottom for **SlotType** = 1 and 2, or for flat slot bottom for **SlotType** = 3 and 4, as shown below.



Slot Type 4 with Flat Bottom

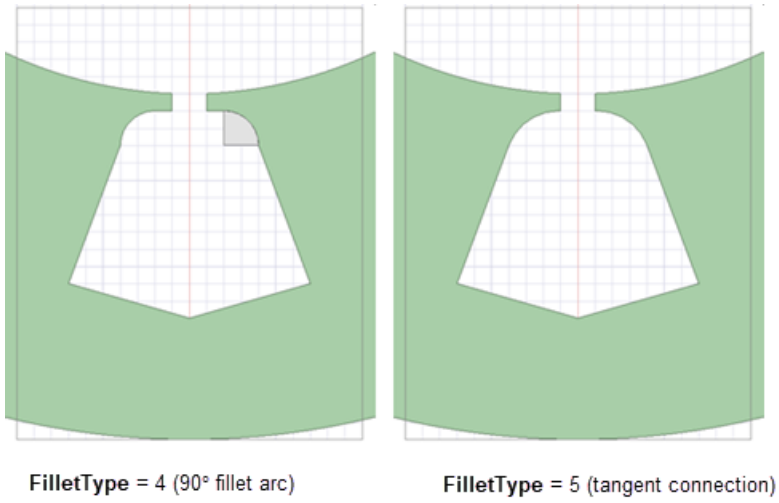
FilletType values 2 and 3 are for outer-core arc slot bottom for **SlotType** = 3 and 4 only, as shown below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Slot Type 4 with Arc Bottom

FilletType values 4 and 5 are for V-shape slot bottom for **SlotType** = 3 and 4 only, as shown below.



Slot Type 4 with V-Shape Bottom (Rs is V-shape height)

VentSlotCore UDP

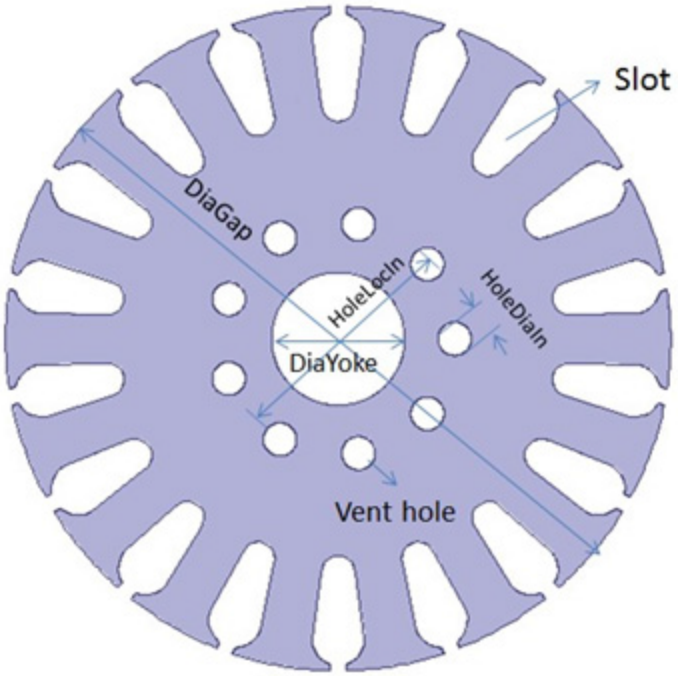
The VentSlotCore UDP can create the same core that the SlotCore UDP does. However, the VentSlotCore UDP can create cores with radial vents and axial holes. If you want to create a slotted core without radial vents or axial holes, use the SlotCore UDP to have less data input.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

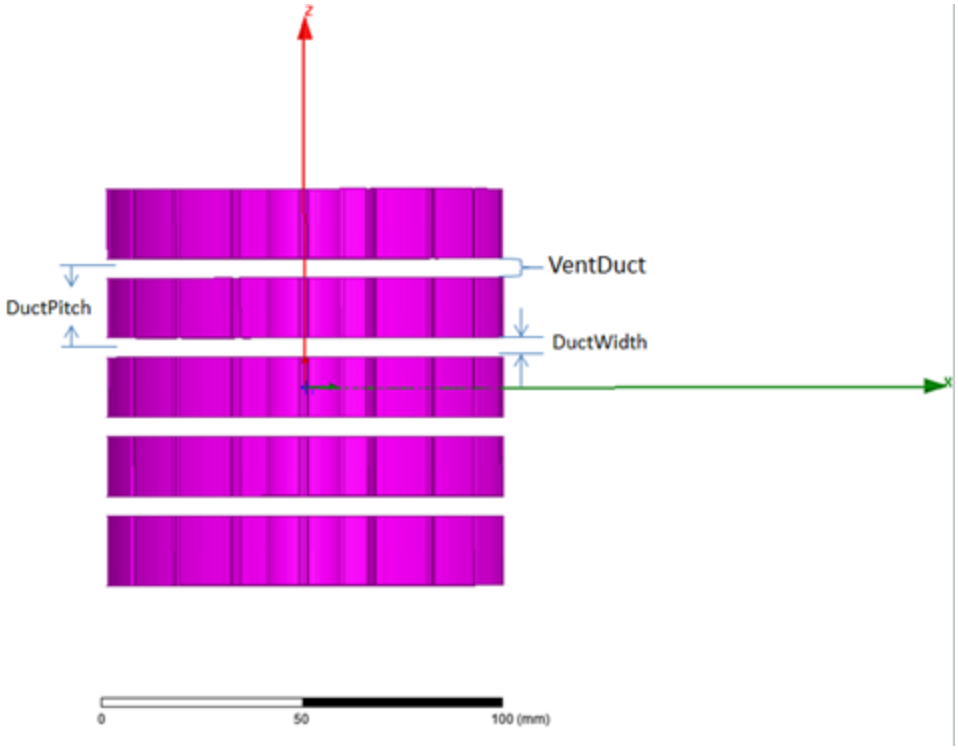
Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaYoke for outer cores
DiaYoke	Core diameter on yoke side, DiaYoke < DiaGap for inner cores
Length	Core length
Skew	Skew angle in core length range
Slots	Number of slots
SlotType	Slot Type: 1 to 6
Hs0	Slot opening height
Hs01	Slot closed bridge height
Hs1	Wedge height
Hs2	Slot body height
Bs0	Slot opening width
Bs1	Slot wedge maximum width
Bs2	Slot body bottom width, 0 for parallel teeth
Rs	Slot body bottom fillet
FilletType	0: a quarter circle; 1: a tangent connection; 2&3: arc bottom.
HalfSlot	0 for symmetric slot; 1 for half slot
VentHoles	Number of axial vent holes per row
HoleDiaIn	Diameter of inner vent holes
HoleDiaOut	Diameter of outer vent holes
HoleLocIn	Diameter of inner vent hole center layout circle
HoleLocOut	Diameter of outer vent hole center layout circle
VentDucts	Number of radial vent ducts
DuctWidth	Axial width of radial vent ducts
DuctPitch	Center-to-center distance between two adjacent ducts
SegAngle	Deviation angle for slot arches (10~30, <10 for true surface)
LenRegion	Region length
InfoCore	0: Core; 100: region

These parameters are used in the following inner VentSlotCore example. The first figure shows the parameters on the cross-section of VentSlotCore:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The next figure shows the parameters on axial direction of VentSlotCore:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** For parameters such as **Skew**, **FilletType**, **SegAngle**, **HalfSlot** and parameters related to slots, refer to SlotCore UDP parameters.

Creating an Inner or Outer Vent Slot Core

Set the value of **DiaYoke** and **DiaGap** as **DiaYoke** < **DiaGap** to create inner cores or **DiaGap** < **DiaYoke** for outer cores. You can set the values when creating the core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Set the **InfoCore** value to 0.

RacetrackSlotCore UDP

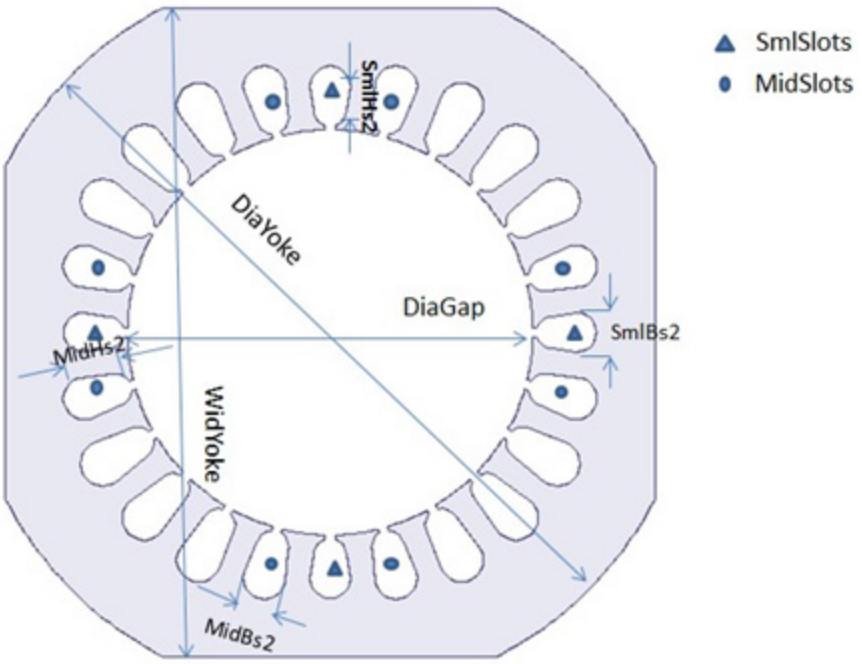
The RacetrackSlotCore UDP is used to create a stator core with a single-phase concentric AC winding for single-phase induction machines. The stator may have up to three different slot sizes.

Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaYoke for outer cores
DiaYoke	Core diameter on yoke side, DiaYoke < DiaGap for inner cores
Length	Core length
Skew	Skew angle in core length range
Slots	Number of slots
SlotType	Slot Type: 1 to 6
Hs0	Slot opening height
Hs01	Slot closed bridge height.
Hs1	Wedge height
Hs2	Slot body height
Bs0	Slot opening width
Bs1	Slot wedge maximum width
Bs2	Slot body bottom width, 0 for parallel teeth
Rs	Slot body bottom fillet
FilletType	0: a quarter circle; 1: a tangent connection; 2&3: arc bottom.
Poles	Number of poles
WidYoke	Racetrack width for 2/4-pole cores
MidSlots	Number of middle-size slots for 2/4-pole cores

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

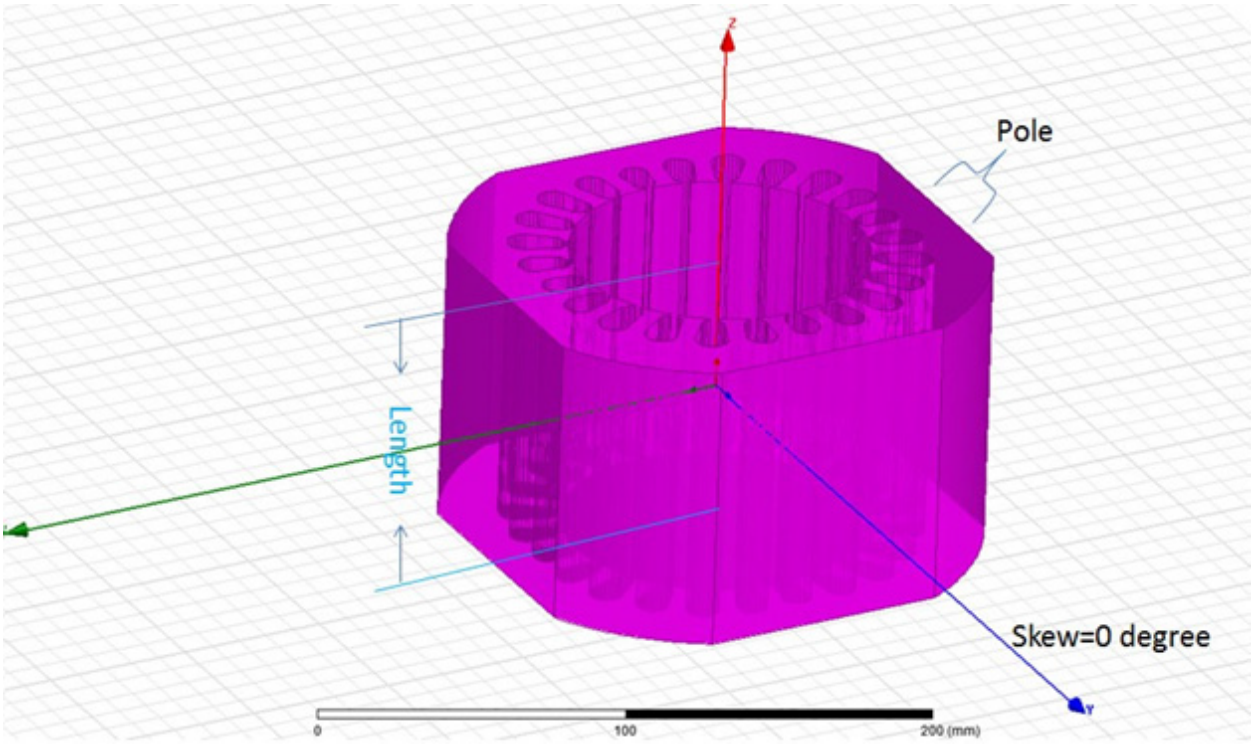
Property	Description
MidHs2	Middle-size slot body height for 2/4-pole cores
MidBs2	Middle-size slot body bottom width for 2/4-pole cores
SmlSlots	Number of small-size slots for 2/4-pole cores
SmlHs2	Small-size slot body height for 2/4 pole cores
SmlBs2	Small-size slot body bottom width for 2/4-pole cores
SegAngle	Deviation angle for slot arches (10~30, <10 for true surface)
LenRegion	Region length
InfoCore	0: Core; 100: region

These parameters are demonstrated in the following Racetrack SlotCore example. The first figure shows the parameters on cross-section of RacetrackSlotCore:



The next figure shows the parameters on axial direction of RacetrackSlotCore:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:** For parameters such as **Skew**, **FilletType**, **SegAngle** and parameters related to slots, refer to **SlotCore UDP parameters.**

Creating an Outer Racetrack Slot Core

Generally, Racetrack Slot Cores are used as outer cores. Set the value of **DiaYoke** and **DiaGap** as **DiaGap** < **DiaYoke** for outer cores. You can set the values when creating the core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Set the **InfoCore** value to 0.

NonSalientPoleCore UDP

The NonSalientPoleCore UDP is used to create non-salient rotor cores for non-salient rotor synchronous machines. This core type may include surface tangential vent ducts, axial vent ducts, as well as a damper.

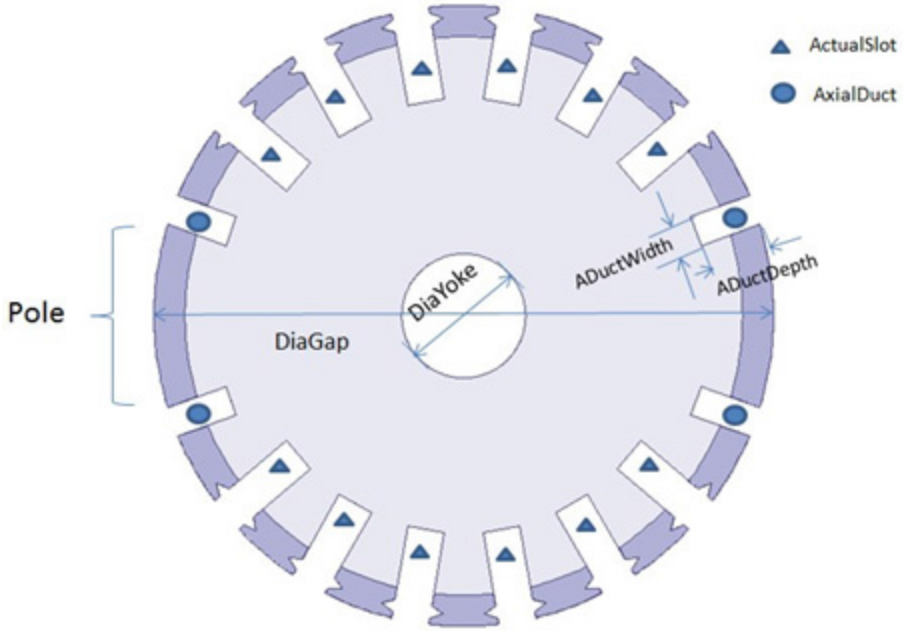
Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaYoke for outer cores.
DiaYoke	Core diameter on yoke side, DiaYoke < DiaGap for inner

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

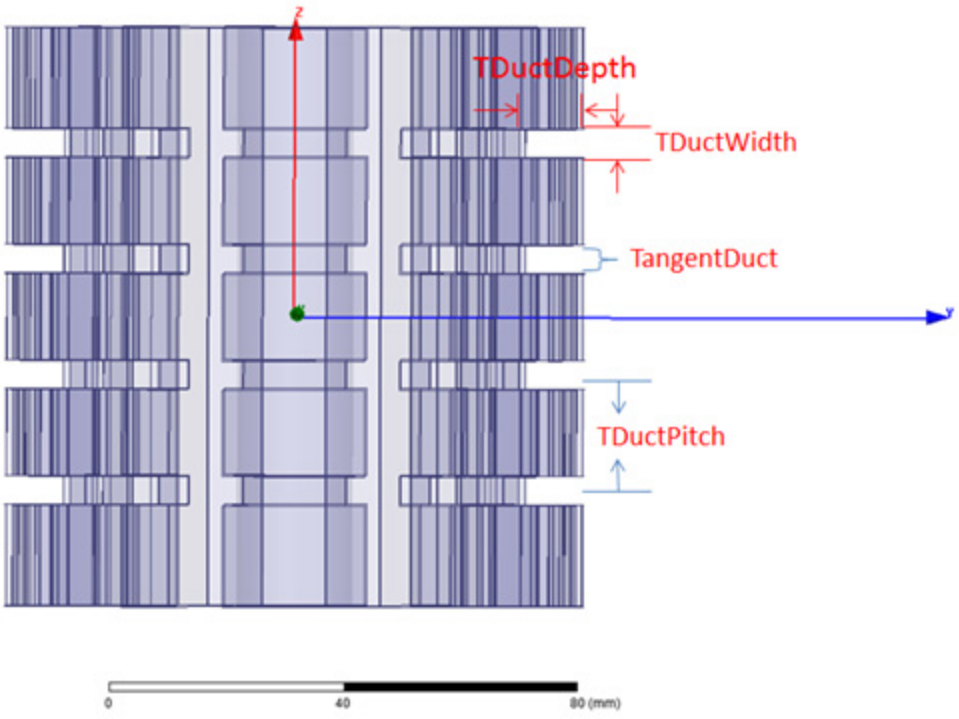
	cores.
Length	Core length.
Skew	Skew angle in core length range.
IndexingSlots	Number of indexing slots for slot pitch calculation.
ActualSlots	Number of actual slots: ActualSlots <= IndexingSlots.
SlotType	Slot Type: 1 to 7.
Hs0	Slot opening height.
Hs1	Slot wedge height.
Hs2	Slot body height.
Bs0	Slot opening width.
Bs1	Slot wedge maximum width.
Bs2	Slot body bottom width, 0 for parallel teeth.
Rs	Slot body bottom fillet.
FilletType	0: a quarter circle; 1: a tangent connection; 2&3: arc bottom.
Poles	Number of poles.
TangentDucts	Number of surface tangential vent ducts.
TDuctWidth	Axial width of surface tangential vent ducts (or damper rings).
TDuctDepth	Radial depth of surface tangential vent ducts (or damper rings).
TDuctPitch	Pitch of surface tangential vent ducts (or damper bar extension).
AxialDucts	Number of axial vent ducts per pole.
ADuctWidth	Width of axial vent ducts in main tooth.
ADuctDepth	Depth of axial vent ducts in main tooth.
SegAngle	Deviation angle for slot arches (10~30, <10 for true surface).
LenRegion	Region length.
InfoCore	0: Core; 100: region.

These parameters are demonstrated with an inner Non-Salient SlotCore example. The first figure shows the parameters on a cross-section of NonSalientPoleCore:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The next figure shows the parameters on the axial direction of NonSalientPoleCore:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** For parameters such as **Skew**, **FilletType**, **SegAngle** and parameters related to slots, refer to SlotCore UDP parameters.

Creating an Inner Non-Salient Pole Core

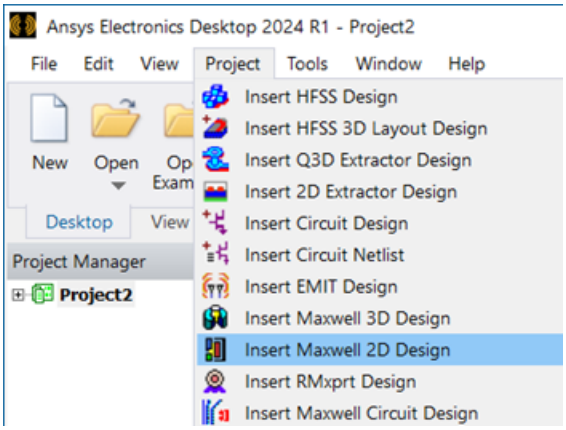
Generally, Non-Salient Pole Cores are used as inner cores. Set the value of **DiaYoke** and **DiaGap** as **DiaGap** > **DiaYoke** for inner cores. You can set the values when creating the core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Set the **InfoCore** value to 0.

Creating a Non-Salient Pole Core with a Q-Axis Damper

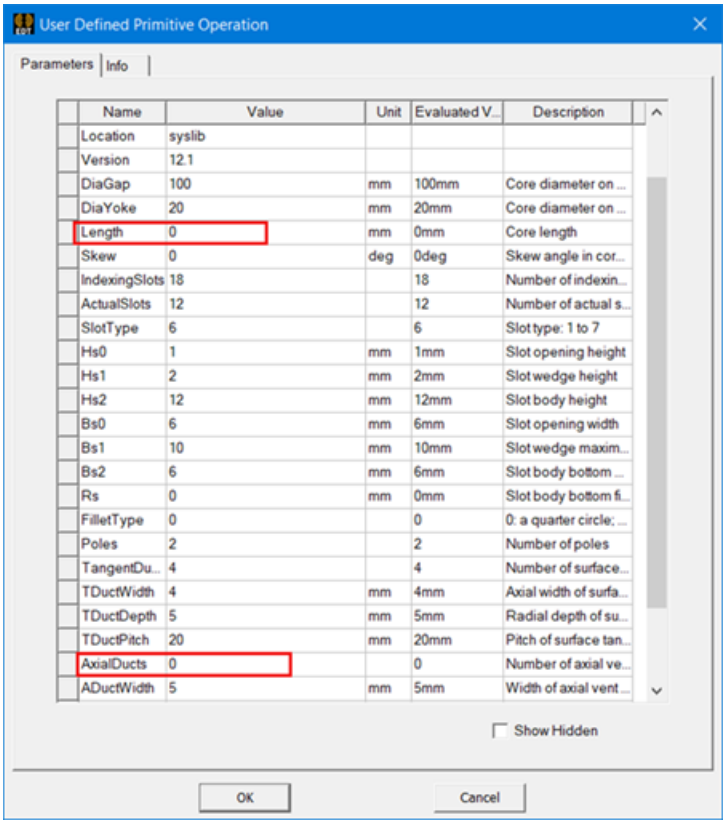
In general, this process is realized by a subtraction of two objects. Several steps are needed to finish the procedure.

1. Click **Project** > **Insert Maxwell 2D Design** to create a Maxwell 2D Design in a project.



2. Click **Draw** > **User Defined Primitive** > **RMxprt** > **NonSalientPoleCore** to draw a Non-Salient Pole Core UDP. In order to show it in a 2D dimension, set the **Length** to 0. Also set **AxialDucts** to 0 for subtraction, as shown in the following figure.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Create a damper: Right click on the NonSalientPoleCore in the model history tree and select **Copy**, and then paste it in the model history tree and rename it to damper.

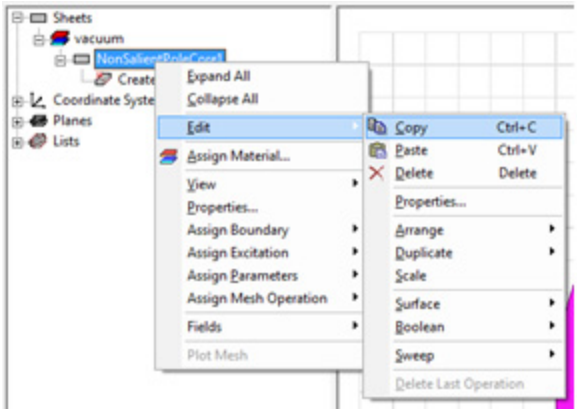
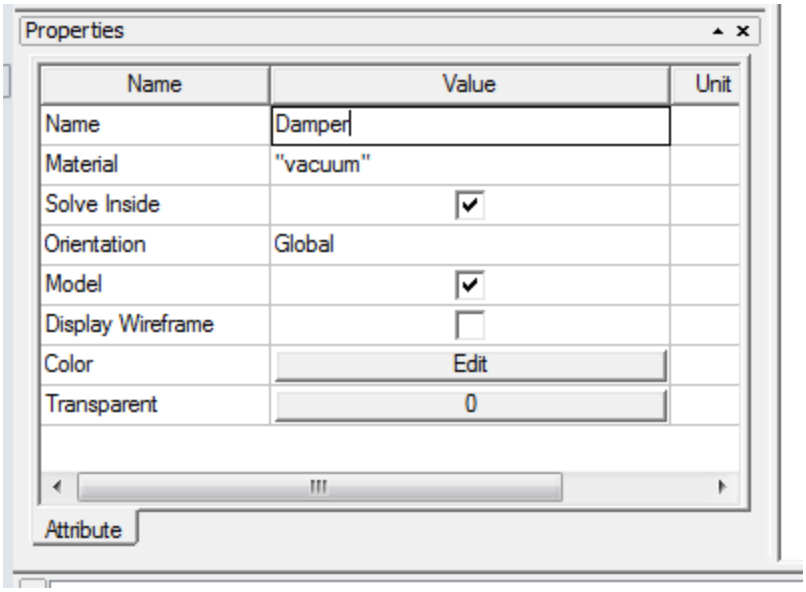
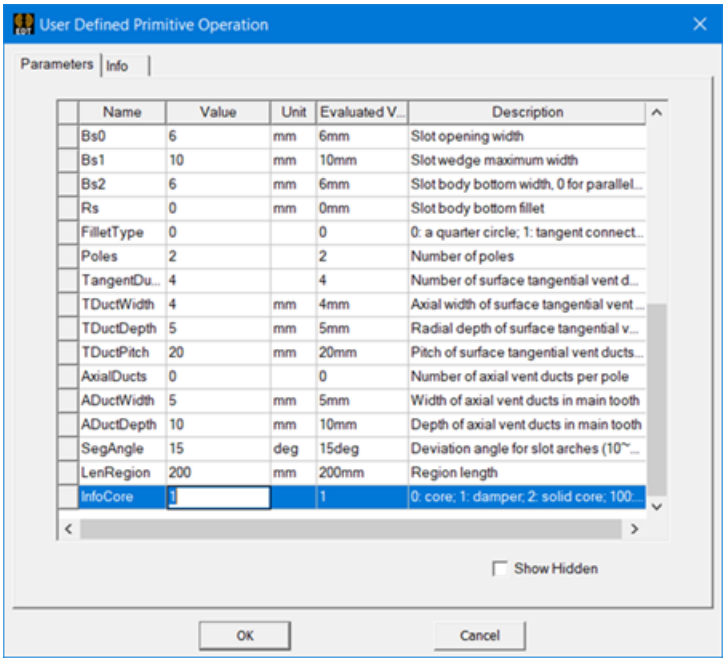


Figure 9-7 Rename the new NonSalientPoleCore to *Damper*:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

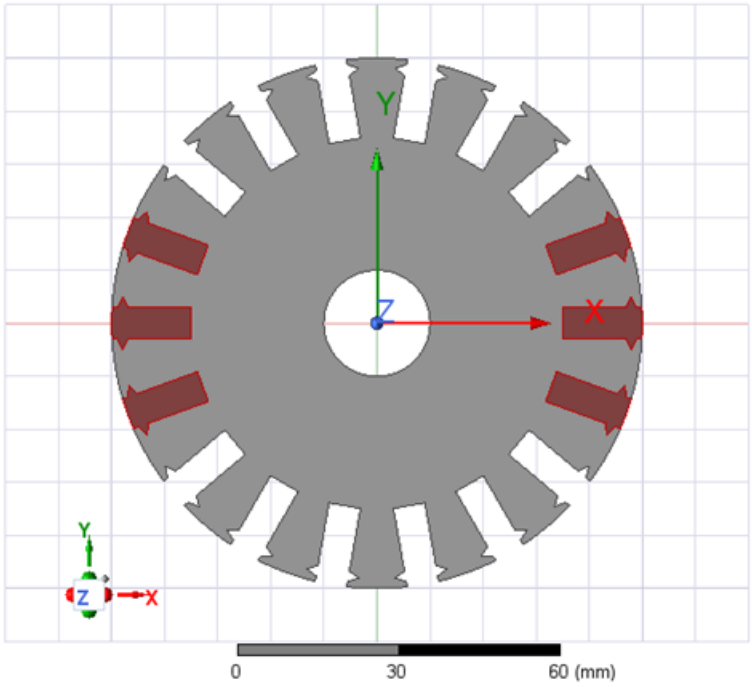


4. Change the **InfoCore** value to **1** to make a damper. Also change the **Color** of the damper to distinguish it from the NonSalientPoleCore.

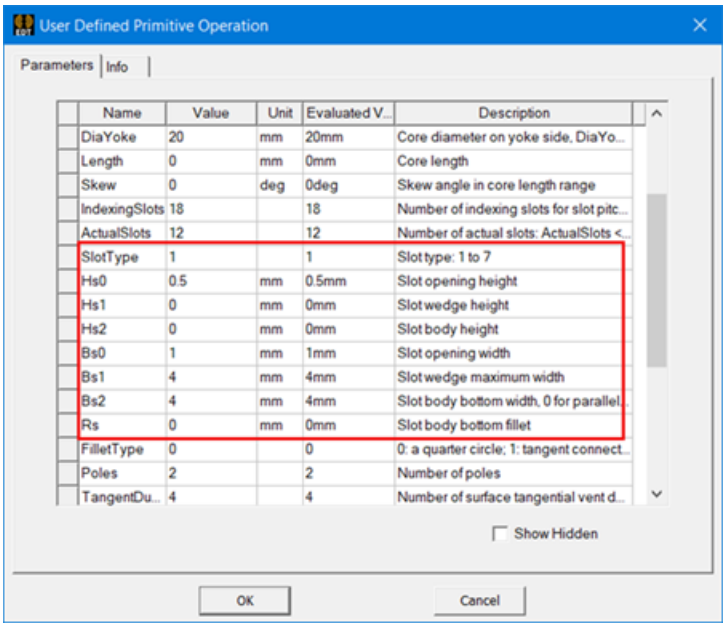


The objects appear as follows in the coordinate system window. The Non-Salient Pole Core is gray, and the damper is red.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

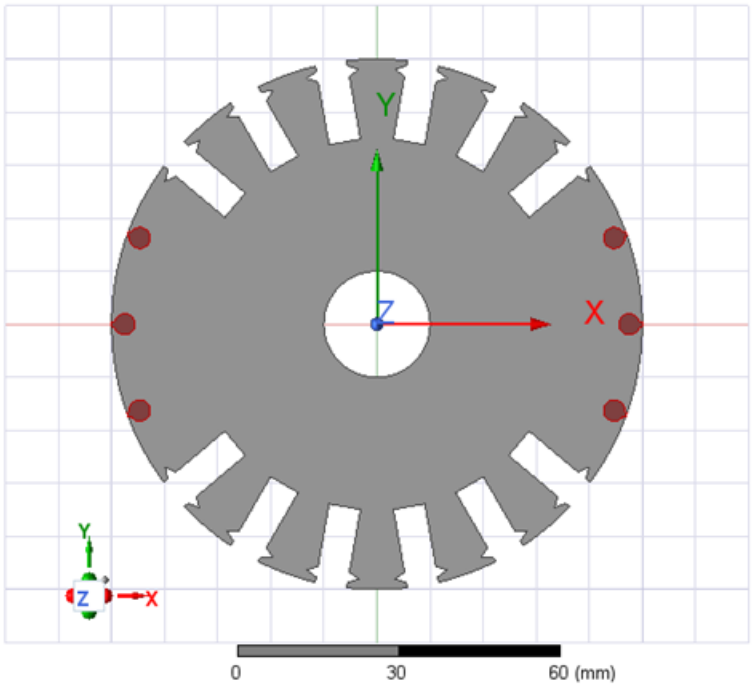


5. You can also change the damper's **SlotType** and slot parameters if necessary. For instance, **SlotType** is reset to **1** and slot parameters are reassigned as in the following figure.



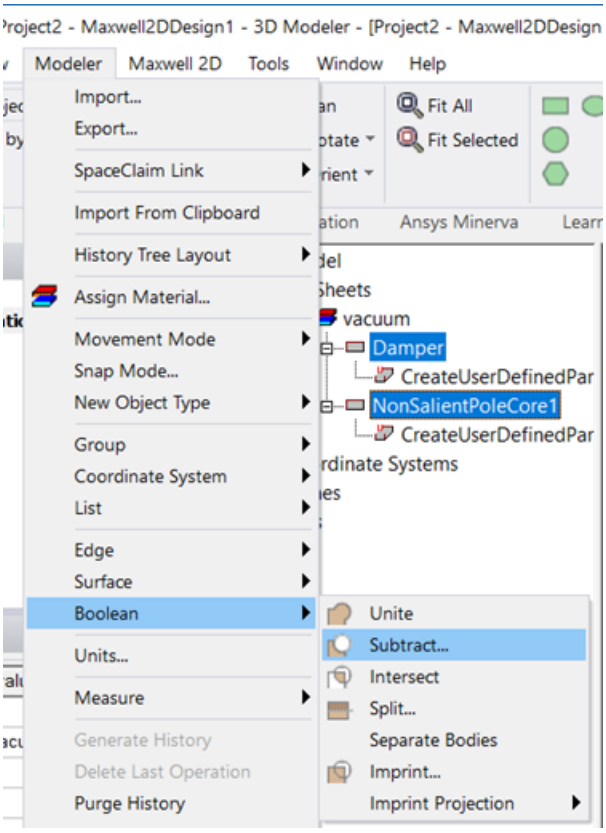
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After you modify the slot-related parameters of the damper, the objects in the coordinate system window appear as in the following example:

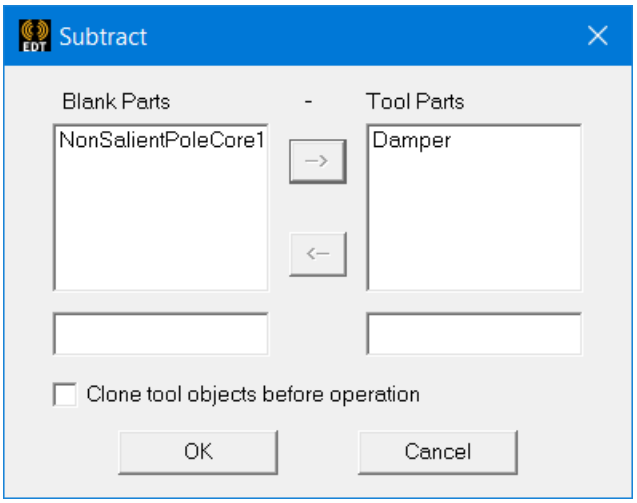


6. Select both objects (Damper and NonSalientPoleCore) in the history tree simultaneously by pressing **Ctrl** on the keyboard. Click **Modeler** > **Boolean** > **Subtract**. A Subtract window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

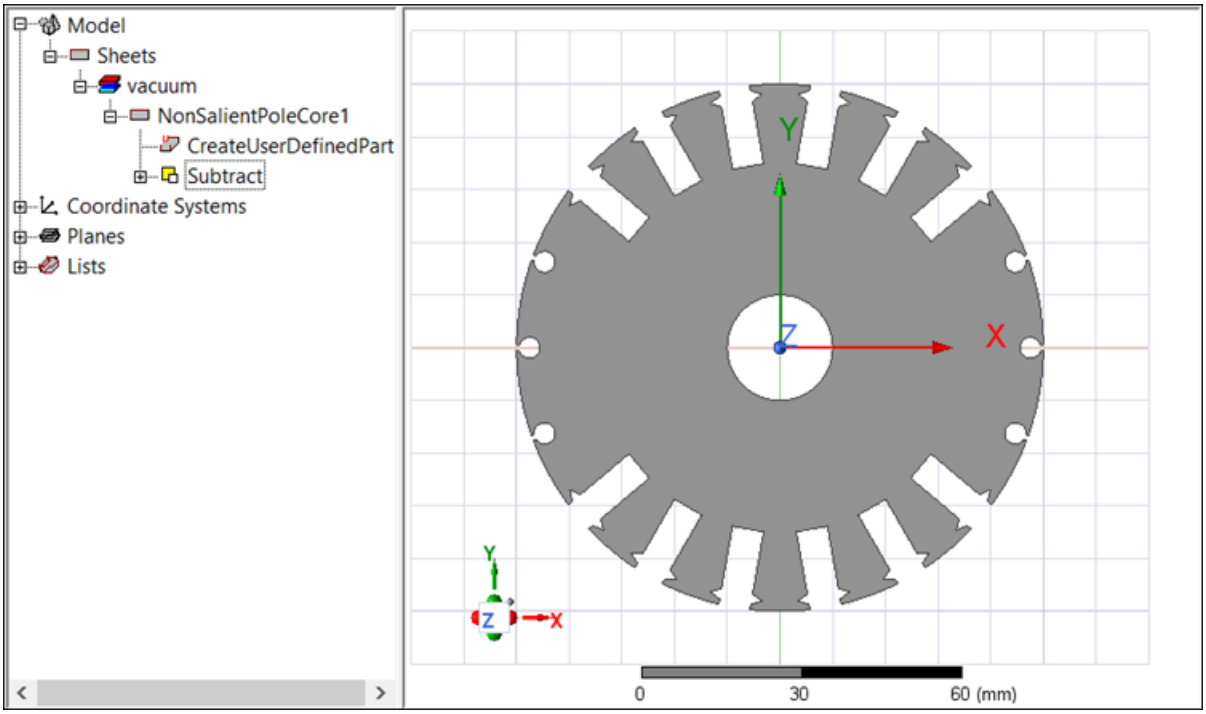


7. Move NonSalientPoleCore to **Blank Parts** and Damper to **Tool Parts**, and check the **Clone tool objects before operation** box. Click **OK** to finish subtraction.



The final history tree and coordinate system window show the objects after subtraction:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

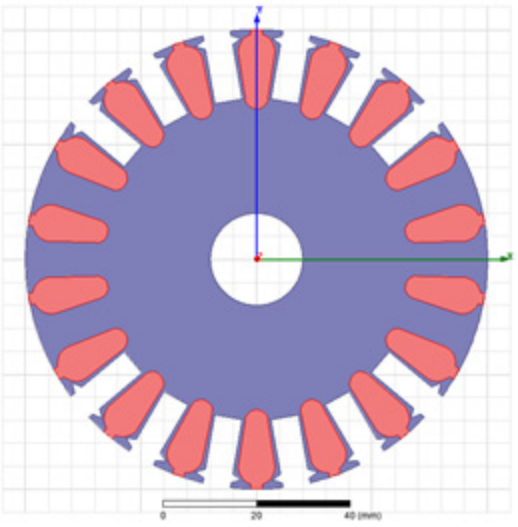


Creating a Non-Salient Pole Core with a Uniform-Distributed Damper

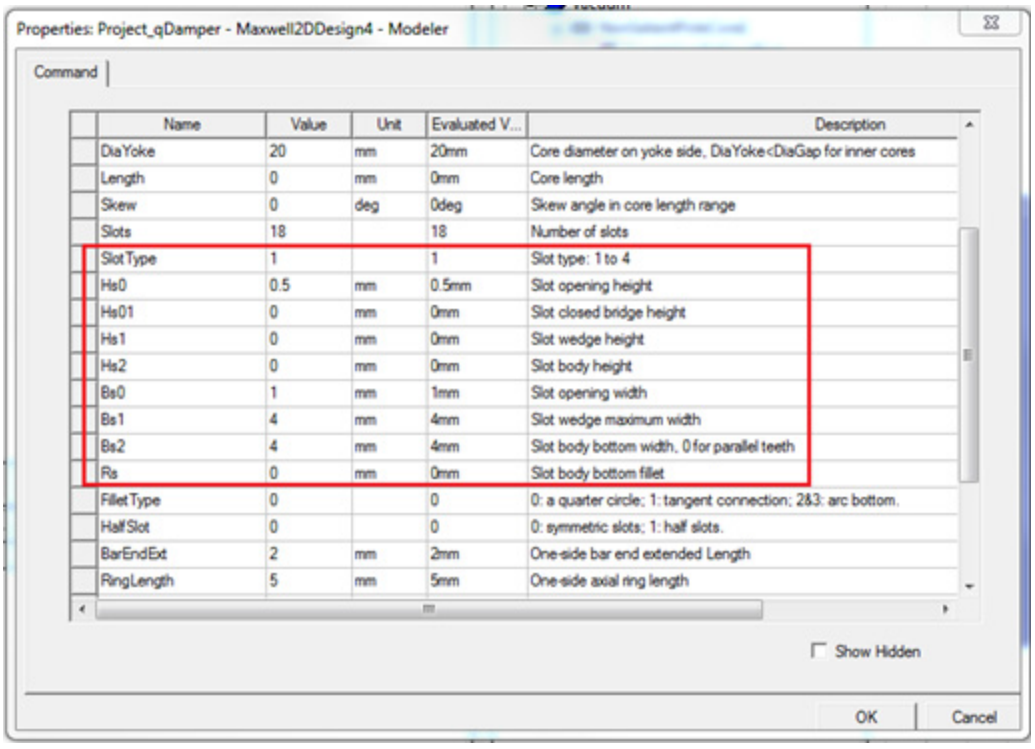
This process relies on subtraction of two objects and is very similar to Creating a Non-Salient Pole Core with a Q-Axis Damper. The major difference is that you create a squirrel cage instead of copying a NonSalientPoleCore to create a damper.

1. Click **Project** > **Insert Maxwell 2D Design** to create a Maxwell 2D Design in a project.
2. Click **Draw** > **User Defined Primitive** > **RMxprt** > **NonSalientPoleCore** to draw a Non-Salient Pole Core UDP. To show it in a 2D dimension, set the **Length** to 0. Also set **AxialDucts** to 0 for subtraction, as shown in the figure that follows.
3. Click **Draw** > **User Defined Primitive** > **RMxprt** > **SquirrelCage** to draw a squirrel cage UDP. Set the **Length** to 0, change its **Color** to a different one, and rename it to “Damper.” Then objects in the coordinate system window appear as in the following figure. The Non-Salient Pole Core is gray, and the damper is red.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

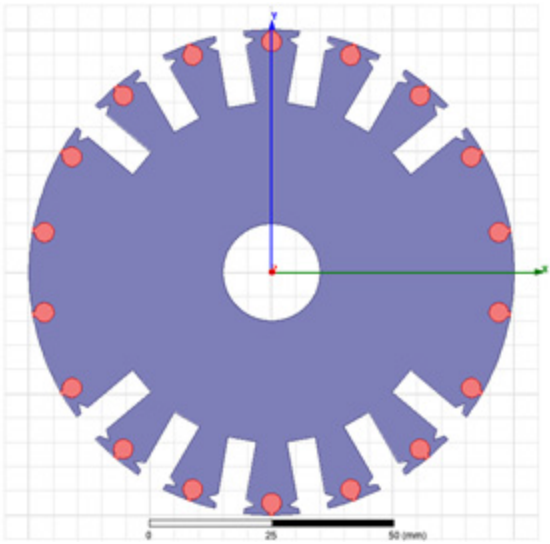


4. You can also change the damper's **SlotType** and slot parameters if necessary. For instance, **SlotType** is reset to **1** and slot parameters are reassigned as in the following figure.

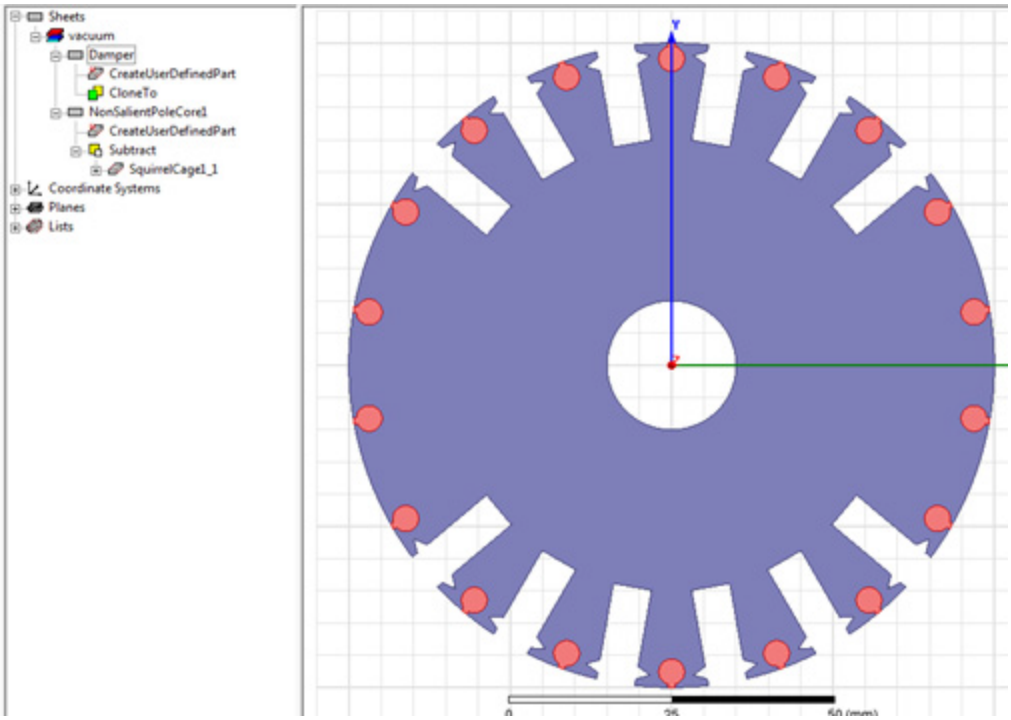


5. After you modify the parameters, the objects in the coordinate system window appear as in the following figure:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. Select both objects (Damper and NonSalientPoleCore) in the history tree simultaneously by pressing **Ctrl** on the keyboard. Click **Modeler** > **Boolean** > **Subtract**. A Subtract window appears.
7. Move NonSalientPoleCore to **Blank Parts** and Damper to **Tool Parts**, and check the **Clone tool objects before operation** box. Click **OK** to finish subtraction.
8. The final history tree and coordinate system window show the Non-Salient Pole Core with a uniform-distributed damper:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LapCoil UDP

The LapCoil UDP is used to create a coil for lap-type distributed AC windings or lap-type DC commutating windings in a slotted core created by the SlotCore or VentSlotCore UDP. This UDP can also create a coil cross-section used as a terminal for current assignment.

Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaYoke for outer cores.
DiaYoke	Core diameter on yoke side, DiaYoke < DiaGap for inner cores.
Length	Core length.
Skew	Skew angle in core length range.
Slots	Number of slots.
SlotType	Slot Type: 1 to 7.
Hs0	Slot opening height.
Hs1	Slot wedge height.
Hs2	Slot body height.
Bs0	Slot opening width.
Bs1	Slot wedge maximum width.
Bs2	Slot body bottom width, 0 for parallel teeth.
Rs	Slot body bottom fillet.
FilletType	0: a quarter circle; 1: a tangent connection; 2&3: arc bottom; 4&5: V bottom.
Layers	Number of winding layers.
CoilPitch	Coil pitch measured in slots.
EndExt	One-side end extended length.
SpanExt	Axial length of end span; 0 for no span.
BendAngle	Bending angle viewed in the rz plane.
SegAngle	Deviation angle for slot arches (5~15, <5 for true surface).
LenRegion	Region length.
InfoCoil	0: winding; 1: coil; 2 terminal1; 3: terminal2; 4:insulation; 100: region.

A single coil is much easier to be described with a slot core as reference. In the following figures, coils are in gray and slot cores are orange. The lap coil's **InfoCoil** property is set to **1** to use a single coil to show parameters of lap coil.

This first figure shows a Single Lap Coil with **Coil Pitch** set to 2:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

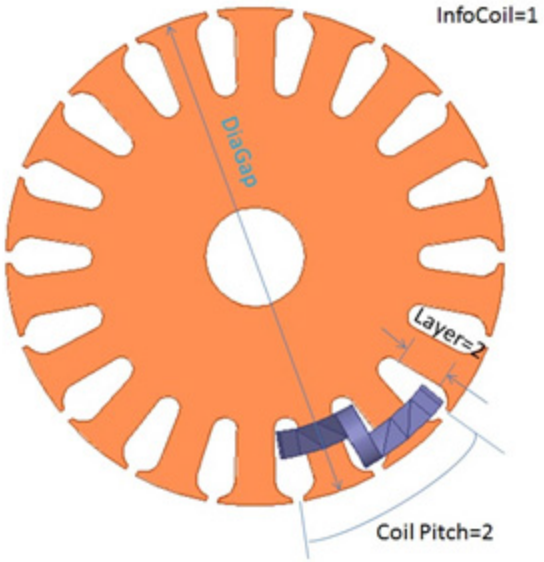


Figure 9-8 This next figure shows a Single Lap Coil with Coil Pitch set to 4:

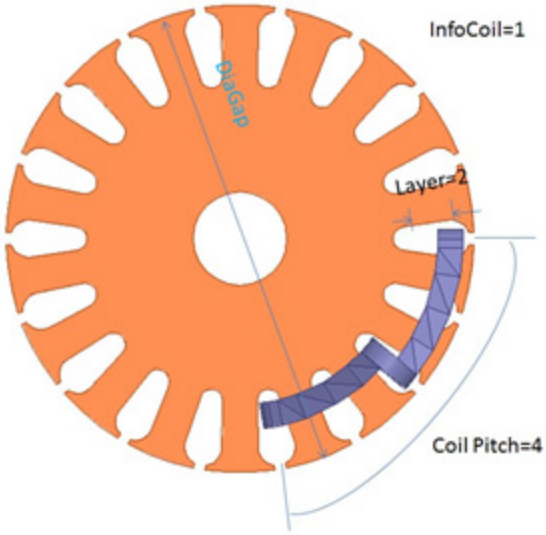


Figure 9-9 Axial Single Lap Coil with Coil Pitch set to 4:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

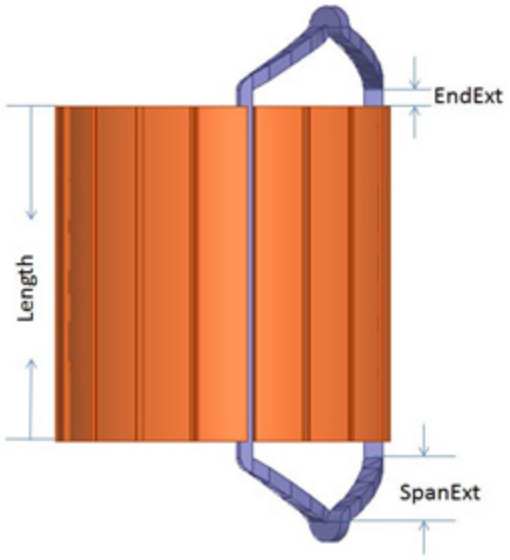


Figure 9-10

Figure 9-11 Below is an axial view of an outer core winding with bended end coil. The lap coil's InfoCoil property is set to 0.

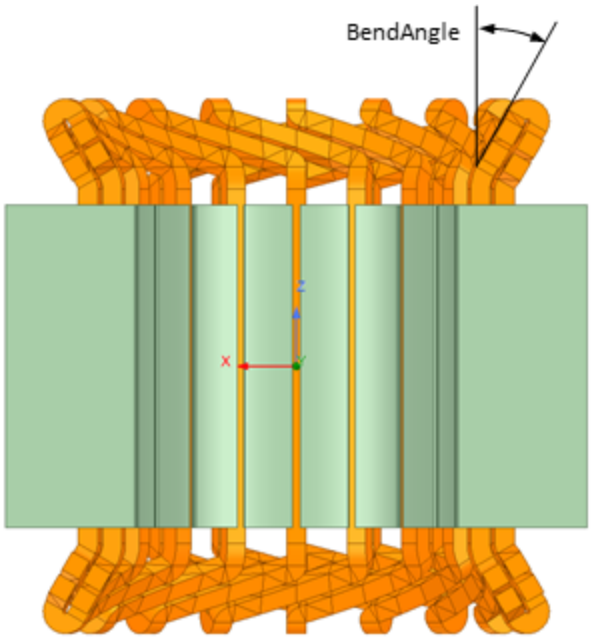


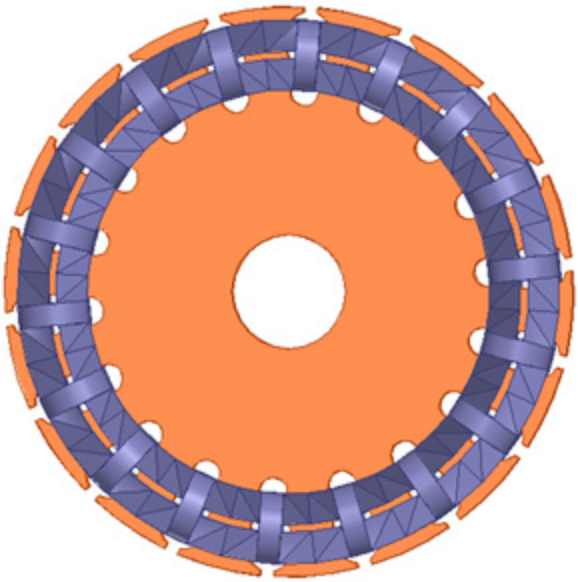
Figure 9-12

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating a Whole Lap Winding

A whole lap winding is created by default or by manually setting the property of **InfoCoil** to 0. You can assign the value of this property either when creating the lap coil or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

An example of a whole lap winding with an orange slot core is shown in the following figure:



Creating a Single Lap Coil and its Terminals

1. Create a single lap coil by manually setting the property of **InfoCoil** to 1. See the figures in "LapCoil UDP " on page 9-39 for a single lap coil with different parameters.
2. For the terminals of the lap coil, set the property of **InfoCoil** to 2 or 3 (terminal1 or terminal2). " Terminal 1 of LapCoil" on the facing page shows terminal1 in the coordinate system.

You can assign the value of this property either when creating the lap coil or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

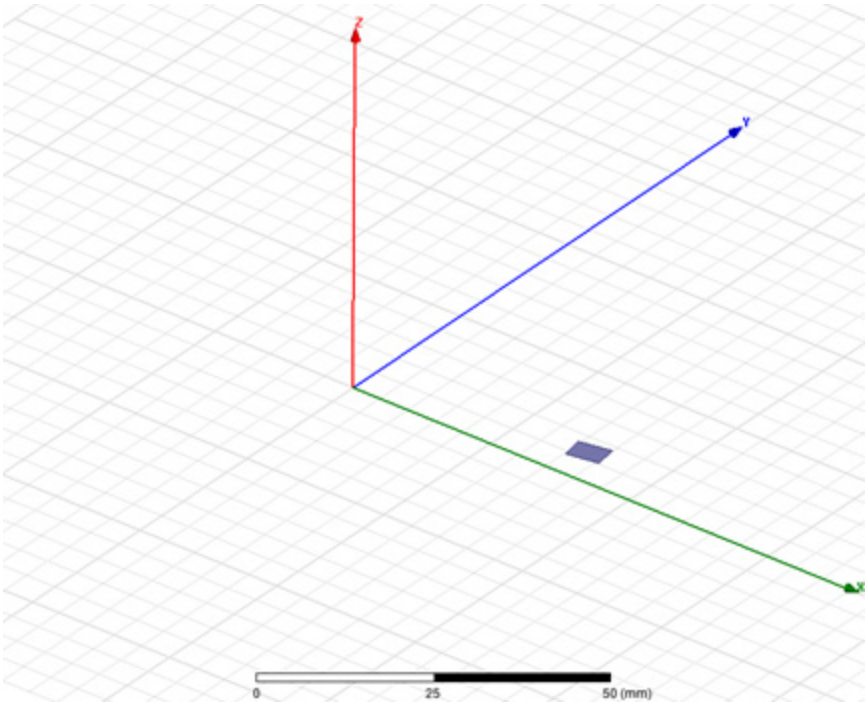


Figure 9-13 Terminal 1 of LapCoil

Creating a 3D Open Insulation

A 3D open insulation is created by manually setting the property of **InfoCoil** to 4. In the history tree, the object changes from a solid to a sheet.

You can assign the value of this property either when creating the lap coil or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

A 3D open insulation is shown in the following figure:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

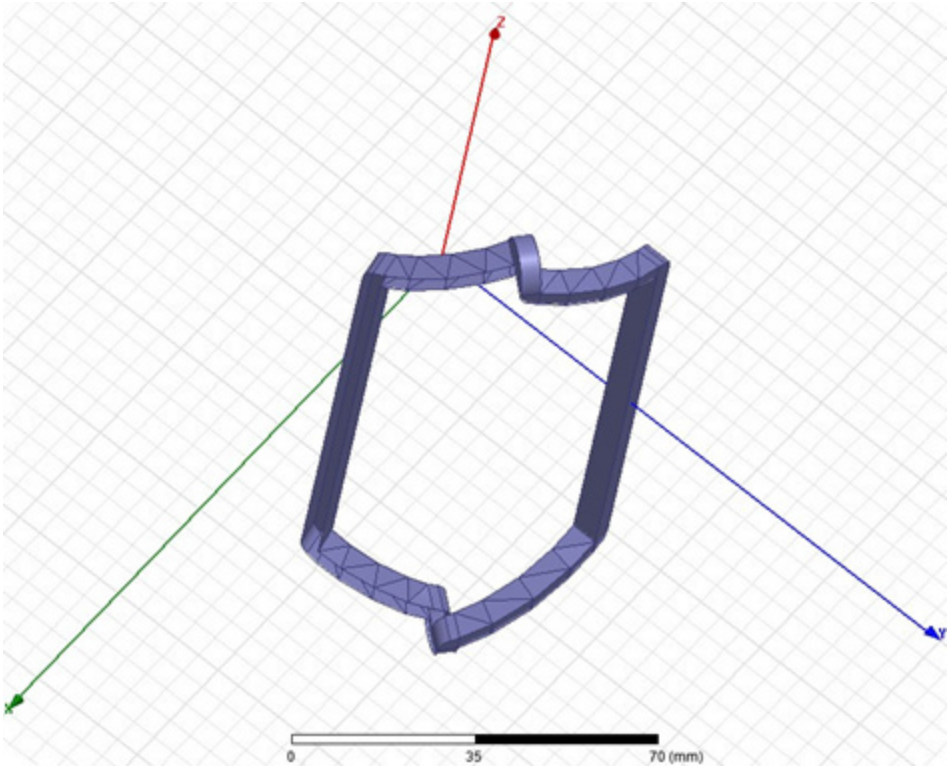


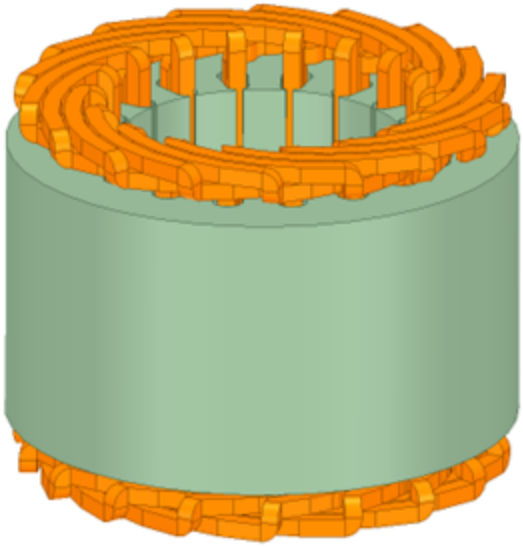
Figure 9-14 Note that it is a sheet instead of a solid: it is empty if you observe it carefully.

Creating a Bended Coil

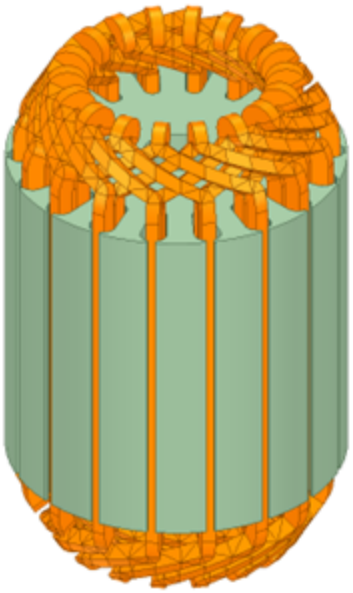
The following figures show an outer core lap winding with a 90° bended end coil (**BendAngle** = 90°), and an inner-core lap winding with a 30° bended end coil (**BendAngle** = 30°).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Outer-core lap winding with 90° bended end coil:



Inner-core lap winding with 30° bended end coil:



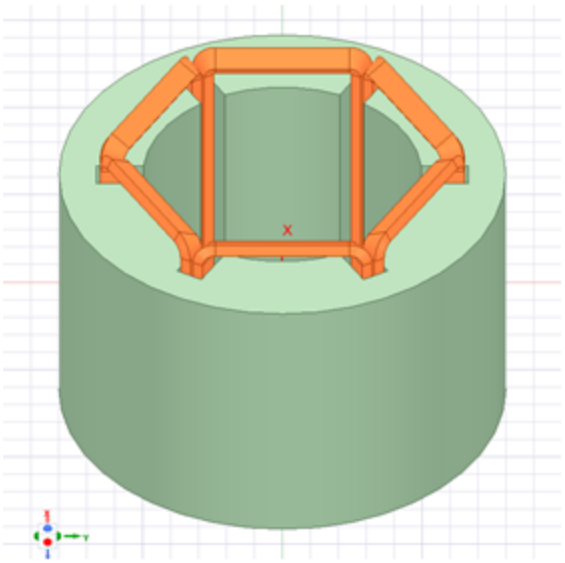
Creating a Tooth Coil

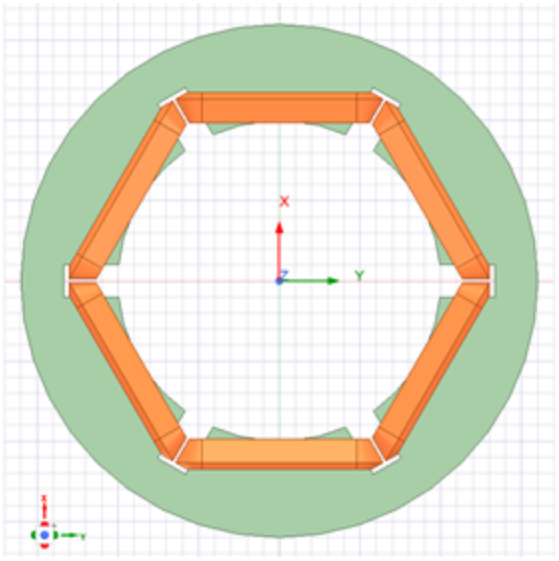
A Tooth Coil – a coil directly wound on a tooth – is a special lap coil with abs(**CoilPitch**) = 1. Examples of a tooth-coil winding with sloped straight end coil (**CoilPitch** = 1), and with flat arced end coil (**CoilPitch** = -1), are shown below. For an outer core with few slots, the straight end coil

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

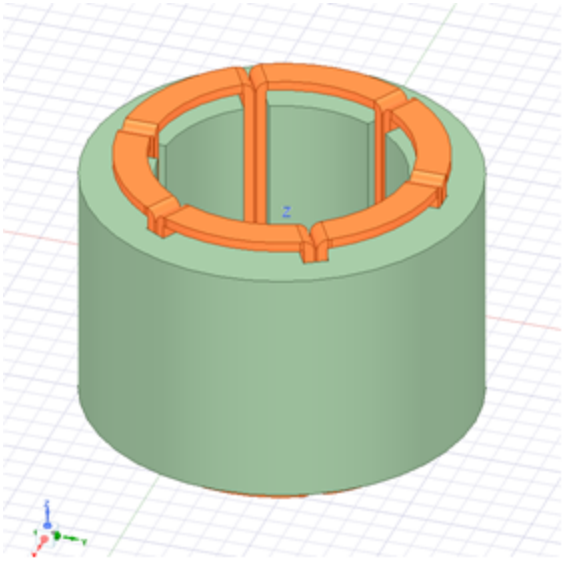
might cover the inner bore when CoilPitch = 1. In such a case, the conductor height inside slots should be reduced to ensure that the end coil will not overlap with the inner core which might be longer than the outer core. However, if you create a tooth coil with a flat arced end coil, you do not need to reduce the conductor height inside slots.

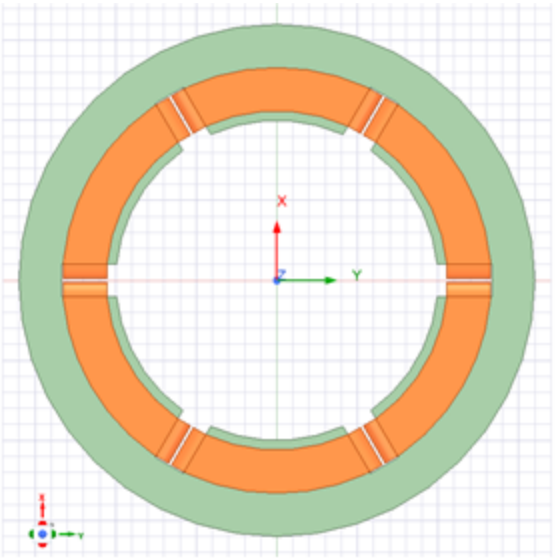
Tooth-coil winding with sloped straight end coil (**CoilPitch** = 1):





Tooth-coil winding with flat arced end coil (**CoilPitch** = -1):





Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ConCoil UDP

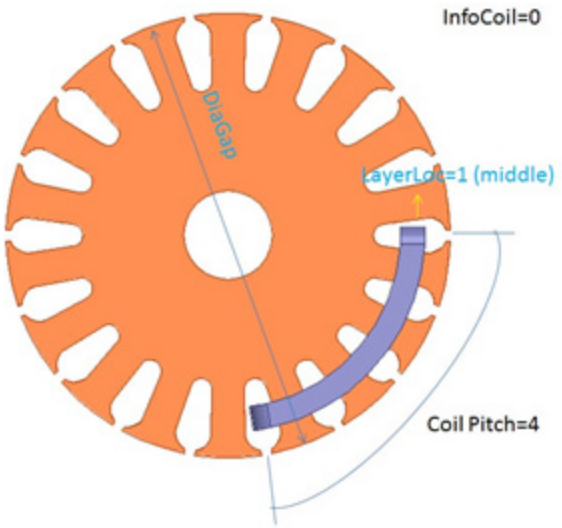
The ConCoil UDP is used to create a coil for concentric distributed AC windings or concentric DC field windings in a slotted core created by the SlotCore, VentSlotCore, RacetrackSlotCore, or NonSalientPoleCore UDP. This UDP can also create a coil cross-section used as a terminal for current assignment.

Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaYoke for outer cores.
DiaYoke	Core diameter on yoke side, DiaYoke < DiaGap for inner cores.
Length	Core length.
Skew	Skew angle in core length range.
Slots	Number of slots.
SlotType	Slot Type: 1 to 7.
Hs0	Slot opening height.
Hs1	Slot wedge height.
Hs2	Slot body height.
Bs0	Slot opening width.
Bs1	Slot wedge maximum width.
Bs2	Slot body bottom width, 0 for parallel teeth.
Rs	Slot body bottom fillet.
FilletType	0: a quarter circle; 1: a tangent connection.
LayerLoc	0: whole; 1: middle; 2: top; 3: bottom.
CoilPitch	Coil pitch measured in slots.
EndExt	One-side end extended length.
LayerExt	Span layer extension in the axial direction.
LayerDiff	Span layer difference in the radial direction.
AltEnd	0: same end layers; 1: alternate end layers.
LenRegion	Region length.
InfoCoil	0: coil; 1 terminal1; 2: terminal2; 100: region.

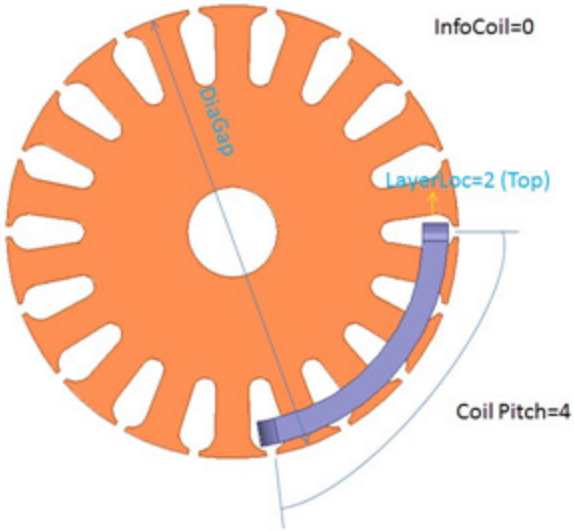
The following four 2D figures show the parameter **LayerLoc** with different values. Slot cores are used as references to help better understand concoils.

LayerLoc set to 1:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

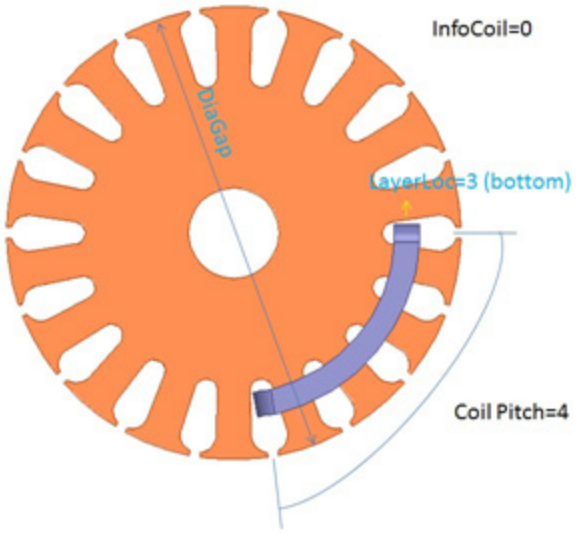


LayerLoc set to 2:

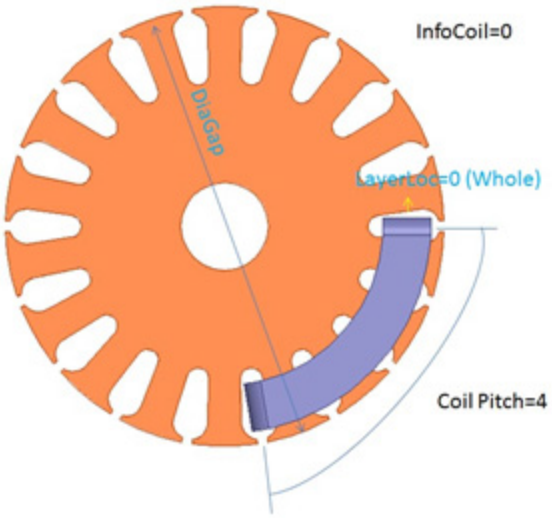


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LayerLoc set to 3:



LayerLoc set to 0:



The following four 3D figures illustrate parameters of ConCoil such as **LayerExt**, **LayerDiff** and **AltEnd**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reference 3D ConCoil:

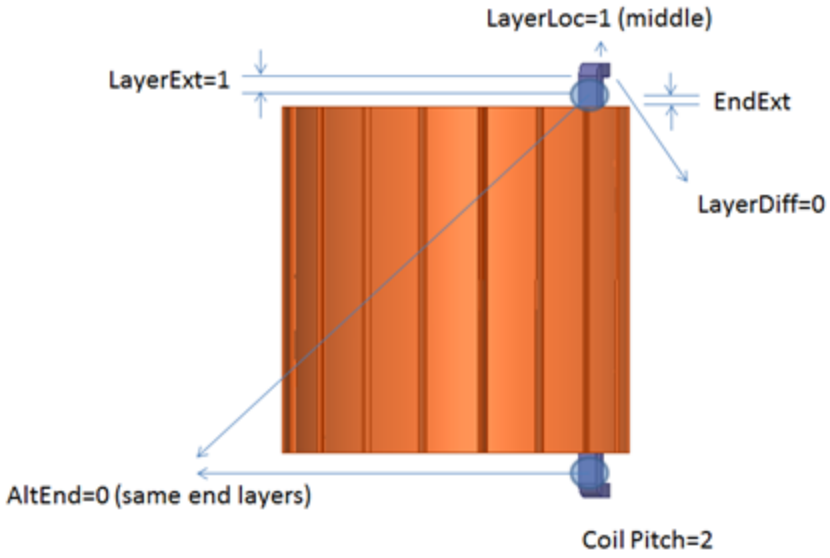
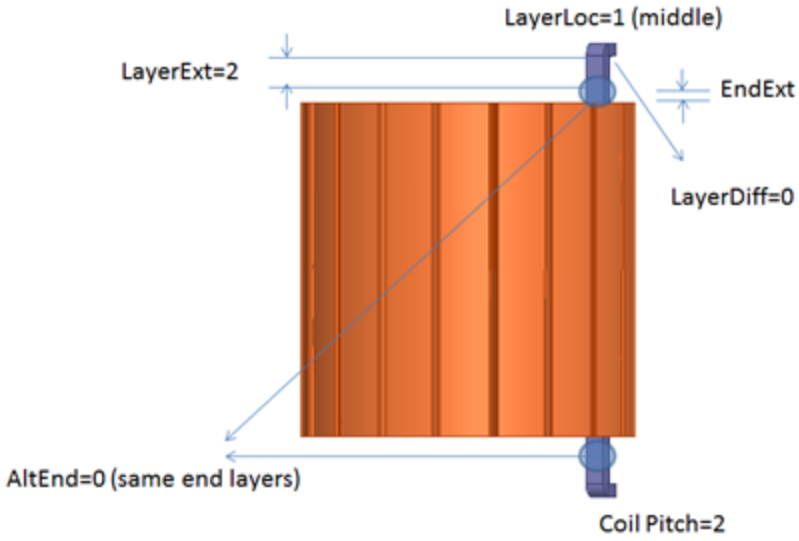


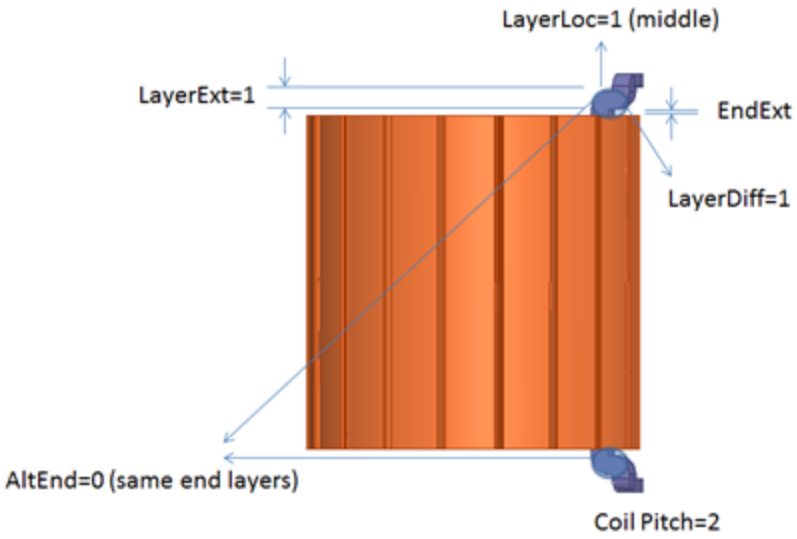
Figure 9-15

3D ConCoil with a different **LayerExt** than shown in reference concoil:

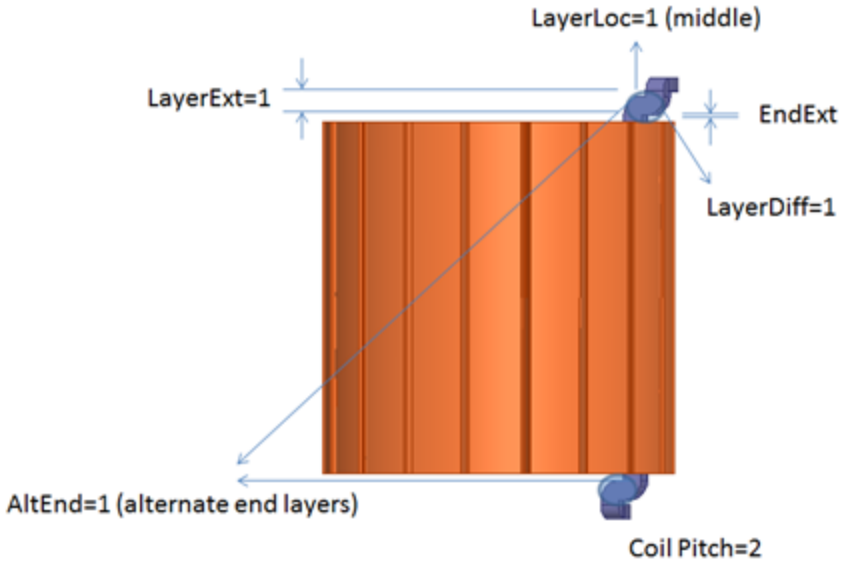


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3D ConCoil with a different **LayerDiff** than shown in reference concoil:



3D ConCoil with a different **AltEnd** than shown in previous figure"3D ConCoil with a different LayerDiff than shown in reference concoil:" above



> **Note:** For parameters such as **Skew**, **FilletType** and parameters related to slots, refer to SlotCore UDP parameters.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating a Single Concentric Coil and its Terminals

You create a single concentric coil by manually setting the property of **InfoCoil** to 0. See "ConCoil UDP " on page 9-47 for examples of the single concentric coil.

For the terminals of a concentric coil, set the property of **InfoCoil** to 1 or 2 (terminal1 or terminal2). The figure shows terminal 1 of a concentric coil.

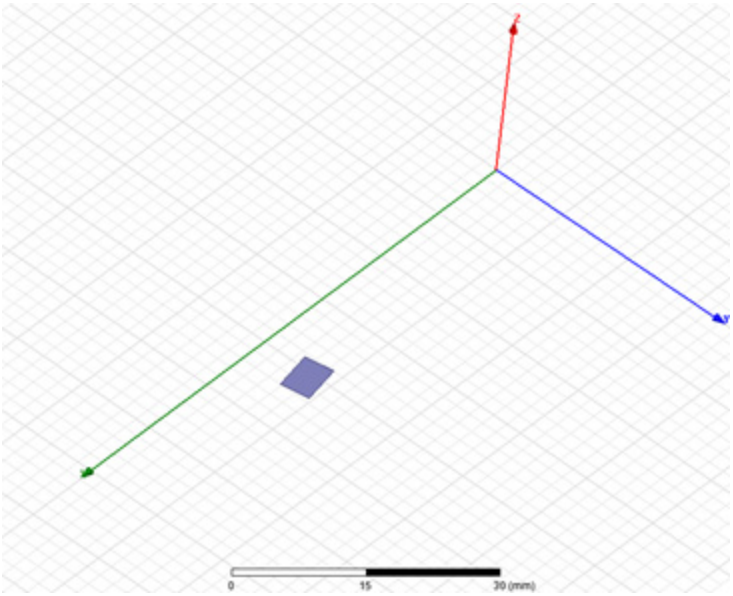


Figure 9-16 You can assign the value of this property either when creating a concentric coil or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

WaveCoil UDP

The WaveCoil UDP is used to create a wave-type coil in a slotted core created by the SlotCore or VentSlotCore UDP.

Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaYoke for outer cores
DiaYoke	Core diameter on yoke side, DiaYoke < DiaGap for inner cores
Length	Core length
Skew	Skew angle in core length range
Slots	Number of slots

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Property	Description
SlotType	Slot Type: 1 to 7
Hs0	Slot opening height
Hs1	Slot wedge height
Hs2	Slot body height
Bs0	Slot opening width
Bs1	Slot wedge maximum width
Bs2	Slot body bottom width, 0 for parallel teeth
Rs	Slot body bottom fillet
FilletType	0: a quarter circle; 1: a tangent connection
Layers	Number of winding layers
CoilPitch	Coil pitch measured in slots
EndExt	One-side end extended length
SpanExt	Axial length of end span; 0 for no span
SegAngle	Deviation angle for slot arches (5~15, <5 for true surface)
PolePitch	Pole pair pitch measured in slots
LenRegion	Region length
InfoCoil	0: winding; 1: one coil; 100: region

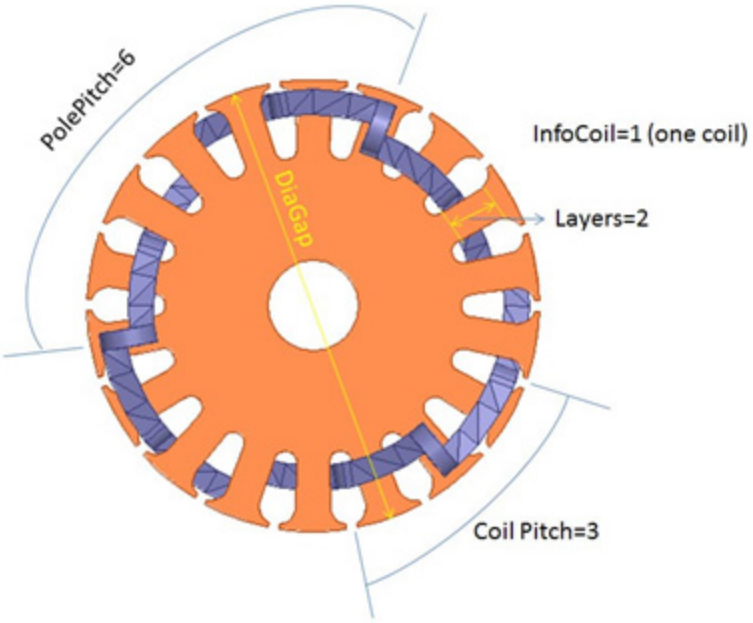
The following three figures illustrate these parameters of wave coil.

Note:

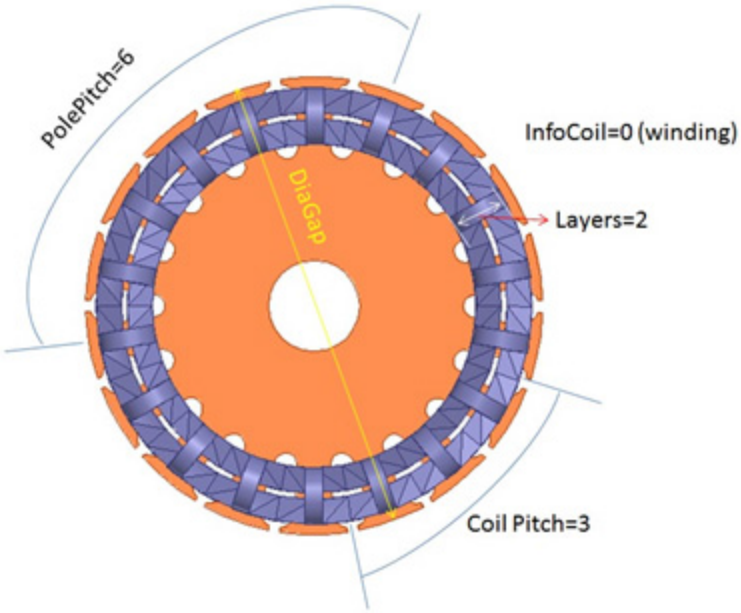
- A slot core (orange) is used as reference to better present the parameters and position of the wave coil.
- Most parameters in a wave coil are similar or exactly the same as those in a lap coil except **PolePitch**. "One Wave Coil" on the next page shows the definition of the parameter **PolePitch**. "A Whole Winding Wave Coil" on the next page presents a winding wave coil that can be compared with a single wave coil in "One Wave Coil" on the next page. Further, "Parameters of Wave Coil in Axial Direction" on page 9-55 gives an overview of parameters in axial direction.
- For parameters such as **Skew**, **FilletType**, **SegAngle** and parameters related to slots, refer to SlotCore UDP parameters.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

One Wave Coil

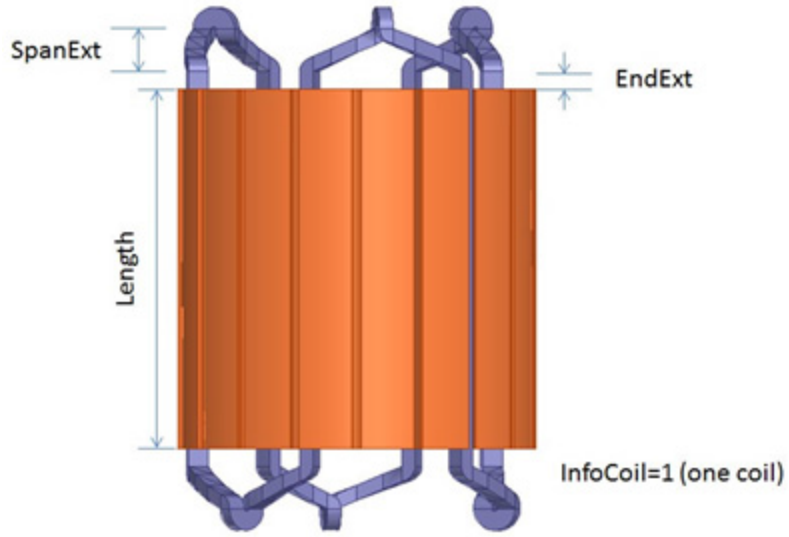


A Whole Winding Wave Coil



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameters of Wave Coil in Axial Direction



Creating a Whole Wave Winding

You can create a whole wave winding by manually setting the property of **InfoCoil** to 0.

You can assign the value of this property either when creating a wave coil or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

An example is shown in "A Whole Winding Wave Coil" on the previous page.

Creating a Single Wave Coil

In Maxwell, wave coil is set to a single wave coil by default when being created (**InfoCoil** is equal to 1).

You can assign the value of this property either when creating a wave coil or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Examples are shown in "One Wave Coil" on the previous page and "Parameters of Wave Coil in Axial Direction" above.

SquirrelCage UDP

The SquirrelCage UDP is used to create a squirrel-cage winding in a core created by the SlotCore or VentSlotCore UDP for single-phase or multi-phase induction machines.

Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaYoke for outer

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

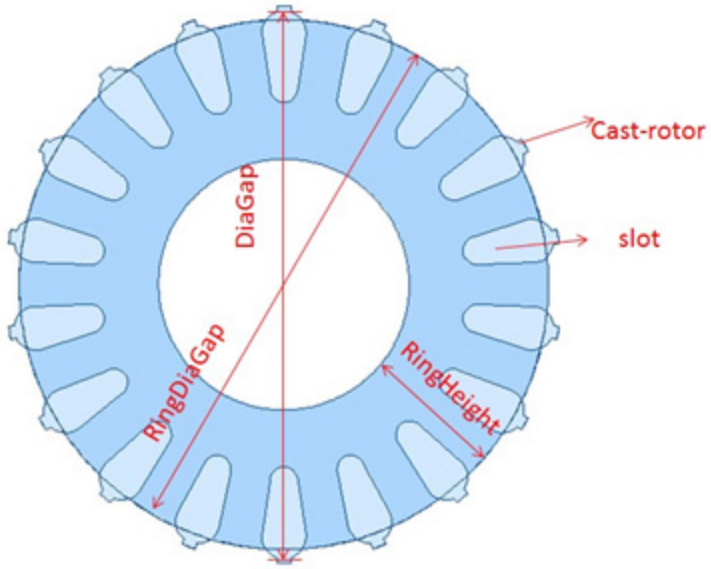
Property	Description
	cores
DiaYoke	Core diameter on yoke side, DiaYoke < DiaGap for inner cores
Length	Core length
Skew	Skew angle in core length range
Slots	Number of slots
SlotType	Slot Type: 1 to 7
Hs0	Slot opening height
Hs01	Slot closed bridge height
Hs1	Slot wedge height
Hs2	Slot body height
Bs0	Slot opening width
Bs1	Slot wedge maximum width
Bs2	Slot body bottom width, 0 for parallel teeth
Rs	Slot body bottom fillet
FilletType	0: a quarter circle; 1: a tangent connection
HalfSlot	0: symmetric slots; 1: half slots
BarEndExt	One-side bar end extended Length
RingLength	One-side axial ring length
RingHeight	Radial ring height
RingDiaGap	Ring diameter on gap side
CastRotor	0: insert-bar; 1: cast-rotor
SegAngle	Deviation angle for slot arches (10~30, <10 for true surface)
LenRegion	Region length
InfoCoil	0: bars & rings; 1: bars; 2: rings; 100: region

These parameters are used in the figures that follow. Note the following:

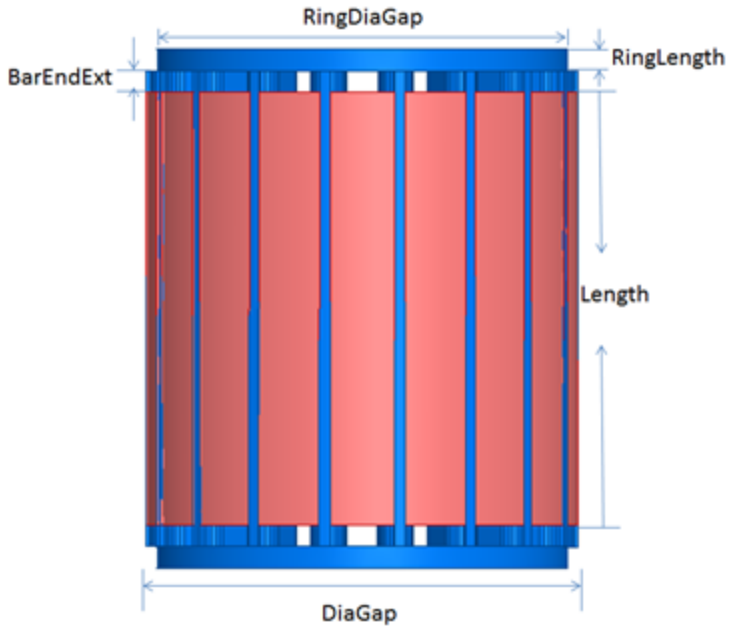
- **RingDiaGap** is set to 95 mm whereas **DiaGap** is 100 mm to distinguish these two parameters in the following figures.
- In the following 3D figure, a slot core is drawn to show the **BarEndExt** and **Length** of Squirrel Cage.
- For parameters such as **Skew**, **FilletType**, **SegAngle**, and parameters related to slots, refer to SlotCore UDP parameters.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameters of squirrel cage on cross-section:



Parameters of squirrel cage on axial direction:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

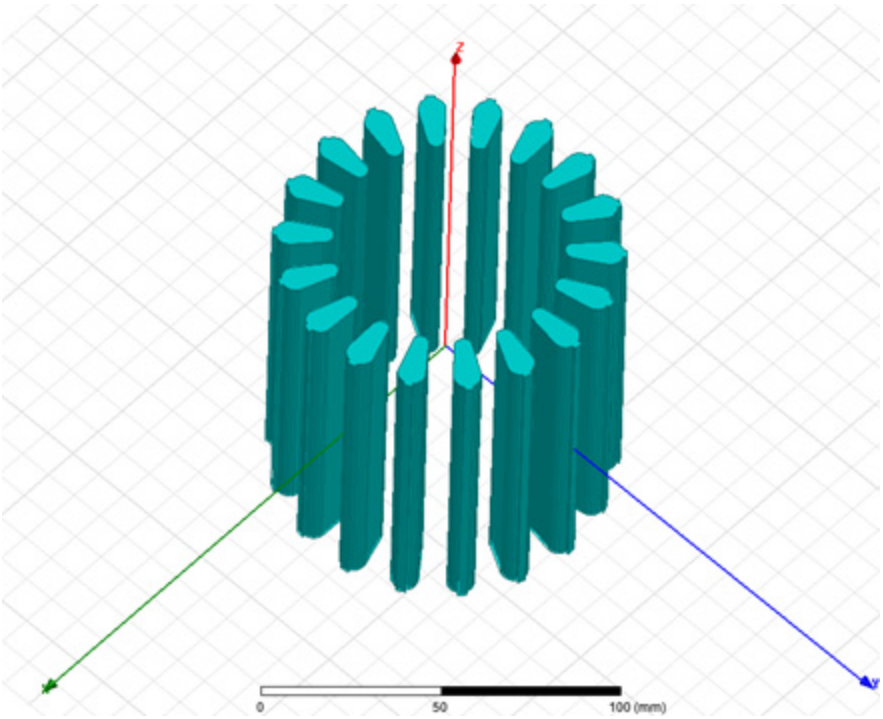
Creating a Whole Squirrel-Cage Winding

A whole squirrel cage winding is set by default when being created (**InfoCoil** is equal to 0). An example is shown in "Parameters of squirrel cage on axial direction:" on the previous page as a blue object.

You can assign the value of this property either when creating a squirrel cage or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Creating Bars Only (Squirrel Cage)

You can create bars of a squirrel cage by manually setting the property of **InfoCoil** to 1. The following figure shows bars of a squirrel cage.

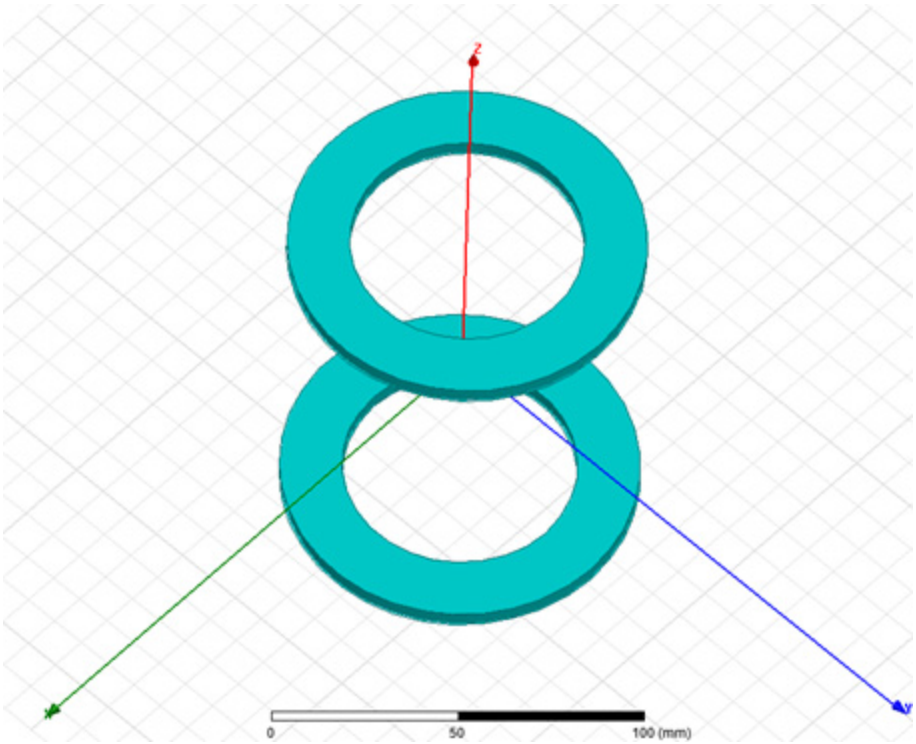


You can assign the value of this property either when creating a squirrel cage or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Creating Two Rings Only

You can create two rings of a squirrel cage by manually setting the property of **InfoCoil** to 2. The figure that follows shows rings of a squirrel cage.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can assign the value of this property either when creating a squirrel cage or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

DoubleCage UDP

The DoubleCage UDP is used to create a double squirrel-cage winding in a slotted core created by the SlotCore UDP or VentSlotCore UDP for induction machines with a double squirrel-cage rotor. The created double squirrel-cage winding can also be used as a subtract tool to derive a core with top and bottom slots.

Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaYoke for outer cores.
DiaYoke	Core diameter on yoke side, DiaYoke < DiaGap for inner cores.
Length	Core length.
Skew	Skew angle in core length range.
Slots	Number of slots.
SlotType	Slot Type: 1 to 7.
Hs0	Slot opening height.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

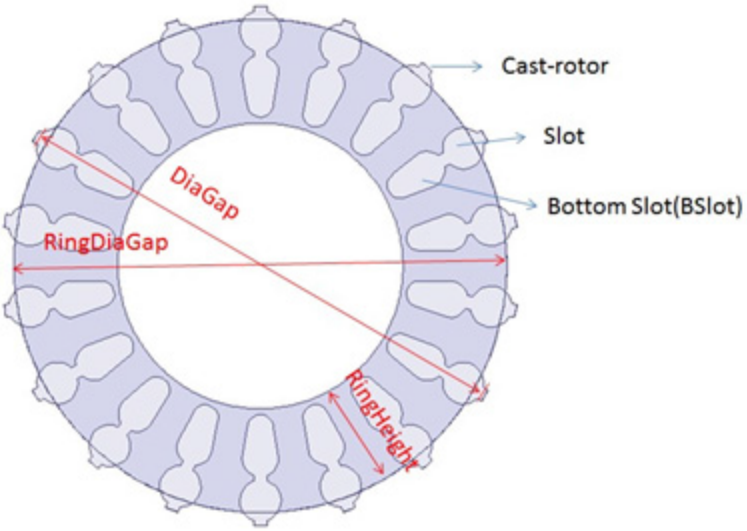
Property	Description
Hs01	Slot closed bridge height.
Hs1	Slot wedge height.
Hs2	Slot body height.
Bs0	Slot opening width.
Bs1	Slot wedge maximum width.
Bs2	Slot body bottom width, 0 for parallel teeth.
Rs	Slot body bottom fillet.
FilletType	0: a quarter circle; 1: a tangent connection.
HalfSlot	0: symmetric slots; 1: half slots.
BarEndExt	One-side bar end extended Length.
RingLength	One-side axial ring length.
RingHeight	Radial ring height.
RingDiaGap	Ring diameter on gap side.
DoubleCage	0: normal squirrel cage; 1: double squirrel cage.
BSlotType	Bottom slot type: 1 to 4.
BHs0	Slot opening height.
BHs1	Slot wedge height.
BHs2	Slot body height.
BBs0	Slot opening width.
BBs1	Slot wedge maximum width.
BBs2	Slot body bottom width, 0 for parallel teeth.
BRs	Slot body bottom fillet.
CastRotor	0: insert-bar; 1: cast-rotor.
SegAngle	Deviation angle for slot arches (10~30, <10 for true surface).
LenRegion	Region length.
InfoCoil	0: bars & rings; 1: bars; 2: rings; 100: region.

The main difference between double cage and squirrel cage is that a double cage has double slots instead of single slots. The following 2D figure shows the bottom slot, which is connected to the normal slot. The parameters for a bottom slot are similar to those of a normal slot. Other parameters are exactly the same as a squirrel cage.

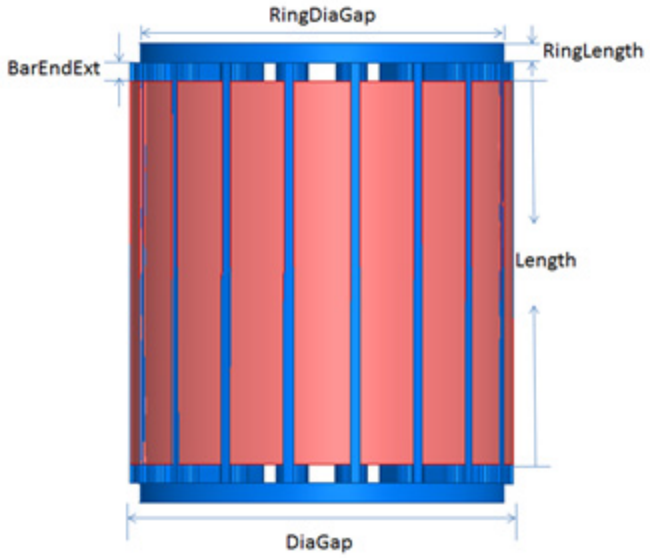
> **Note:** For parameters such as **Skew**, **FilletType**, **SegAngle** and parameters related to slots, refer to SlotCore UDP parameters.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameters of double cage on cross-section



Parameters of double cage on axial direction



Creating a Whole Double-Cage Winding

A whole double-cage winding is set by default when being created (**InfoCoil** is equal to 0). An example is shown in "Parameters of double cage on axial direction" above as a blue object.

You can assign the value of this property either when creating a squirrel cage or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating Bars Only (Double Cage)

You can create bars of a double cage by manually setting the property of **InfoCoil** to 1. The figure that follows shows bars of a double cage.

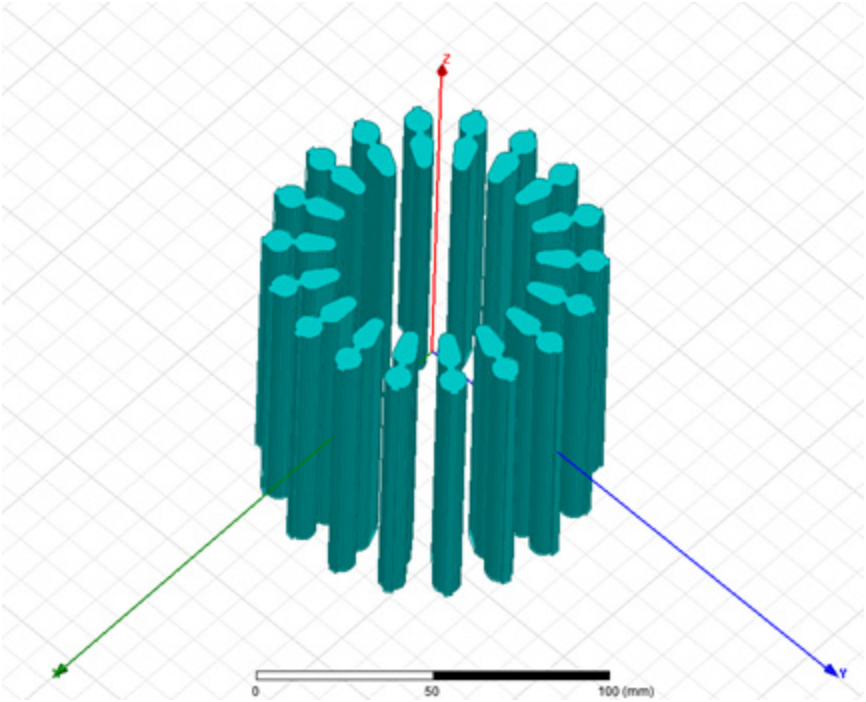
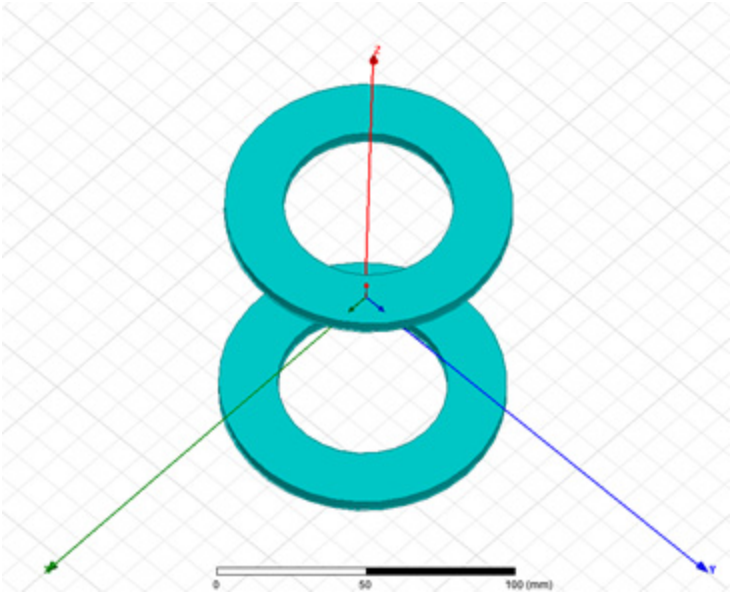


Figure 9-17 You can assign the value of this property either when creating a double cage or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Creating Two Rings Only (Double Cage)

You can create two rings of a double cage manually by setting the property of **InfoCoil** to 2. The following figure shows rings of a double cage.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can assign the value of this property either when creating a double cage or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

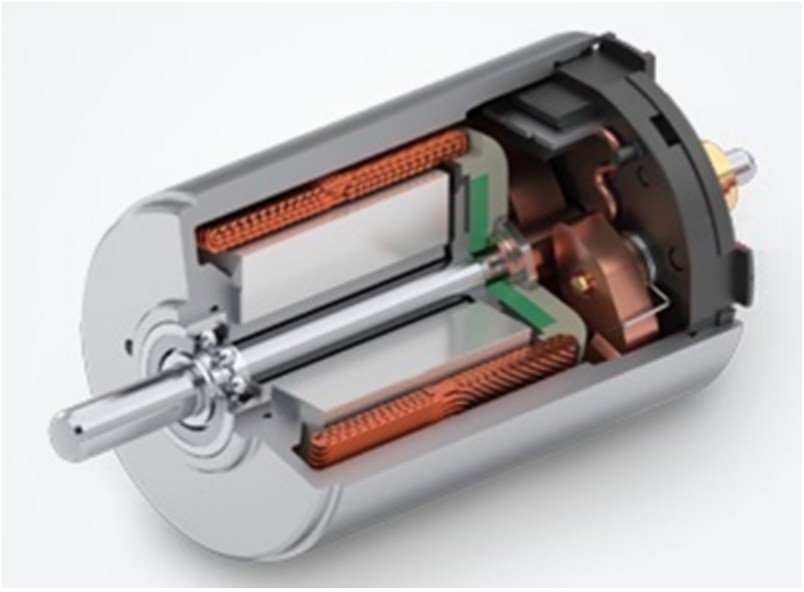
CupCoil UDP

The CupCoil UDP is used to create a coil for distributed slotless AC windings or coreless DC commutating windings in the air-gap between outer and inner cores. This UDP can also create a coil cross-section, which can be used as a terminal for current assignment.

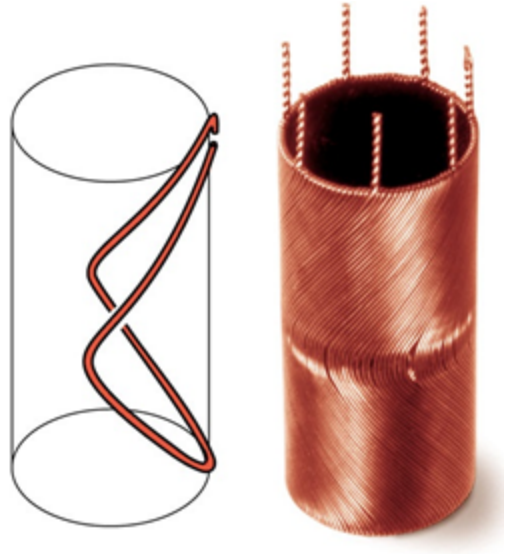
Property	Description
DiaOuter	Coil outer diameter
DiaInner	Coil inner diameter
Length	Length of coil straight part
Coils	Number of total coils in a circle
CoilType	1: lap; 2: wave
CoilPitch	Coil pitch in degrees
LenTotal	Tip-to-tip length of a coil, 0 for 2D
LenRegion	Region length
InfoCoil	0: coils; 1: coil; 2 terminal1; 3: terminal2; 100: region

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coreless DC motor



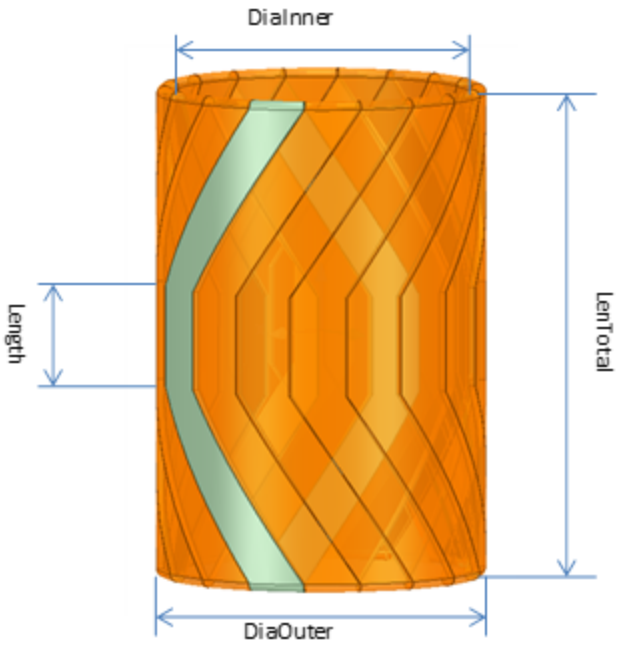
Coreless DC Winding



The following figure shows some of the parameters described in the table above. To avoid too small a mesh size, wire turns are not created one-by-one as solid conductors. Instead, all adjacent wires carrying the same current are grouped as a strand coil with one or more turns.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A coreless lap-type DC winding created by CupCoil UDP



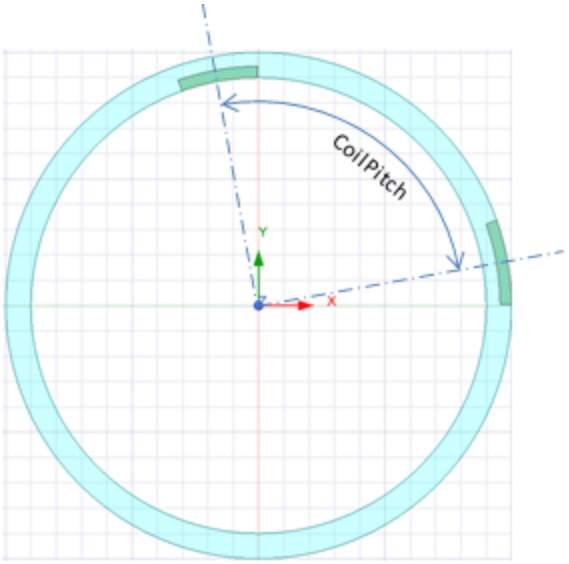
Creating a Coil and its Terminals (CupCoil)

When **InfoCoil** = 0, the created object includes all coils that are united. To separate them, you can use the **Modeler** > **Boolean** > **Separate Bodies** menu command. When **InfoCoil** = 1, the created object includes only one coil, as shown by the coil in green color in the figure. You can use menu command **Edit** > **Duplicate** > **Around Axis** to duplicate it.

When **InfoCoil** = 2, you can create a terminal for coil assignment. You can create a return terminal for a coil in 2D by setting **InfoCoil** = 3. The following figure shows both “go” and “return” terminals, and shows the definition of the **CoilPitch** parameter (90 degrees in this example).

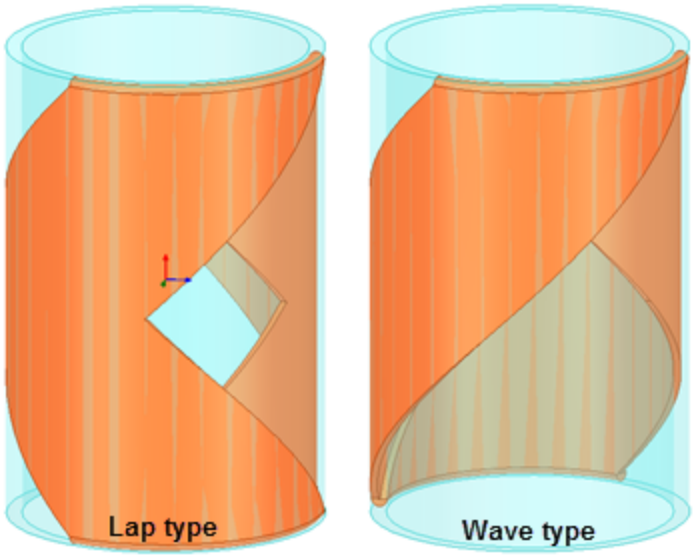
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

“go” and “return” terminals for a CupCoil with a given CoilPitch



When **CoilType** = 1 , the created coil is a lap-type. When **CoilType** = 2, the created coil is a wave-type. For a wave-type coil, the bottom span will be skewed in the same direction as the top span, and therefore, the bottom tip will shift from the top tip by one pole pitch. The pole pitch will be 180°, 90°, 60°, …, for 2, 4, 6, …, poles, respectively. The input value of **CoilPitch** will be adjusted to the closest value of possible pole pitches. The following figure compares a wave-type coil with a lap-type coil.

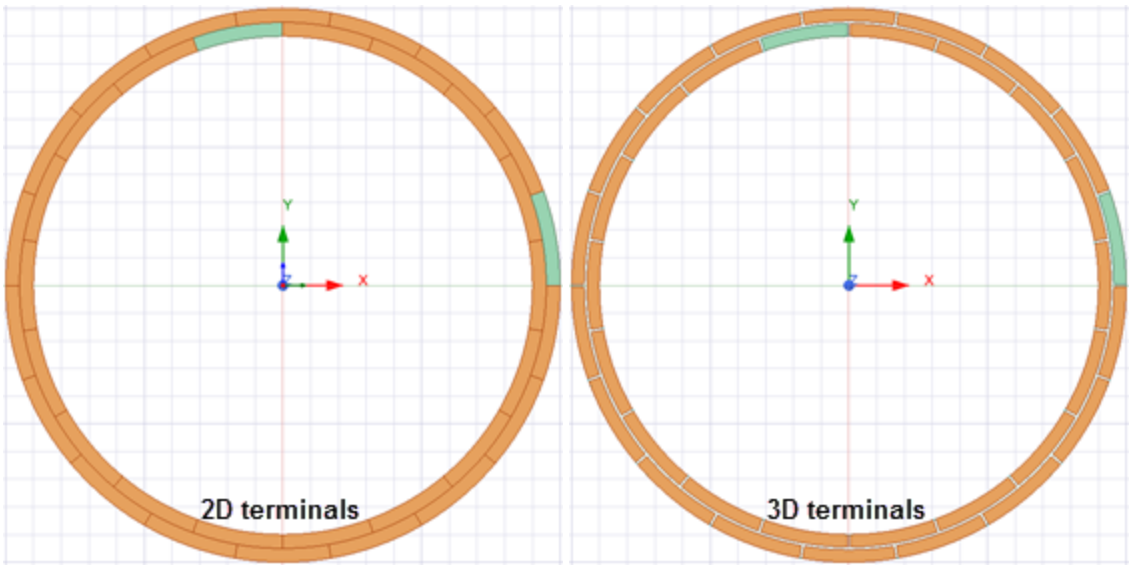
Lap-type coil compared with a wave-type coil



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can set **LenTotal** = 0 to create coils or terminals in 2D, or directly use **InfoCoil** = 2 and **InfoCoil** = 3 with **LenTotal** > 0 to draw 3D terminals for 2D coils. The difference between 2D terminals and 3D terminals is that: there will be a gap between any two adjacent 3D coil terminals, but there will be no gap between 2D terminals to improve mesh quality, as compared in the figure below.

2D terminals compared with 3D terminals



A user needs to input an average CoilPitch to create coil terminals for 2D FEA model. For a wave-type coil, because the bottom span is continuously skewed without changing the skew direction from the top span, the average coil pitch is the same as the real CoilPitch. For a lap-type coil, the average coil pitch can be derived from:

PitchAve = (Length x CoilPitch + (LenCore – Length) x (CoilPitch + PitchMin) / 2) / LenCore

where **LenCore** is the length of the inner and outer cores which carry magnetic field, and

PitchMin = (LenTotal – 2 x RadTip – LenCore) / (LenTotal – 2 x RadTip – Length) x CoilPitch

which must be limited within (0, CoilPitch). In the equation above, **RadTip** is the radius of top and bottom coil tips, which is obtained from:

RadTip = (DiaOuter – DiaInner) / 4

The skew angle for multi-slice 2D FEA model can be obtained from:

AngSkew = (PitchAve – PitchMin) x 2

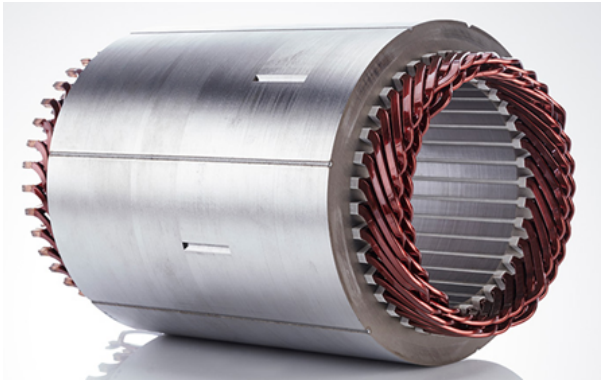
which is applicable for both lap-type and wave-type coils.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HairpinCoil UDP

The HairpinCoil UDP is used to create a hairpin-type coil. Hairpin coils are used to make a hairpin winding in a slotted core. All hairpin coils belonging to the same phase are welded in series to perform a hairpin winding.

Hairpin Coil



Stator with Hairpin Winding

Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaYoke for outer cores
DiaYoke	Core diameter on yoke side, DiaYoke < DiaGap for inner cores
Length	Core length, 0 for 2D
Skew	Skew angle in core length range
Slots	Number of slots
SlotType	Slot Type: 1 to 7
Hs0	Slot opening height
Hs1	Slot wedge height.
Hs2	Slot body height
Bs0	Slot opening width
Bs1	Slot wedge maximum width
Bs2	Slot body bottom width; 0 for parallel teeth
Rs	Slot body bottom fillet
FilletType	0: a quarter circle; 1: a tangent connection; 2&3: arc bottom; 4&5: V bottom
Layers	Number of total coil layers in a slot; must be an

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Property	Description
	even number
CoilPitch	Coil pitch measured in slots
EndExt	One-side end extended length
SpanExt/SpanGap	> 0: axial span extension; = 0: 45° span angle; < 0: negative span gap
SegAngle	Deviation angle for slot arches (5~15, <5 for true surface)
1stPitch	Going-side leg-span pitch
2ndPitch	Returning-side leg pitch; see explanation below.
2ndSpanExt/Span Gap	> 0: axial span extension; = 0: 45° span angle; < 0: negative span gap
WireWidth	Wire width in slot width direction
WireThick	Wire thickness in slot depth direction
WireRad	Wire fillet radius; used to calculate equivalent wire size based on wire area.
LayerSetGap	Gap between two layer-sets; one layer-set is two layers of one coil.
LayerSetIdx	Layer-set index for the location of the coil to be drawn
LayerGap	Gap between two layers in a layer set
WindingType	1: whole-coiled (2 coil sets per pole pair) 2: half-coiled (1 coil set per pole pair)
CoilSetIdx	0: create 1st coil set 1: create the 2nd coil set (for whole-coiled winding only)
LenRegion	Region length, to limit lead length for open coils
InfoCoil	0: winding; 1: one coil set; 2: terminal1; 3: terminal2; 4: coil; 5: end-tip; 6: lead1; 7: lead2; 100: region

Parameter Considerations

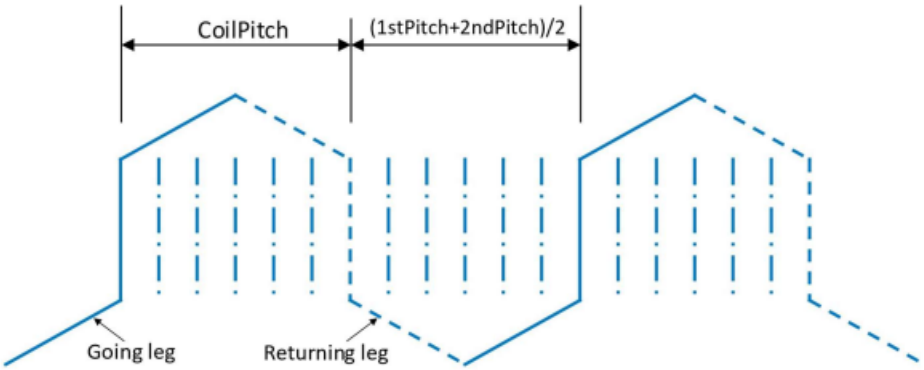
- Parameter **Layers** defines the total number of layers, or total number of conductors per slot. The value must be an even number. Two layers related to one coil are referred as one layer-set.
- For the parameter **SpanExt/SpanGap**, when its value > 0, it defines the span extension length in axial direction (including span connectors, but not including U-type end tips) . When the value is 0, the span extension will be automatically derived based on a 45° span

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

slope angle at the inner diameter of the whole winding.
When the value < 0, the minimum span gap between two adjacent coils at the inner diameter of the whole winding will be defined as a negative value based on which the span extension is automatically derived. In this way, all span extensions at different layers will have the same length.

- Parameters **1stPitch** and **2ndPitch** define the going-side and returning-side leg-span pitches, respectively, as shown in the figure that follows. The going-side and returning-side span tips of two adjacent coils are to be welded. The default value of 0 for the **1stPitch** means symmetrical leg spans, that is **1stPitch** = **2ndPitch**.

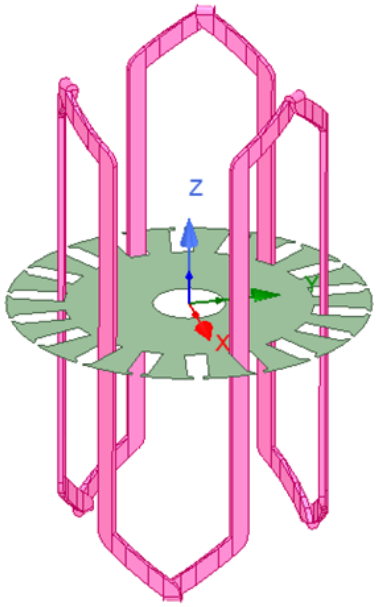
  To be consistent with the previous-version definition, one side leg-span corresponds to a half-span pitch.



Parameters for CoilPitch and 2ndPitch

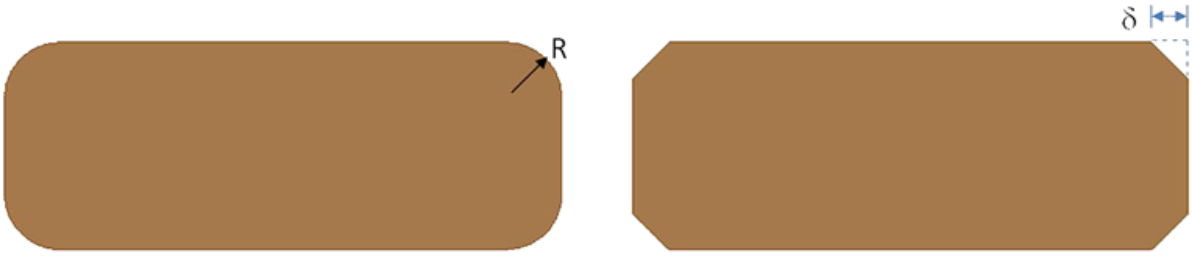
For an integer-slot winding with symmetrical lag spans, the **Slots** value is a multiple of **2ndPitch** plus **CoilPitch** values. In such a case, the last coil in a coil chain will close the first coil, resulting in a closed coil chain, as shown below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Closed hairpin-coil chain with 2ndPitch = 3 in an 18-slot core lamination

- Parameter **2ndSpanExt/SpanGap** defines the span extension length for the returning side leg spans, please refer to the **SpanExt/SpanGap** consideration above.
- As a good compromise between accuracy and geometrical modeling effort, for 3D models, the UDP will convert the round corner fillets of the conductors' cross-section to 45-degree chamfers at the corners as shown in (b) of the figure that follows. This conversion is done in such a way that the cross-sectional area of the conductor is maintained. In 2D models, equivalent conductors with 45-degree corner will be used when SegAngle ≥ 5°, and conductors with true round corner will be used when SegAngle < 5°.



Equivalent conductors with the same area: δ = R (2-π/2)^0.5

(a) Conductor with true round corner

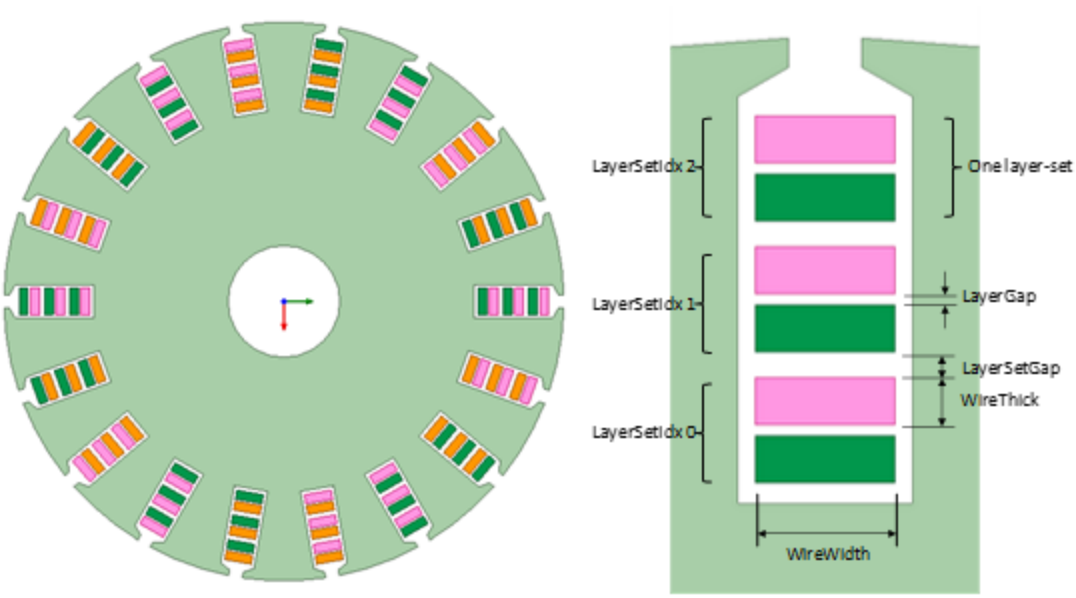
(b) Conductor with 45-degree line corner

- In order to obtain good mesh quality, the UDP will not create wire with wire fillet radius, as shown above, but the effect of wire fillet radius on the wire area will be considered using a scaling factor for both **WireWidth** and **WireThick**. The scaling factor k = (WireArea /

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

WireWidth / WidthThick) ^ 0.5, where WireArea is the wire cross-section area considering wire fillet.

- The UDP will get the maximum available rectangular space inside a slot via an optimization process for wire arrangement. The minimum gap between a wire and a rectangle side will be default as 0.3 mm, considered as the thickness of a slot liner insulation. The gap between two layers in one layer-set is the same for all layer-sets. The value is automatically derived from the height of the rectangular space minus total wire thickness and total layer-set gap.

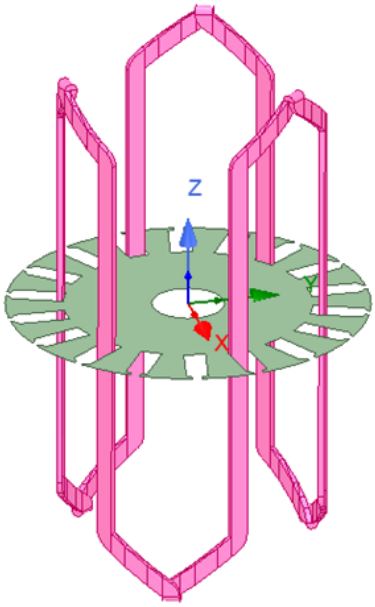


- **WindingType** and **CoilSetIdx** are used for fractional-slot windings only. You do not need to input these values for integer-slot windings. For a fractional-slot winding, the last coil in a coil chain will not close the first coil, and the coil chain will continue to include all coils belonging to the same coil set. Finally, the UDP will create an opened coil set. Two lead wires will extend the opened coil set to touch one region end. For whole-coiled windings, because the two coil sets may have different coil arrangements, we need to create two coil sets separately. Parameter **CoilSetIdx** indicates which coil set is being created. The coil arrangement is based on three-phase windings.
- For other parameters, refer to *LapCoil UDP*.

Creating Closed Three-Phase Hairpin Windings (HairpinCoil)

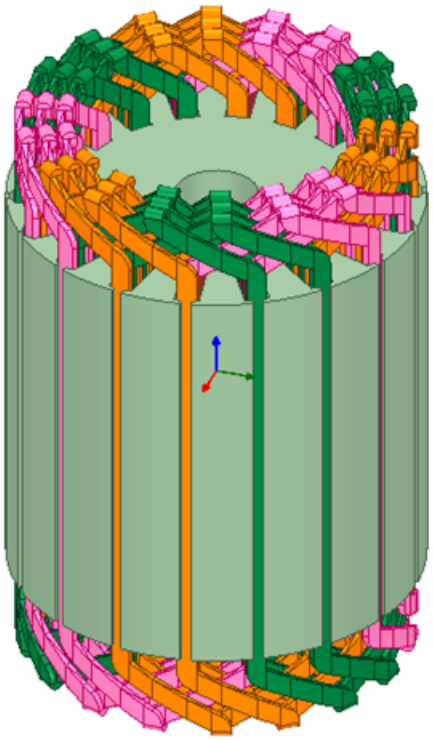
1. In Maxwell 3D, launch HairpinCoil UDP with default setting, that is **Slots**=18, **Layers**=6, **CoilPitch**=3, **2ndPitch**=3, **LayerSetIdx**=0, and **InfoCoil**=1. Because the ratio of **Slots** to (**2ndPitch** + **CoilPitch**) is 3 (an integer number), the UDP will create a closed-coil chain, or a coil loop, located at slot bottom, as shown below (excluding the core lamination).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Make a copy of the coil loop, and change **LayerSetIdx** to 1, this copied coil loop is changed to locate at the slot center.
3. Make a copy of the coil loop again, and change **LayerSetIdx** to 2, this copied coil loop is changed to locate at the slot top.
4. Duplicate all these three coil loops with the following parameters in **Duplicate/Around Axis** command: **Angle** = 20deg, **Total Number** = 6, and change colors for different phases, then you will get the final three-phase windings, as shown below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Closed Three-Phase Hairpin Windings

Creating One Phase Half-Coiled Windings (HairpinCoil)

Set **Slots**=15, **Layers**=2, **CoilPitch**=4, **2ndPitch**=4, **LayerSetIdx**=0, and **InfoCoil**=1. Because the ratio of **Slots** to (**2ndPitch** + **CoilPitch**) is 15/8 (not an integer number), the UDP will create an opened coil set. In this UDP, the software will support a fractional-slot winding that satisfies:

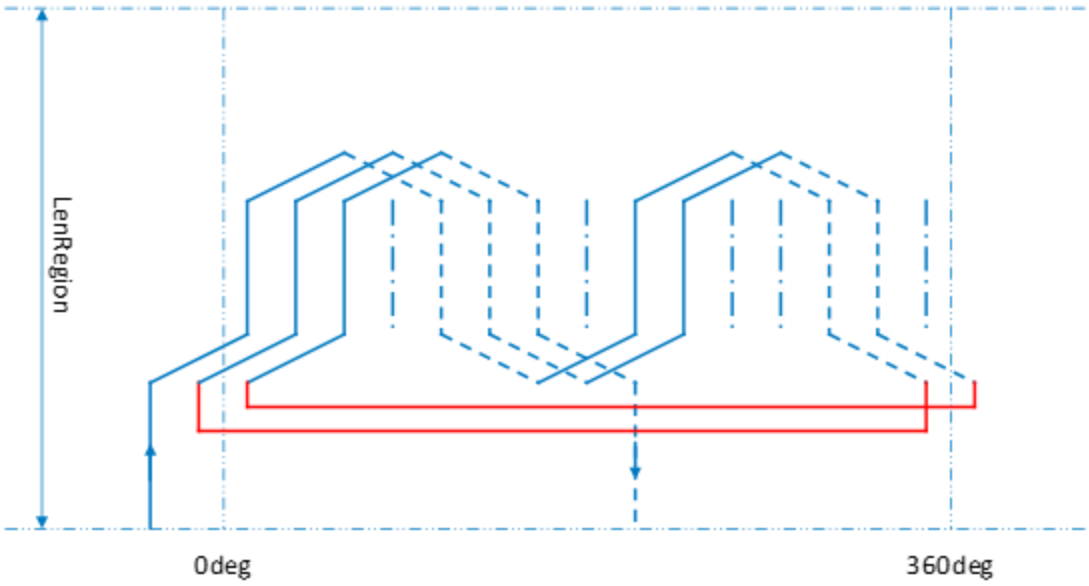
(2ndPitch + CoilPitch) x PolePairs = Slots ± 1

For this example, the number of pole pairs is 2, and the winding arrangement, for top coil side only, will be:

AAABBCCCAABBBCC

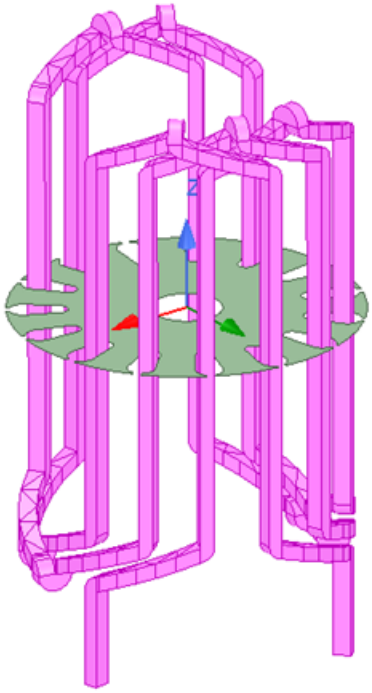
For this half-coiled winding, all hairpin coils labeled with “A” are included in one opened coil set, which has already contained all phase A coils, as shown below, where red lines indicate the short-circuit connection without using physical wires.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Flat Expanded View of Phase-A Hairpin Winding with One Opened Coil Set

The created 3D opened winding is shown below.



3D View of Phase-A Hairpin Winding with One Opened Coil Set

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating One Phase Whole-Coiled Windings (HairpinCoil)

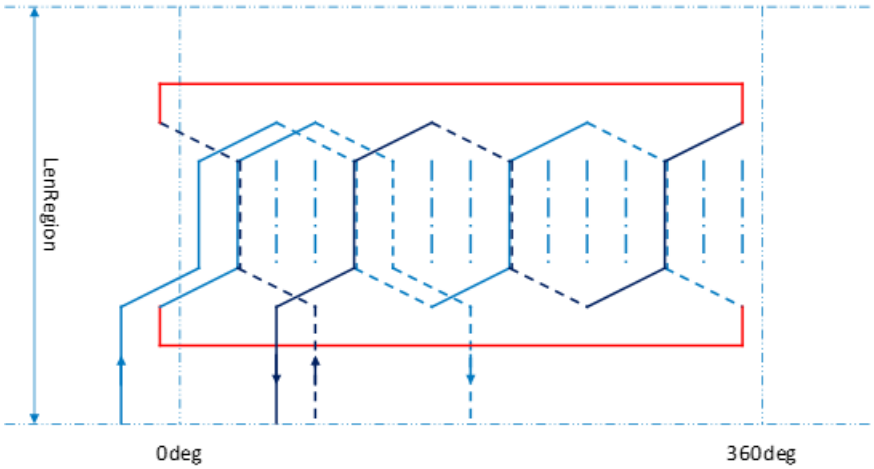
Set **Slots**=15, **Layers**=2, **CoilPitch**=4, **2ndPitch**=4, **LayerSetIdx**=0, **WindingType** = 1, **CoilSetIdx** = 0, and **InfoCoil**=1. Because the ratio of **Slots** to (**2ndPitch** + **CoilPitch**) is 15/8 (not an integer number), the UDP will create an opened coil chain. This fractional-slot winding satisfies:

(2ndPitch + CoilPitch) x PolePairs = Slots ± 1

where the number of pole pairs is 2. The winding arrangement, for top coil side only, is:

AAZBXCCYAZBBXCY

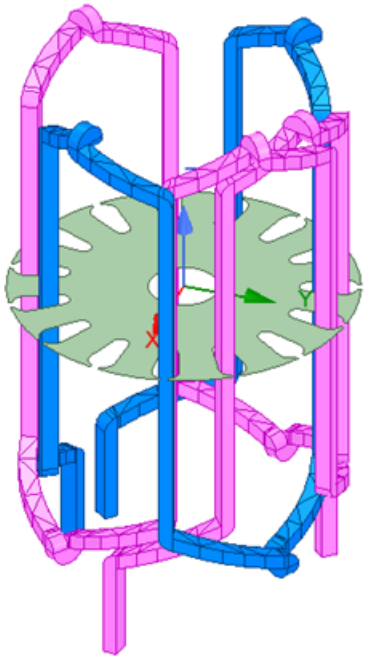
With the above parameter setting, this UDP will create an opened coil set including all coils related to all “A” coils of the above winding arrangement. Copy the coil set, and change **CoilSetIdx** to 1. The coil set is modified to include all coils related to all “X” coils of the winding arrangement. The phase-A winding is made up of these two coil sets, as shown below, where red lines indicate the short-circuit connection without using physical wires. All coils with light blue color relate to “A” coils, and all coils with dark blue color relate to “X” coils, in the winding arrangement.



Flat expanded View ofPphase-A Hairpin Winding with Two Opened Coil Sets

The created 3D opened winding is shown below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

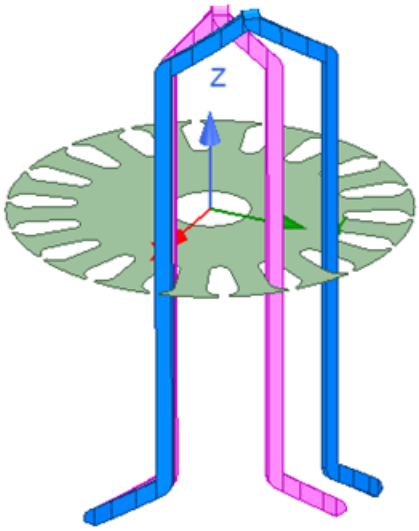


3D View of Phase-A Hairpin Winding with Two Opened Coil Sets

Creating Two Hairpin Coils Connected with End-Tip and Leads

1. Set **Slots**=18, **Layers**=6, **CoilPitch**=2, **2ndPitch**=3, **LayerSetIdx**=0, and **InfoCoil**=4. These parameter settings will create a single hairpin coil located at the slot bottom, as shown by the pink-color coil in the figure below.

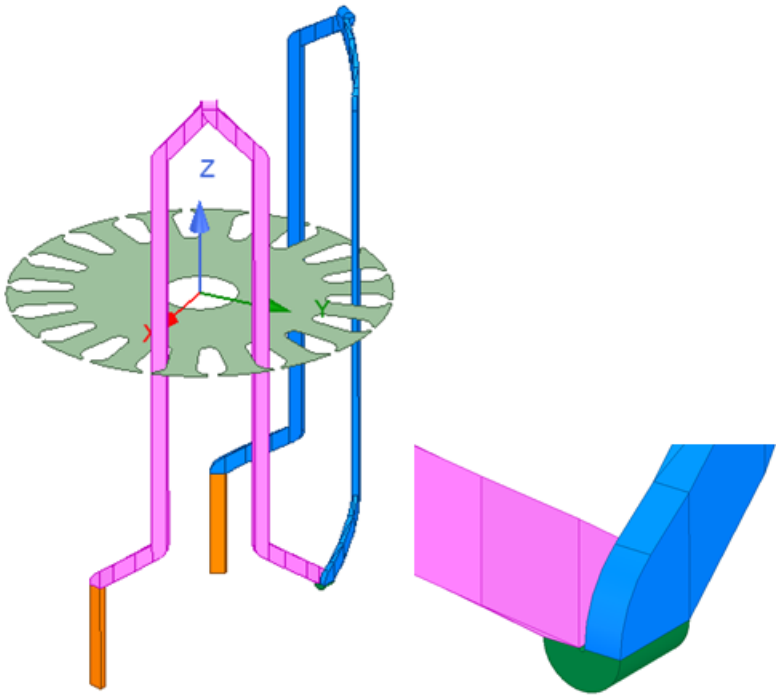
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Two Hairpin Coils with Different Coil Pitch at Different Slot Locations

2. Make a copy of the coil, and change **CoilPitch** to 3, and **LayerSetIdx** to 1. The copied coil is modified to be located at the slot center, as shown by the blue-color coil in the above figure.
3. Rotate the blue-color coil by 100 deg, or 5 slot-angle, the bottom leg-span of the blue-color coil will align with the top leg-span of the pink-color coil, as shown in the figure below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Two Hairpin Coils at Different Slot Locations Connected with End-Tip and Leads

4. Make a copy of the blue-color coil, and change **LayerSetIdx** to 0.5 (for end-tip between layer-set index of 0 and 1) and **InfoCoil** to 4. The copied coil is modified to an end tip, as shown by the green-color object in the above figure.
5. Make a copy of the pink-color coil and change **InfoCoil** to 6, the copied coil is modified to a going lead; and make a copy of the blue-color coil and change **InfoCoil** to 7. The copied coil is modified to a returning lead, as shown by the golden-color objects in the figure.

The two hairpin coils, two leads, and one end-tip can thus be united to become a single connected coil set.

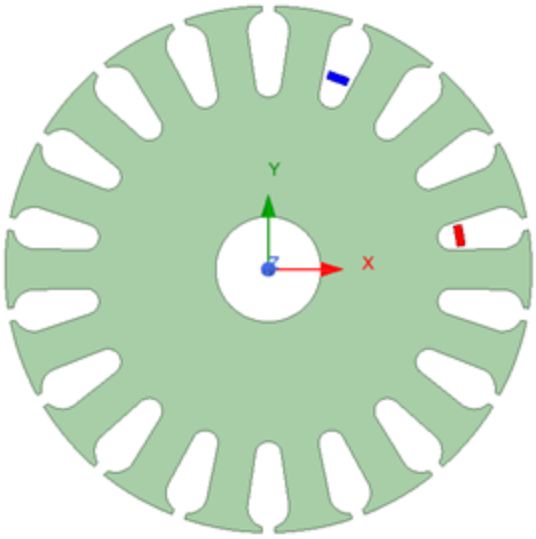
> **Note:** In order for the end-tip and leads to be able to perfectly connect two hairpin coils, the parameter value of **2ndSpanExt** for all components must be the same to keep the same axial span length. If you see a message such as: “HairpinCoil - 2ndSpanExt is modified from xxx to xxx”, you need to increase the value of **2ndSpanExt** for all components to be at least the maximum modified value of all components.

Creating a Terminal for a Hairpin Coil

A terminal is a 2D cross-section of a 3D coil that can be used to assign current or voltage excitations. You can create a *going* terminal by setting **InfoCoil** to 2; or you can create a

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

returning terminal by setting **InfoCoil** to 3. In a connected 3D coil set, you need to create only one terminal, but you need to create both *going* and *returning* terminals for each hairpin coil for a 2D model. *Going* and *returning* terminals are shown below.



***Going* Terminal (red) and *Returning* Terminal (blue) Inside Two Slots**

UDPs for Pole Cores and Coils

UDPs defined for pole cores and coils include

- SalientPoleCore UDP
- DCMCore UDP
- ShadedPoleCore UDP
- UnivMCore UDP
- SRMCore UDP
- SynRMCore UDP
- ClawPoleCore UDP
- PMCore UDP
- IPMCore UDP
- PMDamperCore UDP
- StepMCore UDP
- Square Yoke UDP
- UPMCore UDP
- VPMCore UDP
- VentCore UDP

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SalientPoleCore UDP

The SalientPoleCore UDP is used to create a salient-pole core, a field winding, and a damper for salient-pole synchronous machines. It can also create a coil cross-section used as a terminal for current assignment.

Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaYoke for outer cores
DiaYoke	Core diameter on yoke side, DiaYoke < DiaGap for inner cores
Length	Core length
Skew	Skew angle in core length range
Slots	Number of slots
SlotType	Slot Type: 1 to 4
Hs0	Slot opening height
Hs01	Slot closed bridge height
Hs1	Slot wedge height
Hs2	Slot body height
Bs0	Slot opening width
Bs1	Slot wedge maximum width
Bs2	Slot body bottom width, 0 for parallel teeth.
Rs	Slot body bottom fillet
FilletType	0: a quarter circle; 1: a tangent connection; 2 and 3: arc bottom; 4 and 5: V bottom
SlotPitch	Slot pitch in mechanical degrees, refer to offset point
CenterPitch	Center slot pitch in mechanical degrees, refer to offset point
Poles	Number of poles
PoleType	Pole type: 1 or 2 (not valid for inner cores)
WidthShoe	Pole shoe width
HeightShoe	Pole shoe height at pole center
FilletShoe	Pole shoe tip fillet radius
WidthBody	Pole body width
HeightBody	Pole body height
AirGap2	Second air gap length
Offset	Pole arc offset
Off2_x	The second pole arc offset perpendicular to the pole-center line
Off2_y	The second pole arc offset parallel with the pole-center line

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Property	Description
Bp1/Rp1	Bp1: for pole type 1; or Rp1: shoe fillet radius for pole type 2
Bp2/Rp2	Bp2: for pole type 1; or Rp2: shoe fillet radius for pole type 2
CoilEndExt	One-side coil end extended length
EndRingType	0: whole press board; 1: pole press board; 2: pole ring; 3: whole ring
BarEndExt	One-side damper bar end extended Length; for types 2 and 3 only
RingLength	One-side axial ring length, or conductor press board thickness
RingHeight	Radial ring height, for types 2 and 3 only.
SegAngle	Deviation angle for slot arches (10~30, <10 for true surface)
LenRegion	Region length
InfoCore	0: core; 1: core & coils; 2: coil; 3: damper; 4: terminal1; 5: terminal2; 6:poles; 7: yoke; 100: region

These parameters are used in the following figures. In figures with both gray and red objects, the gray objects are used as references.

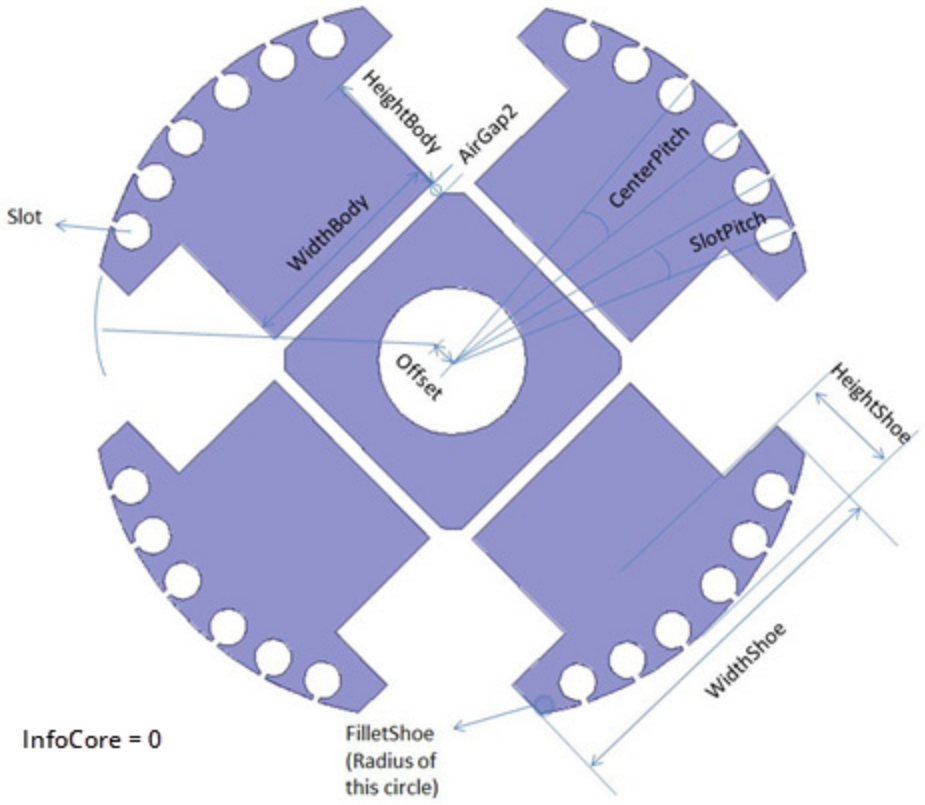
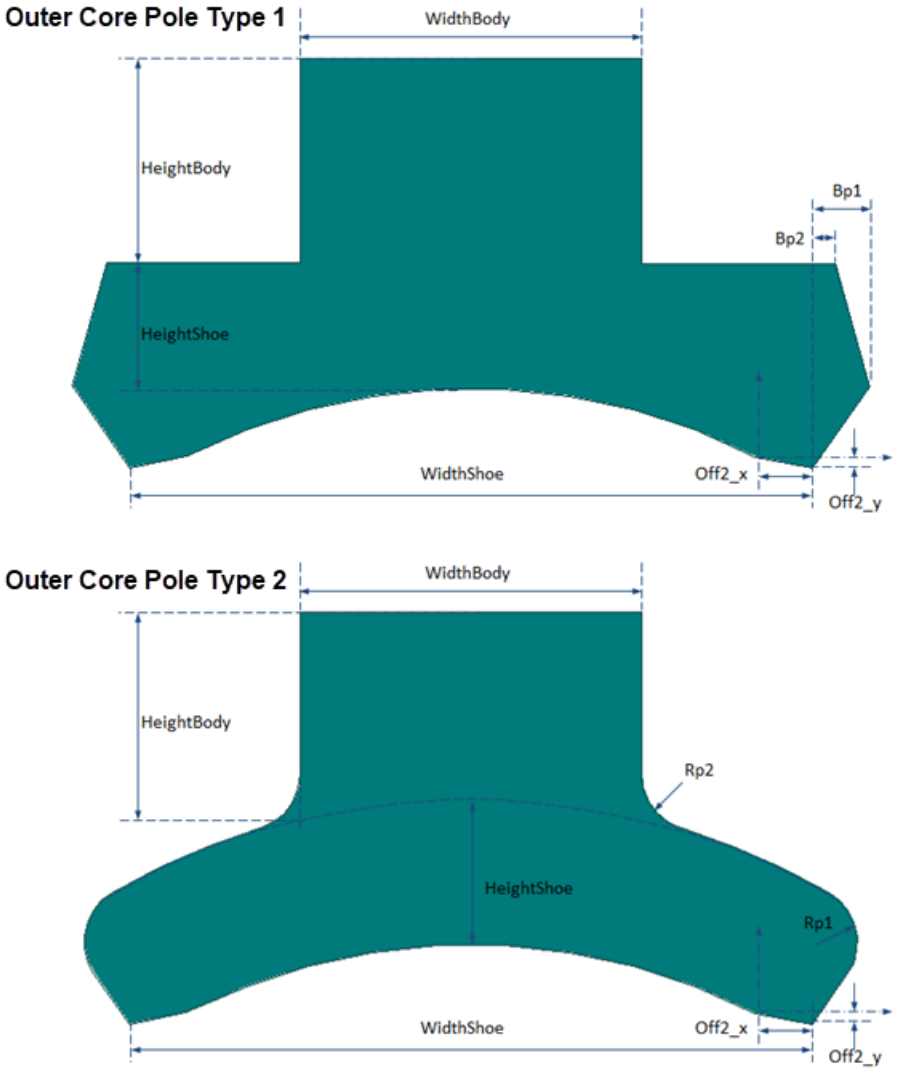


Figure 9-18 Salient Pole Core

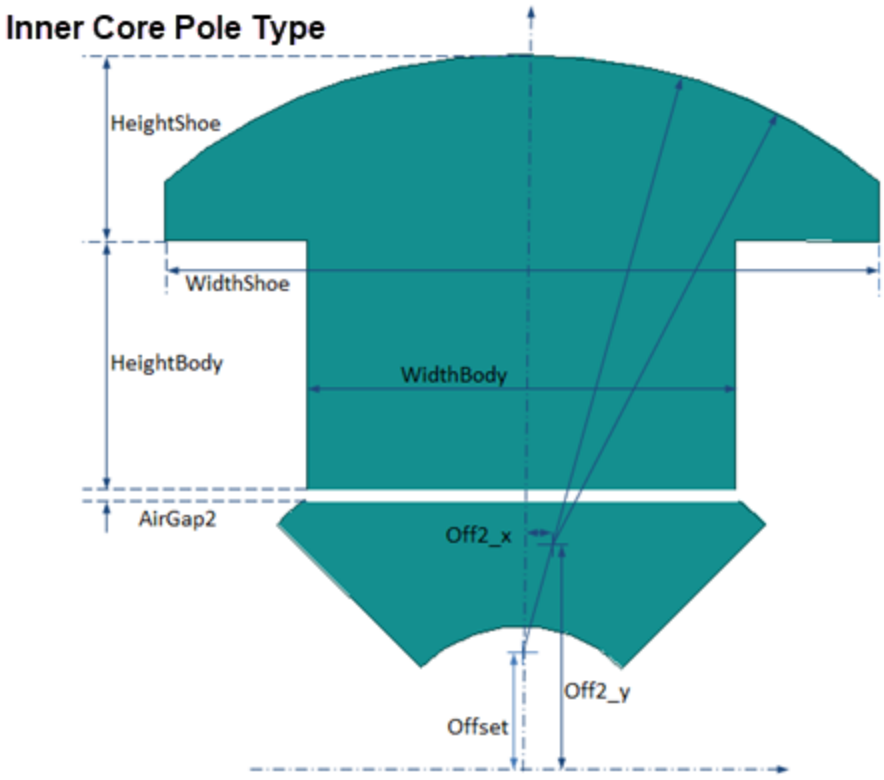
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This UDP supports two pole types for outer cores, as shown below:



But this UDP supports only one pole type for inner cores:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Axial parameters for coils are shown in the following figure:

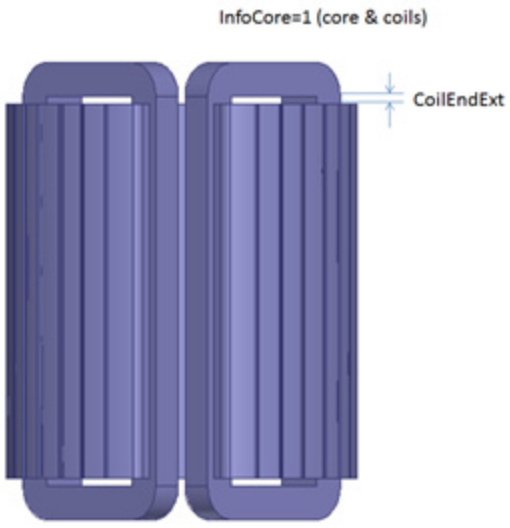


Figure 9-19 3D Salient Pole Core and Coils

Axial parameters for dampers are shown in this next figure:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

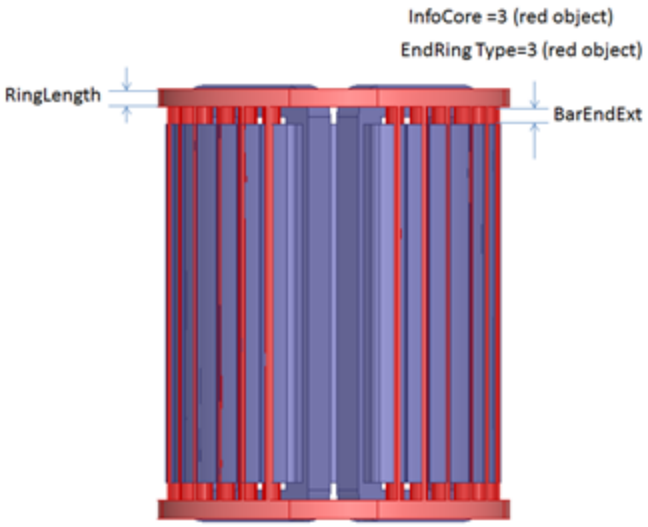
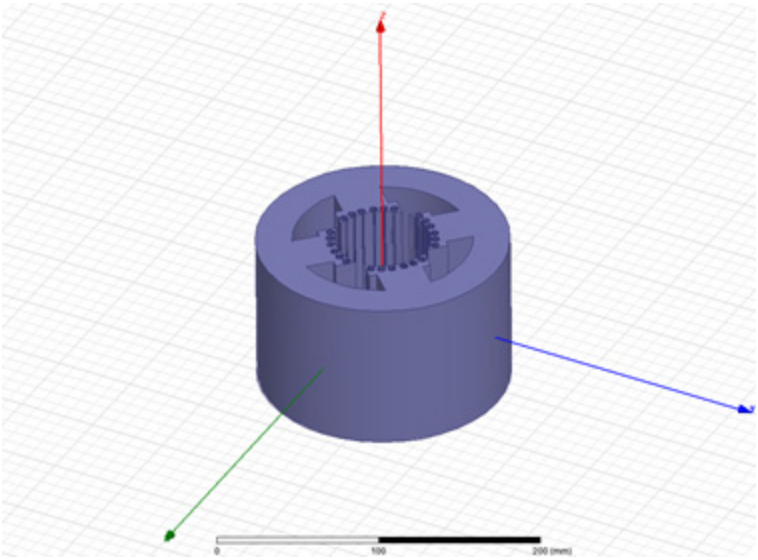


Figure 9-20 SalientPoleCore Damper

Creating an Inner or Outer Salient Pole Core

Set the value of **DiaYoke** and **DiaGap** as **DiaYoke** < **DiaGap** to create inner cores or **DiaGap** < **DiaYoke** for outer cores.

For an example of an inner salient pole core, see SalientPoleCore UDP. An outer salient pole core with **InfoCore** value of 0 is shown in the following figure:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can set the values either when creating the core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Creating a Coil and its Terminals (Salient Pole)

You can create a coil of a salient pole core by manually setting the property of **InfoCoil** to 2. An example is shown below.

Its terminals are created manually by setting the property of **InfoCoil** to 4 or 5 (Terminal1 or Terminal2). The red sheet in the figure shows terminal 1 of the coil.

You can assign values for these properties either when creating a salient pole core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

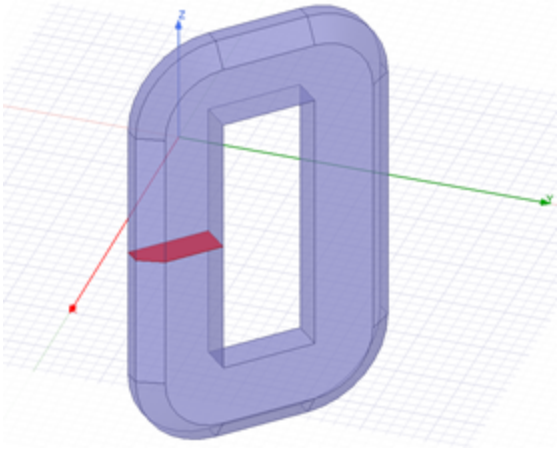


Figure 9-21 Salient Pole Core Coil and its Terminal

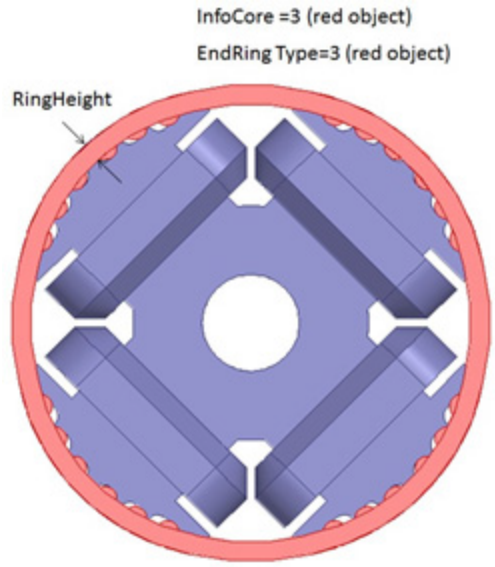
Creating a Damper (Salient Pole)

You can create a damper of a salient pole core by manually setting the property of **InfoCoil** to 3. An example is shown in " SalientPoleCore Damper" on the previous page as a red object.

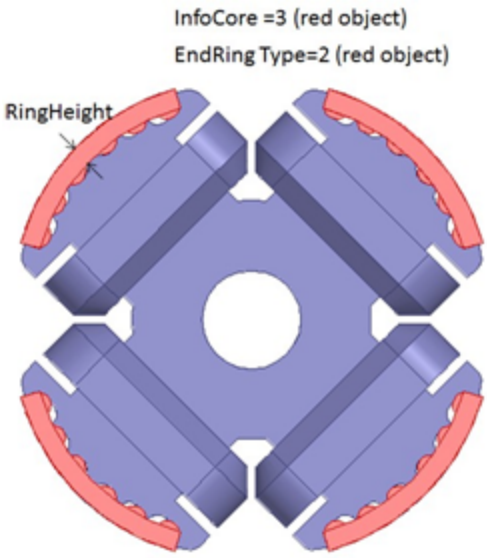
"Damper with EndRing Type set to 3" on the facing page, "Damper with EndRing Type set to 2:" on the facing page, "Damper with EndRing Type set to 1" on page 9-88 , and "Damper with EndRing Type set to 0" on page 9-88 illustrate the shapes of the damper (InfoCore set to 3) with different EndRingType values. Gray objects are used as references.

You can assign the value of these properties either when creating a salient pole core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

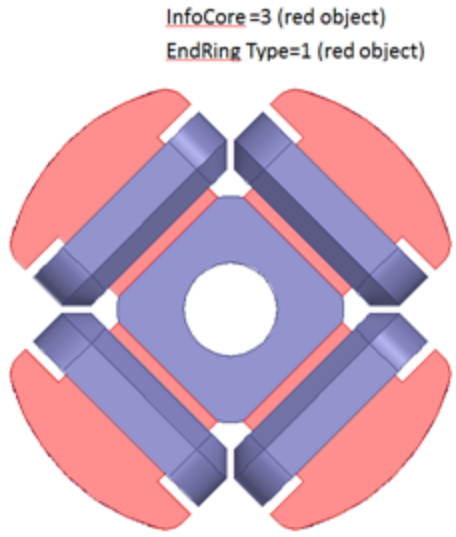


Damper with EndRing Type set to 3

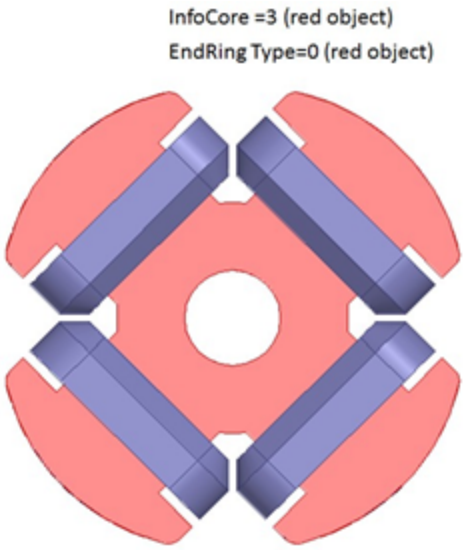


Damper with EndRing Type set to 2:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Damper with EndRing Type set to 1



Damper with EndRing Type set to 0

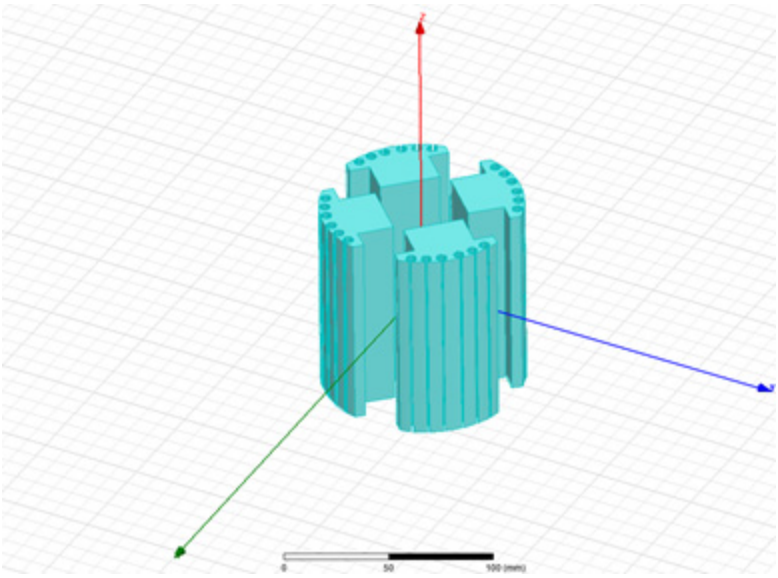
Creating Poles or the Yoke

You can create poles or the yoke of SalientPoleCore UDP through assigning the **InfoCore** parameter to 6 or 7, respectively.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can assign the value of this property either when creating a Salient Pole Core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

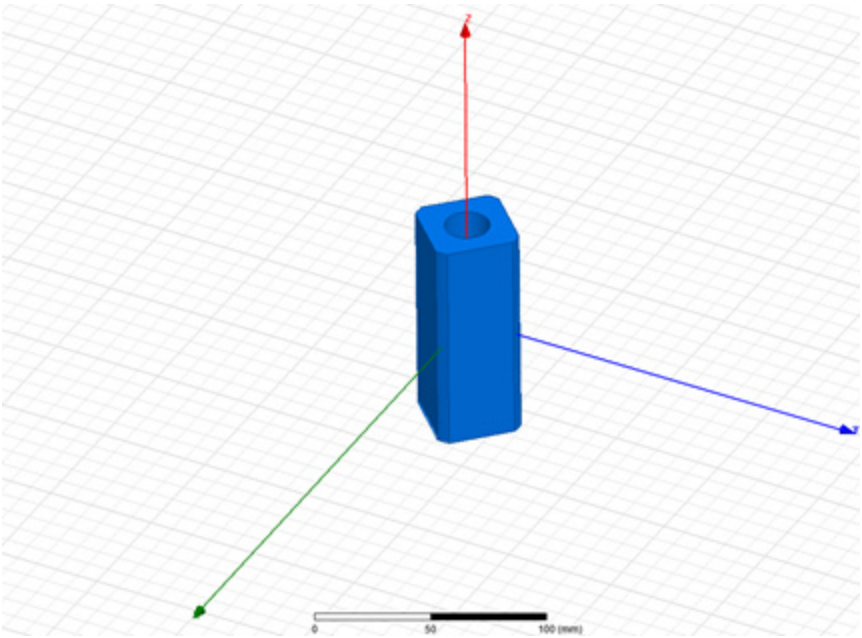
Examples of poles and the yoke are shown in "Poles of Salient Pole Core" below and "Yoke of Salient Pole Core" on the next page, respectively.



Poles of Salient Pole Core

Figure 9-22

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

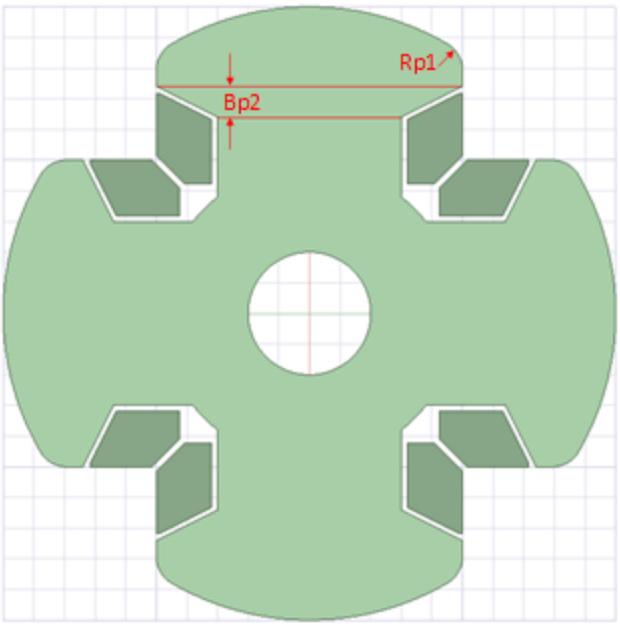


Yoke of Salient Pole Core

Creating Inner Poles with Shoe Slope

For inner-core type-1 pole, parameters **Rp1/Bp1** and **Rp2/Bp2** stand for shoe-tip fillet radius and shoe-slope height, respectively, as shown below, where all coils are included.

Parameters **Rp1** and **Bp2** for inner core type 1 pole:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DCMCore UDP

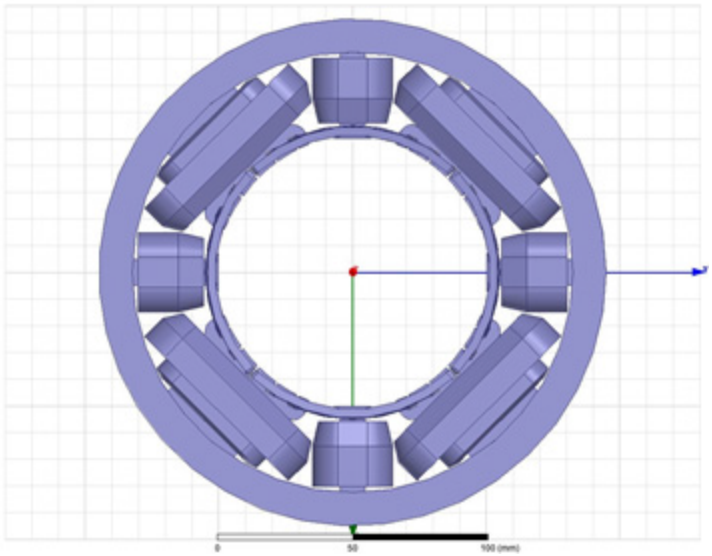
The DCMCore UDP is used to create a stator core with or without a compensating winding, a shunt and/or a series field winding, and a commutating pole with a commutating winding for DC machines. It can also create coil terminals for all windings for current assignment.

Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaYoke for outer cores.
DiaYoke	Core diameter on yoke side, DiaYoke < DiaGap for inner cores.
Length	Core length
Skew	Skew angle in core length range
FrameWidth	Overall width of a racetrack frame
FrameThick	Frame thickness
FrameLength	Frame length
Poles	Number of poles
PoleType	Pole type: 1 to 2
Dmax	Diameter of shoe tip with maximum air gap length
Bp0	Pole arc width with uniform air gap; 0 for eccentric air gap.
Bp1	Total pole width (the width between shoe tips)
Bp2/Rp0	Bp2 (max shoe width for pole type 1), or Rp0 (shoe fillet radius for pole type 2)
Bp3/Rp1	Bp3 (min shoe width for pole type 1), or Rp1 (pole fillet radius for pole type 2)
Hp	Pole body height
Bm	Pole body width
FieldWndgs	Number of field windings, 2 for both series and shunt windings.
EndExt	Coil one-side end extended length
SlotsPerPole	Compensating slots per pole distributed under pole arc surface
Bc0	Opening width of compensating slots
Hc0	Opening height of compensating slots
Bc2	Width of compensating slots
Hc2	Height of compensating slots
CmpEndExt	One-side end extended length of compensating coils
ComPoleWidth	Width of the commutating poles
ComPoleHeight	Height of the commutating poles
ComPoleLength	Length of the commutating poles
ComShoeWidth	Shoe width of the commutating poles

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Property	Description
ComShoeHeight	Shoe height of the commutating poles
ComShoeLength	Shoe length of the commutating poles
ComGap2	Air gap length between commutating poles and the frame
CmtEndExt	One-side end extended length of commutating coils
LenRegion	Region length
InfoCore	0: core & coils; 1: poles; 2: frame; 3: com poles; 4: shunt coil; 5: series coil; 6: com coil; 100: region
Info Term	0: whole coil; 1: terminal1; 2: terminal2

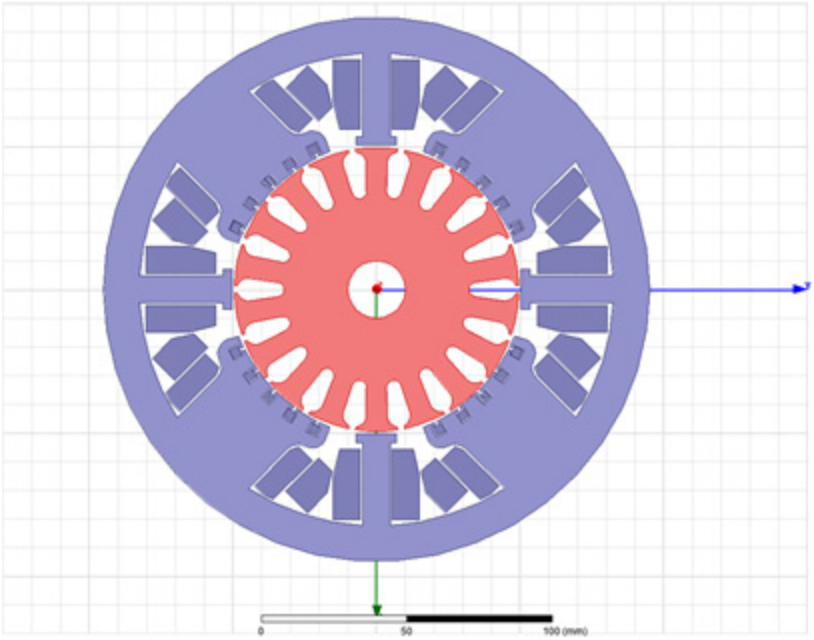
The DC Machine core is complex compared with other UDPs. When creating a DCM core in a Maxwell 3D design, it looks at first like the following figure in the coordinate system window.



Initial DCM Core

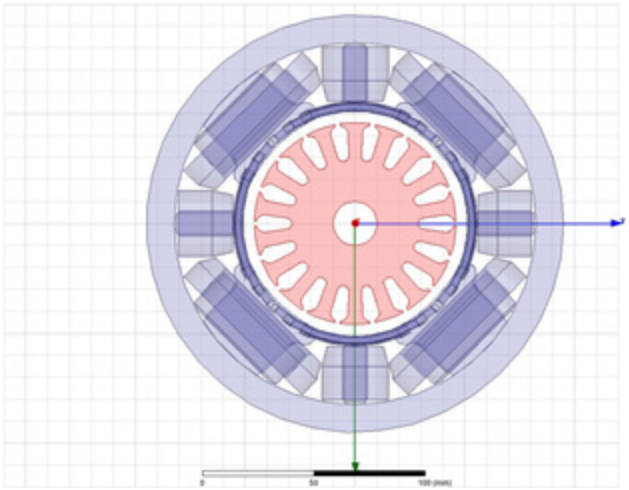
In order to better describe the parameters in this UDP, a slot core was added inside the DCM core and their lengths were set to be 0. Also the **DiaGap** of slot core was changed to 92 to show the air gap between two UDPs. The model now looks like the following example:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



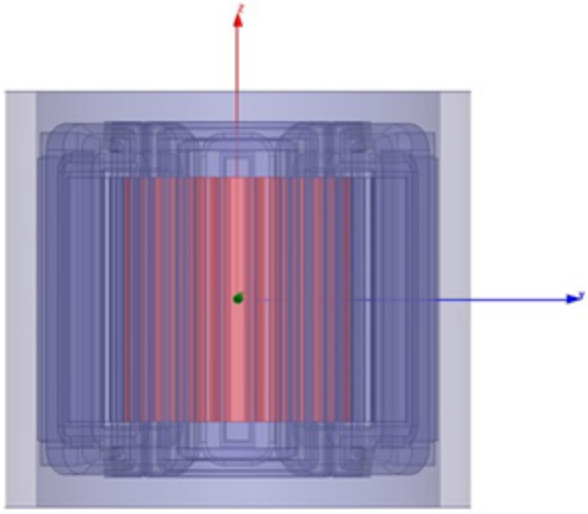
DCM Core Sheet (gray) with Slot Core Sheet (red)

The following figures show two different overviews of the gray, transparent DCM core and the referencing red slot core from different angles.



Transparent DCM core (gray) with Slot Core from Different Angles

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



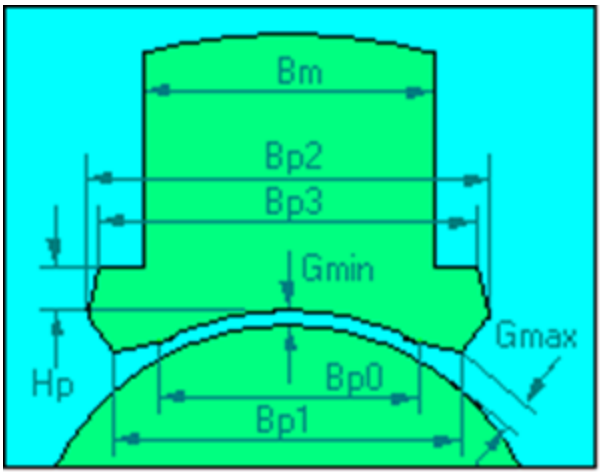
Transparent DCM Core (gray) with Slot Core from XZ View

The following figures show the parameters of a DCM core.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

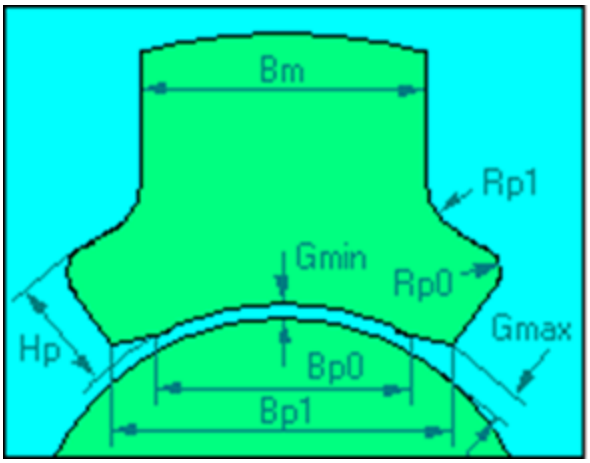
1. A slot core in red is drawn as reference.
2. Because the structure of a DCM core is complicated and there are many parameters, figures are drawn using different **InfoCore** values of DCM core UDP to clearly describe parameters.
3. "Parameters of PoleType 1" below and "Parameters of PoleType 2" on the next page show parameters of two different pole types in the DCM core UDP. In these two figures, **Bp0** is not 0, whereas in " Parameters of DCM Core on 2D Plane" on page 9-97 , **Bp0** is 0, which means the air gap is eccentric.
4. **Dmax**, by definition, is the diameter of shoe tip with maximum air gap length, which means **Dmax**=Drout+2*Gmax. “Drout” means the outer diameter of the rotor (which means **DiaGap** of slot core in this example and is shown in " Parameters of DCM Core on 2D Plane" on page 9-97), and “Gmax” is the maximum air gap between poles and rotor, which is shown in "Parameters of PoleType 1" below and "Parameters of PoleType 2" on the next page.
5. Correspondingly, there is **Dmin** (which is called **DiaGap** in DCM UDP and " Parameters of DCM Core on 2D Plane" on page 9-97) and **Gmin**. Their relationship is **Dmin**=Drout+2*Gmin. They are also shown in"Parameters of PoleType 1" below, "Parameters of PoleType 2" on the next page, and " Parameters of DCM Core on 2D Plane" on page 9-97 .
6. The value of **ComGap2** in " Parameters of DCM Core on 2D Plane" on page 9-97 is 0, whereas it is positive in "Parameters of a Commutating Pole (when ComGap is non-zero)" on page 9-98 . In order to make the value of **ComGap2** larger than 0, make sure to reserve enough space for **ComGap2**. This can be done by reducing the value of **ComPoleHeight** or **ComShoeHeight** or both.



Parameters of **PoleType** 1

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

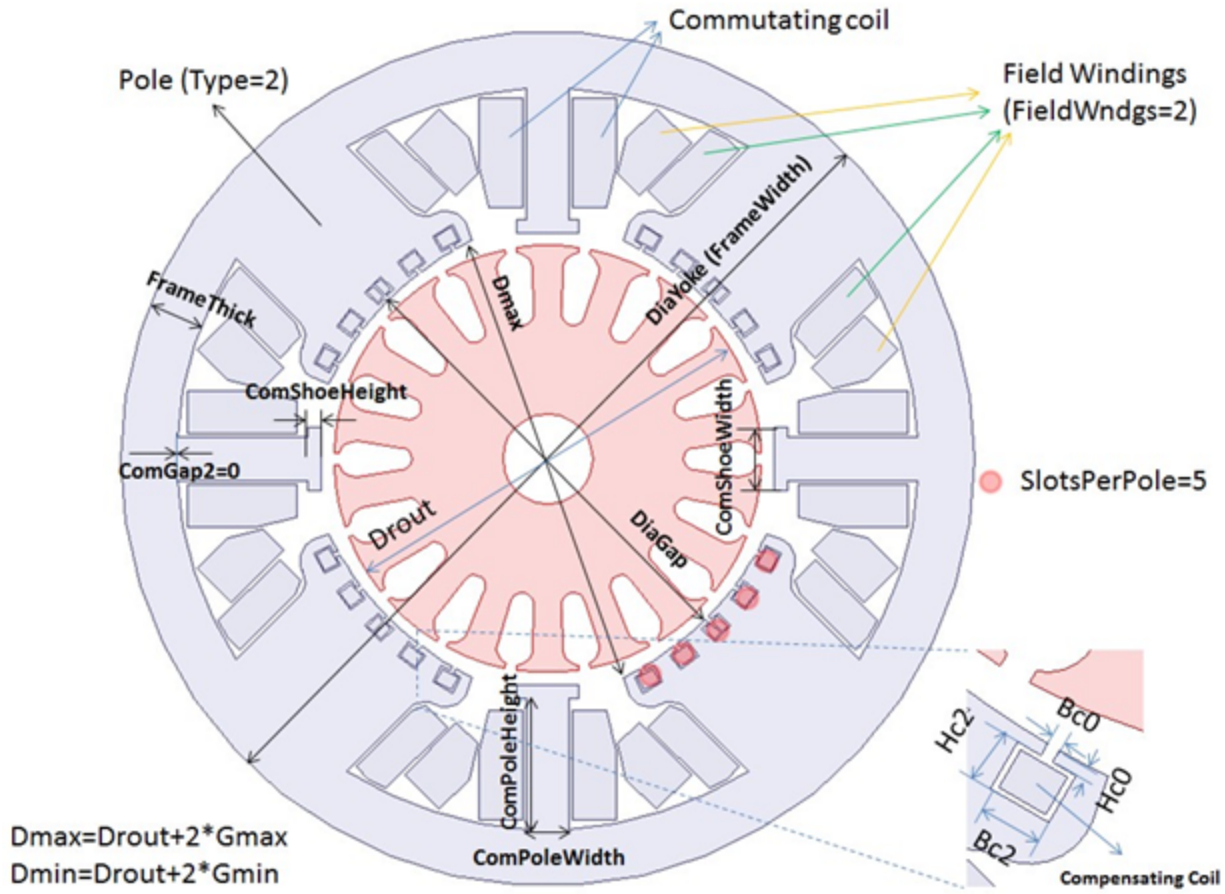
Figure 9-23



Parameters of **PoleType** 2

Figure 9-24

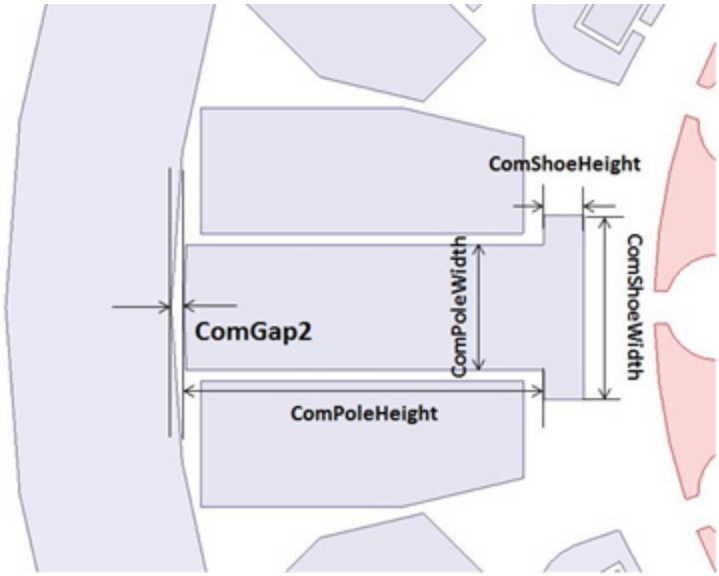
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



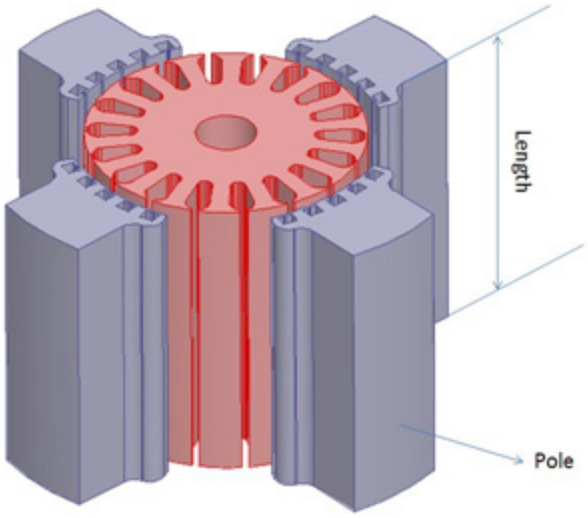
Parameters of DCM Core on 2D Plane

Figure 9-25

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

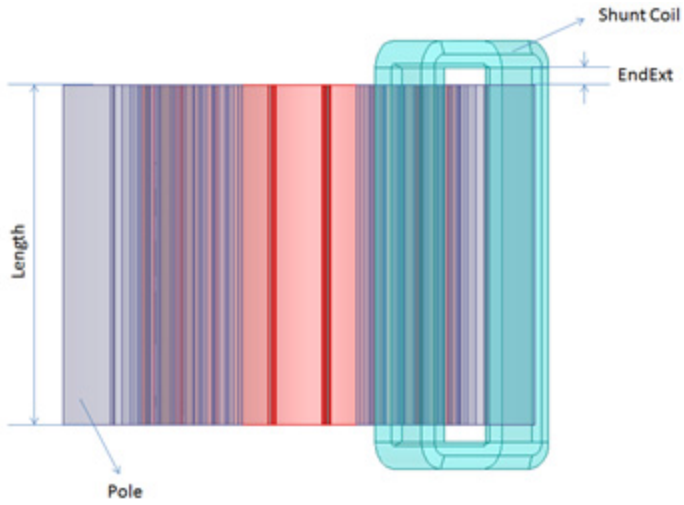


Parameters of a Commutating Pole (when ComGap is non-zero)

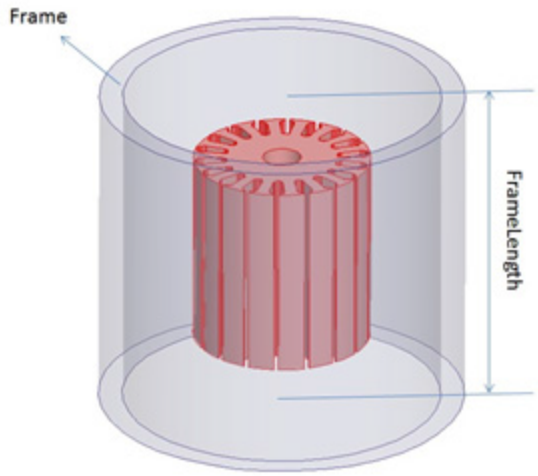


Length of Poles

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

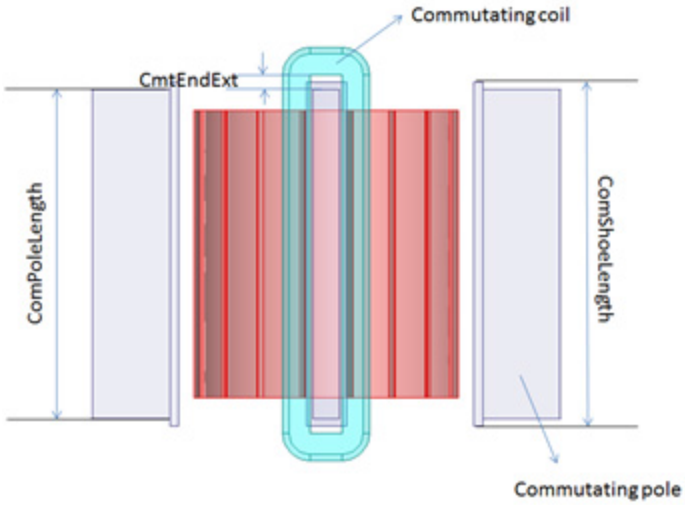


Axial parameters of poles and field windings



Length of Frame

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Axial Parameters of Commutating Poles and Commutating Coils

Creating an Outer DC Machine Core and All Coils

This core is used only as outer core, which means the value of **DiaYoke** is larger than that of **DiaGap**. Also, set both **InfoCore** and **Info Term** values to be 0. You can set the value either when creating the core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

An example of an outer DC machine core is shown in the following figure:

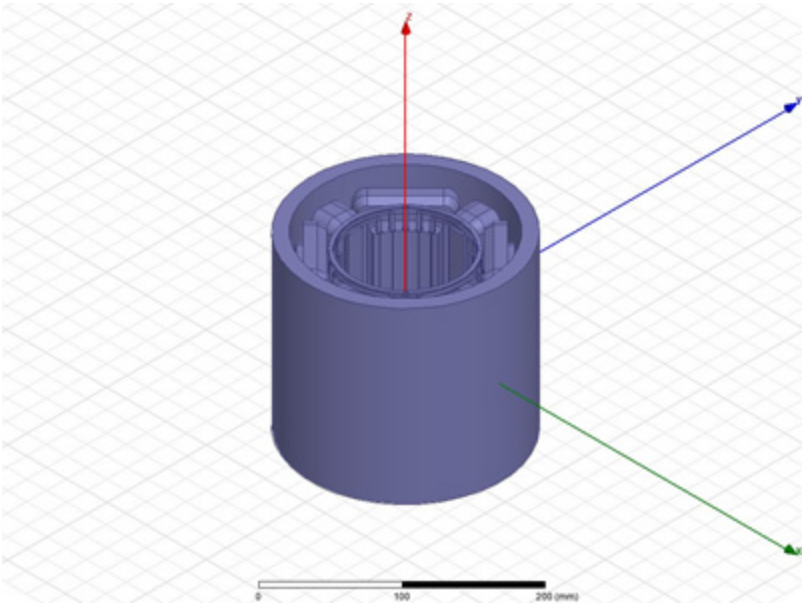


Figure 9-26 Outer DC Machine Core and All Coils

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating Main Poles, Frame and Commutating Poles Separately

To create the main poles of a DC Machine core, manually set the property of **InfoCore** to 1 and **Info Term** to 0. The result is shown as the gray object in " Length of Poles" on page 9-98 and "Axial parameters of poles and field windings" on page 9-99 .

To create the frame of a DC Machine core, manually set the property of **InfoCore** to 2 and **Info Term** to 0. The result is shown as the gray object in "Length of Frame" on page 9-99 .

To create the commutating poles of a DC Machine core, manually set the property of **InfoCore** to 3 and **Info Term** to 0. The result is shown as the gray object in "Axial Parameters of Commutating Poles and Commutating Coils" on the previous page.

You can assign the values of these properties either when creating a DC Machine core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Creating a Shunt Coil and its Terminals

To create a shunt coil of a DC Machine core, manually set the property of **InfoCore** to 4 and **Info Term** to 0. The result is shown as a blue object in "Axial parameters of poles and field windings" on page 9-99 .

To create its terminals, set the property of **InfoCore** to 4 and **Info Term** to 1 or 2 (for terminal1 and terminal2, respectively).

You can assign the values of these properties either when creating a DC Machine core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Creating a Series Coil and its Terminals

To create a series coil of a DC Machine core, manually set the property of **InfoCore** to 5 and **Info Term** to 0. An example is shown in the following figure.

To create its terminals, set the property of **InfoCore** to 5 and **Info Term** to 1 or 2 (for terminal1 and terminal2, respectively).

You can assign the values of these properties either when creating a DC Machine core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

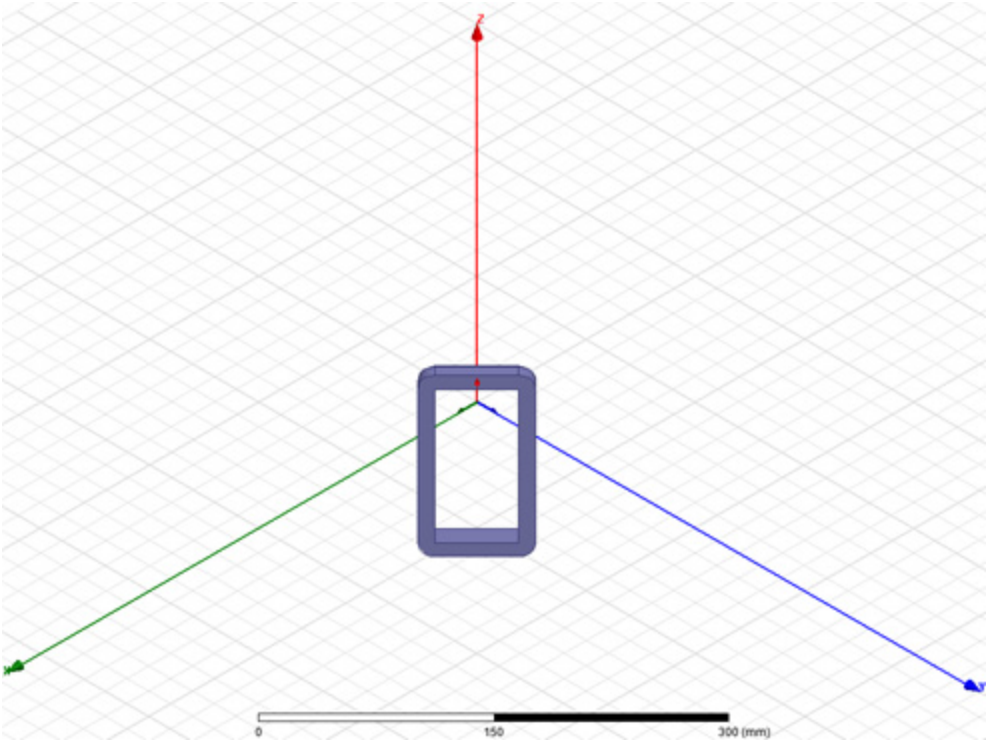


Figure 9-27 A Series Coil of DCM Core

Creating a Commutating Coil and its Terminals

To create a commutating coil of a DC Machine core, manually set the property of **InfoCore** to 6 and **Info Term** to 0. The result is shown as a blue object in "Axial Parameters of Commutating Poles and Commutating Coils" on page 9-100 .

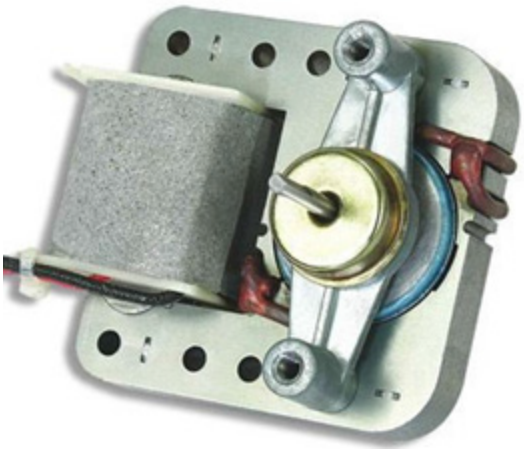
To create its terminals, set the property of **InfoCore** to 6 and **Info Term** to 1 or 2 (for terminal1 and terminal2, respectively).

You can assign the values of these properties either when creating a DC Machine core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

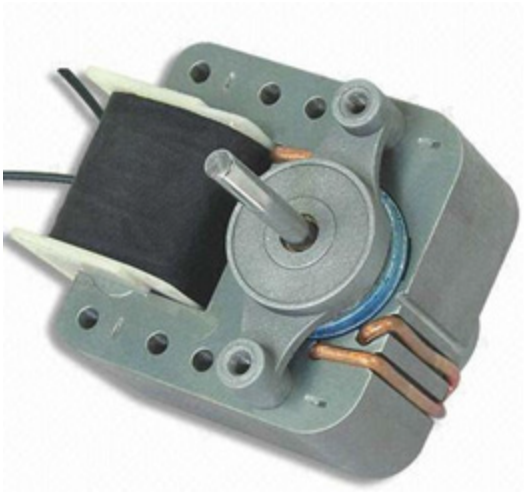
ShadedPoleCore2 UDP

The ShadedPoleCore2 UDP is used to create the stator core for shaded-pole induction motors.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Shaded Pole Induction Motor Example 1



Shaded Pole Induction Motor Example 2

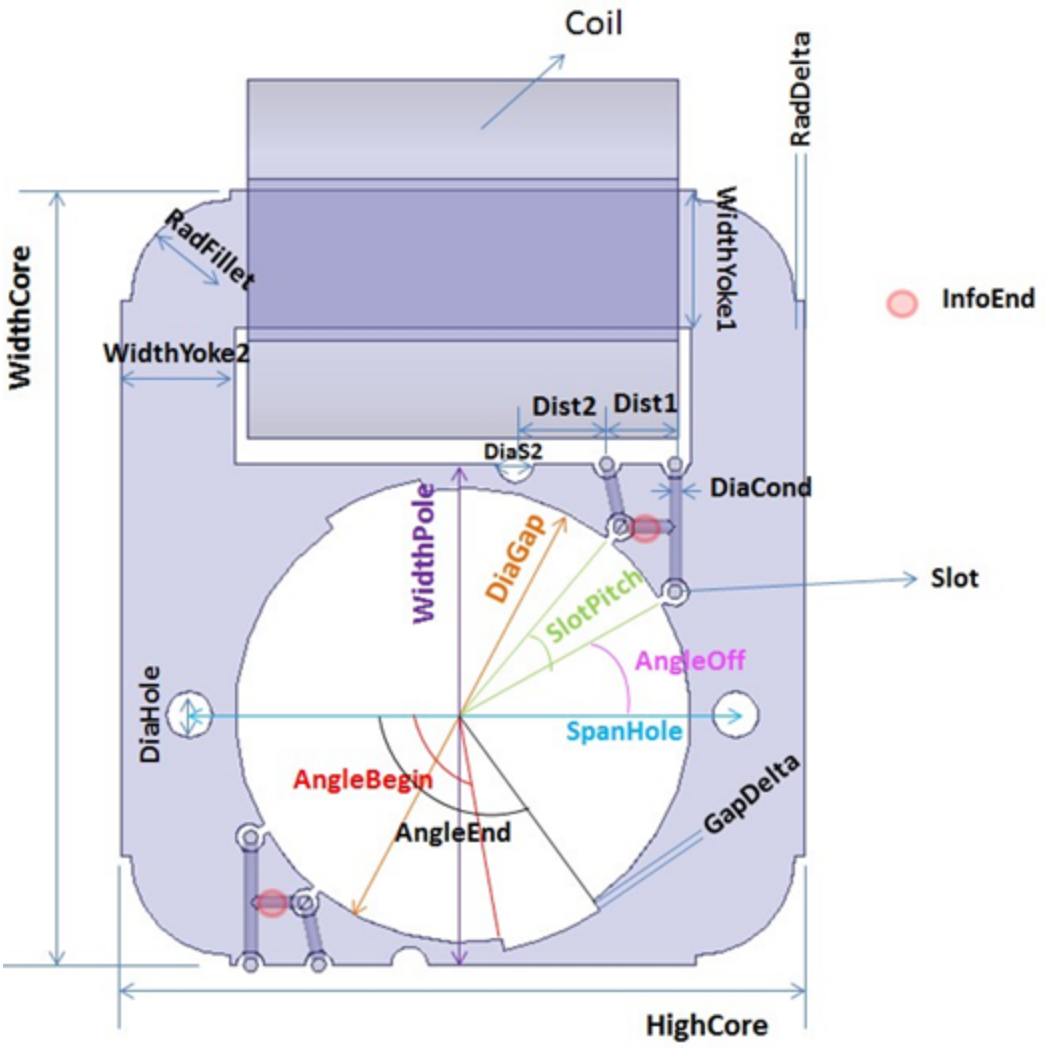
Property	Description
DiaGap	Core diameter on gap side, or inner diameter
HighCore	Core height in d-axis direction
WidthCore	Core width in q-axis direction
WidthPole	WidthPole = DiaGap + 2 * (pole bridge thickness)
Length	Core length in the axial direction
WidthYoke1	Width of the winding-core yoke

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Property	Description
WidthYoke2	Width of the parallel yokes
Slots	Number of slots per pole (1 or 2)
Hs0	Slot opening height
Bs0	Slot opening width
Bs1	Slot diameter
DiaCond	Diameter of the short-circuit solid conductor
AngleOff	Angle offset in mechanical degrees from the first slot to d-axis
SlotPitch	Slot pitch in mechanical degrees for two slots
Dist1	Outer-slot center-to-center distance when Slots = 2
DiaS2	Additional outer-slot diameter
Dist2	Additional outer-slot distance
InfoEnd	0: Separated end connectors; 1: Jointed end connectors
GapDelta	Maximum gap - minimum gap
AngleBegin	Begin angle of maximum gap in mechanical degrees
AngleEnd	End angle of maximum gap in mechanical degrees
RadFillet	Radius of outer-corner fillet
RadDelta	Step depth in outer-corner fillet
DiaHole	Diameter of holes for shaft bearing covers
SpanHole	Hole span in d-axis direction
LenRegion	Region length
InfoCore	0: Core; 1: All; 2: Coil; 3: Cond1; 4: Cond2; 5: Hole filler; 100: Region
InfoTerm	0: Whole coil; 1: Terminal1; 2: Terminal2

These parameters are demonstrated in the following figure (**InfoCore** set to 1, **Info Term** set to 0):

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

- The number of parameters is large in this UDP, so different colors are used to indicate different parameters in the stator bore (area of the circle of **DiaGap**). Each parameter is shown with an arc/line and text in the same color.
- For parameters such as **Length** and parameters related to slots, refer to SlotCore UDP parameters.

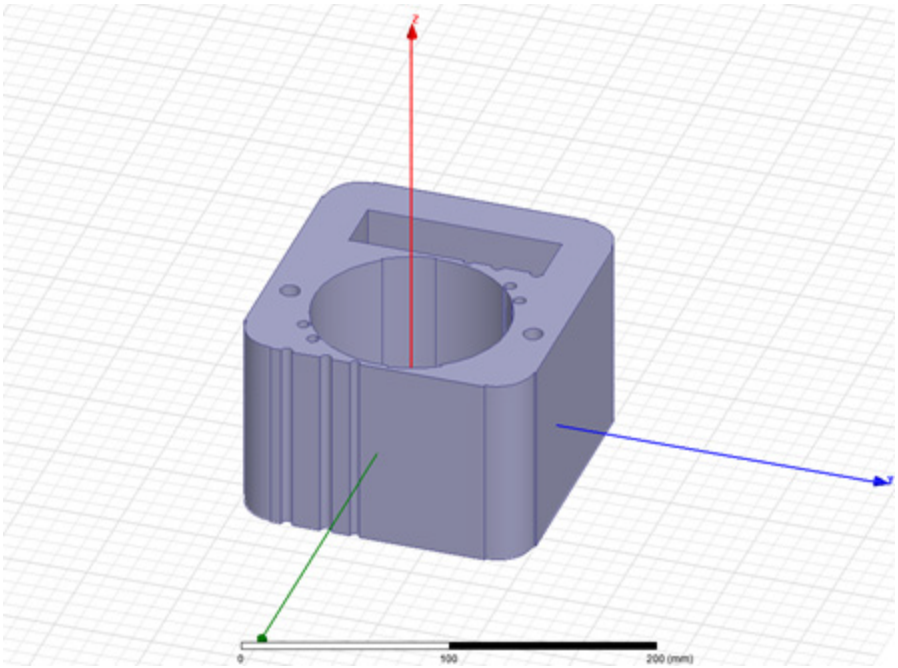
Creating an Outer Shaded Pole Core

This core is used only as an outer core. There is no **DiaYoke** parameter in this UDP, so to create an outer shaded pole core, set both **InfoCore** and **Info Term** values to be 0. You can set the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

values either when creating the core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

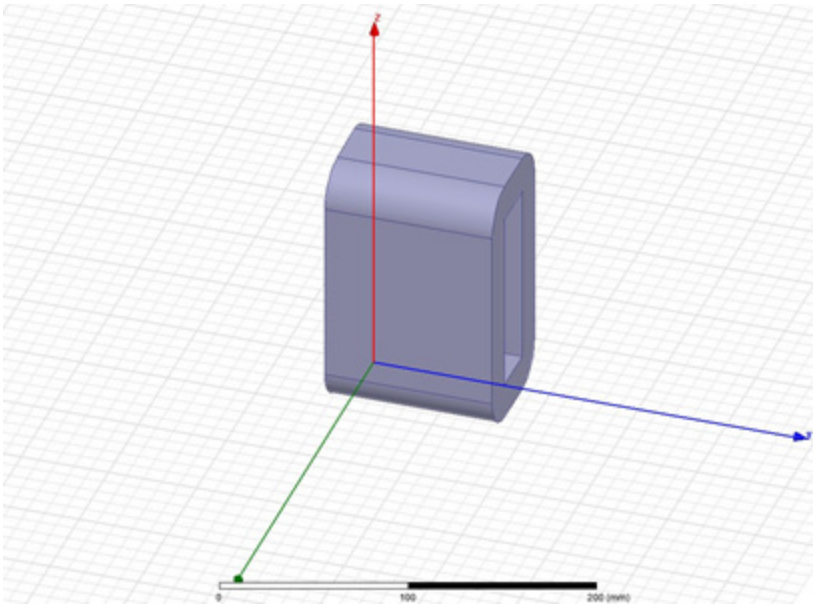
An example of the core for a ShadedPoleCore2 is shown in the following figure:



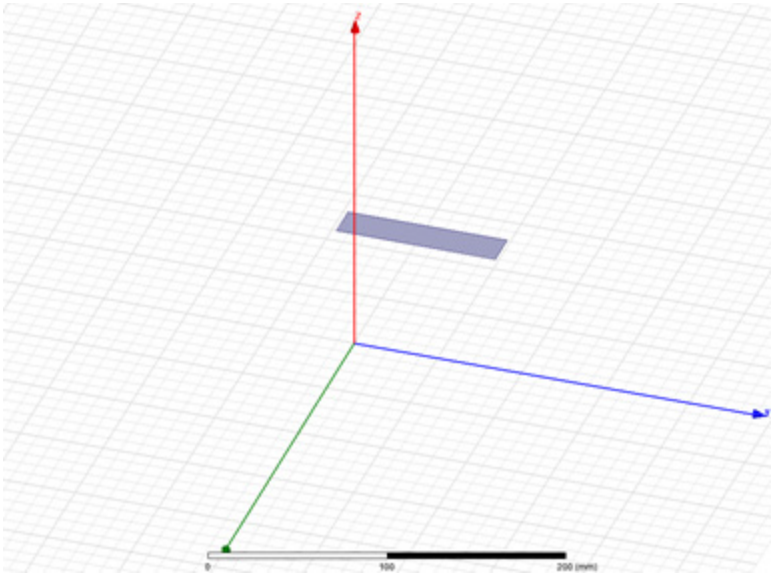
Creating a Coil and its Terminals (Shaded Pole Core)

A coil of a shaded pole core is created by manually setting the property of **InfoCore** to 2 and **Info Term** to 0. The following figure is an example of the coil for a ShadedPoleCore2 UDP:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can create its terminals by setting the property of **InfoCore** to 2 and **Info Term** to 1 or 2 (for terminal1 and terminal2 respectively). The following figure is an example of a terminal:

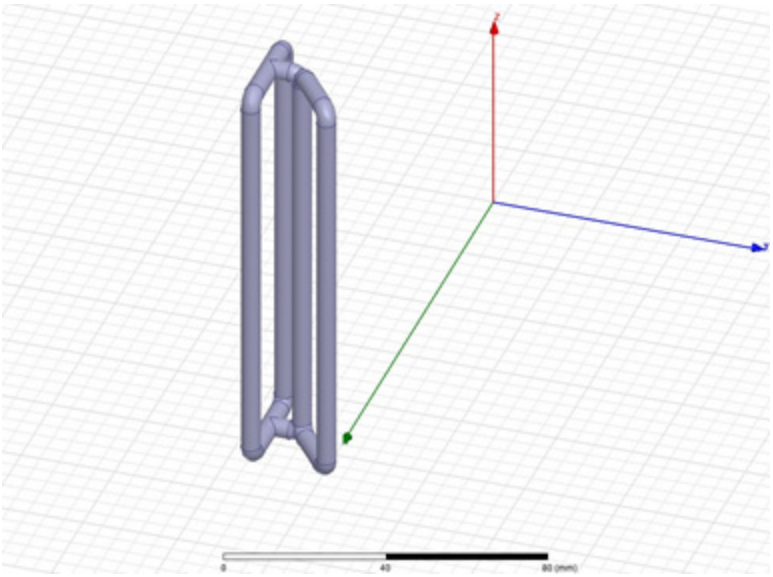


You can assign the values of these properties either when creating a shaded-pole core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating a Jointed or a Separated Shaded Conductor and Its Terminals

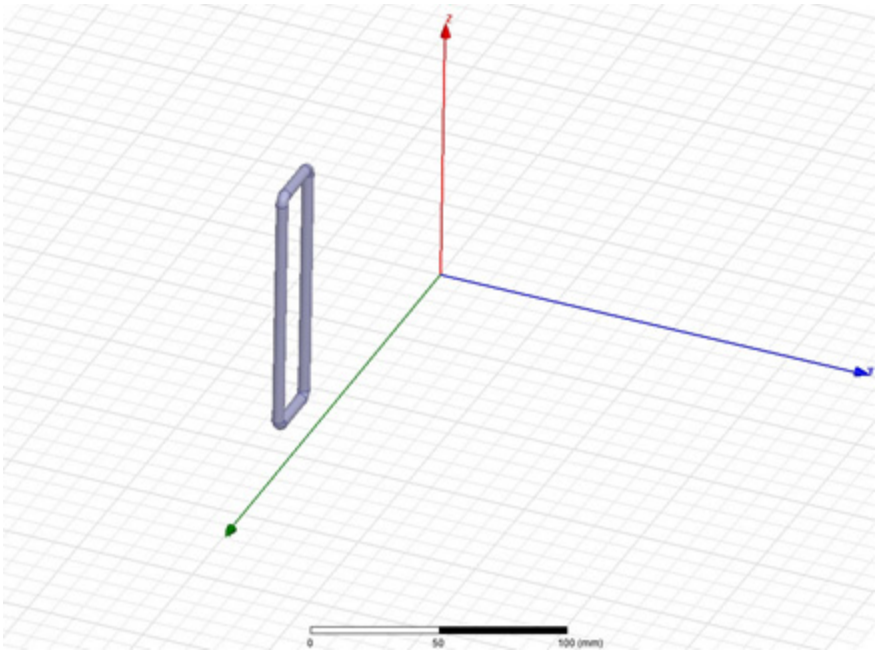
To create a jointed conductor, manually set the parameter **InfoCore** to 3 and **Info Term** to 0. At the same time, set **InfoEnd** to 1 to make the conductor connected. An example of a jointed conductor is shown in the following figure.



A Jointed Shaded Conductor for a ShadedPole Core

To create a separated conductor, change **InfoEnd** to 0. Set **InfoCore** to 3 or 4 to create conductor 1 or 2, respectively. **Info Term** remains 0. The following figure provides an example of the separated conductor.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A Separated Shaded Conductor for a ShadedPole Core

To create terminals of a conductor, change the **Info Term** to 1 or 2 for terminal1 or terminal2, respectively. At the same time, **InfoCore** should be 3 or 4. " Terminal of Conductor for a ShadedPole Core" below shows the results of setting the parameters.

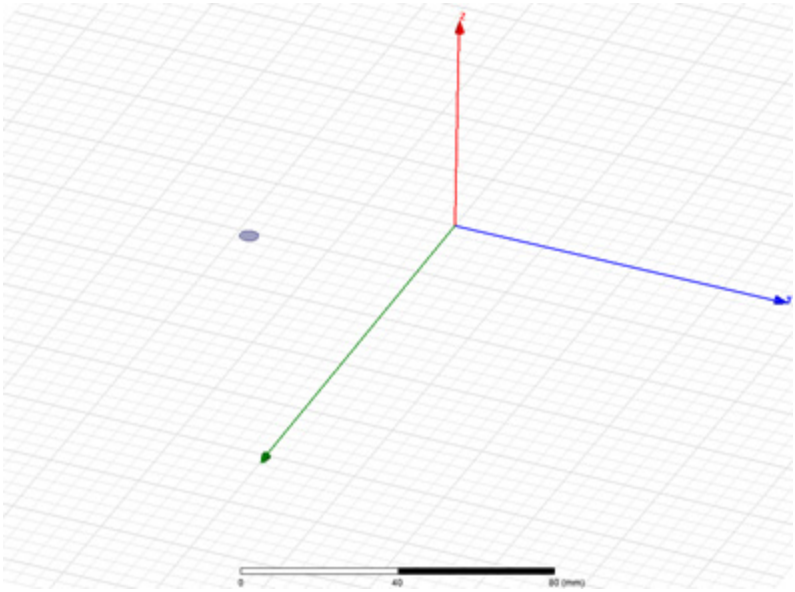


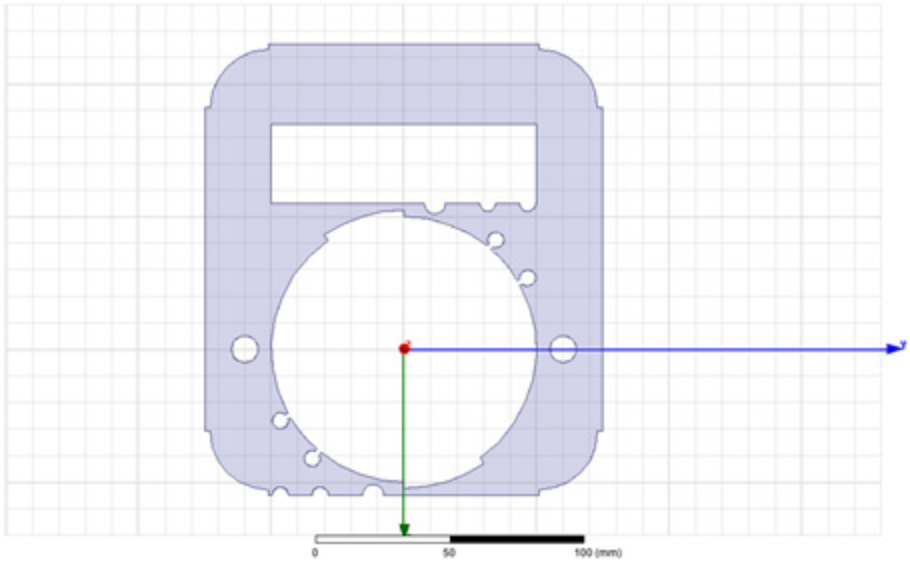
Figure 9-28 Terminal of Conductor for a ShadedPole Core

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating a Core with Poly Holes

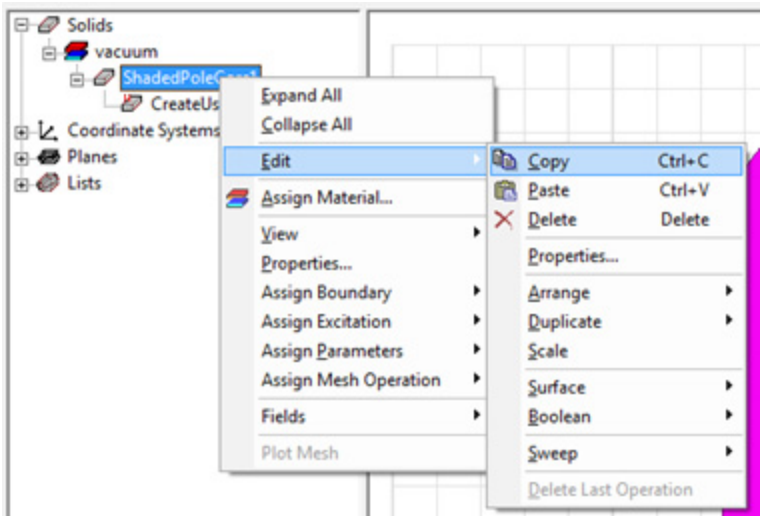
Normally, a Shaded Pole Core UDP has only one pair of holes. In order to create a poly-hole Pole Core UDP, you need to use subtraction and move commands. Detailed steps are as follows.

1. Create a core as described in Creating an Outer Shaded-Pole Core.



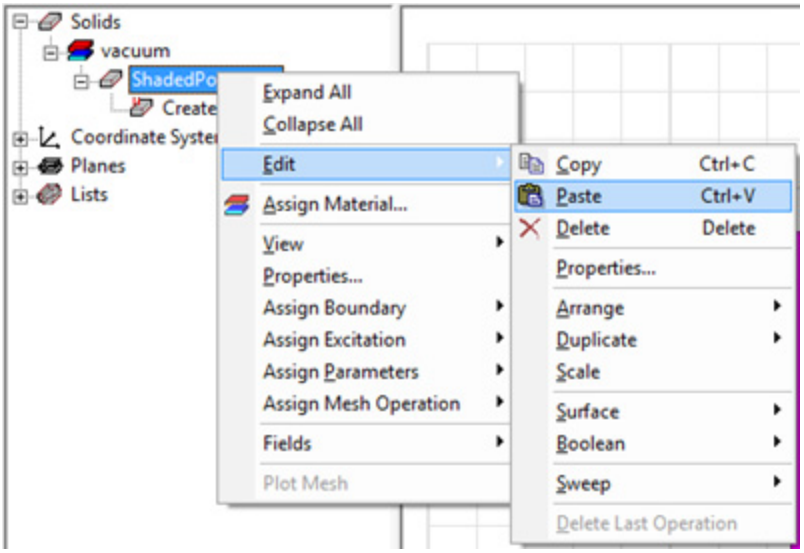
Original ShadedPoleCore

2. Make a copy of the current core.
3. Select the desired inner core in the history tree, and right-click to choose **Edit** > **Copy**.



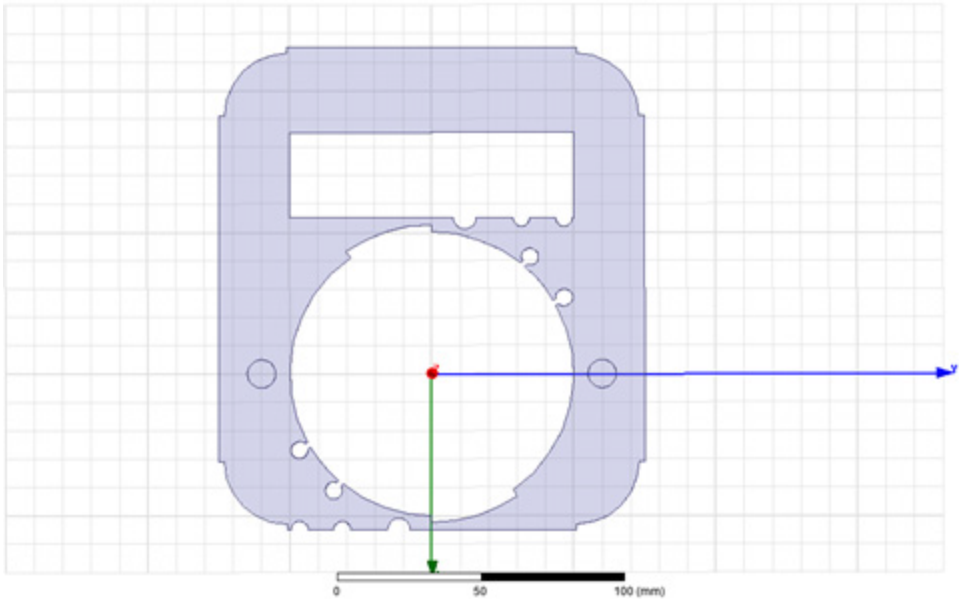
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Then right-click the same object in the history tree to choose **Edit** > **Paste**. This process is shown in the following figure:



5. Change the new object to a Hole filler.

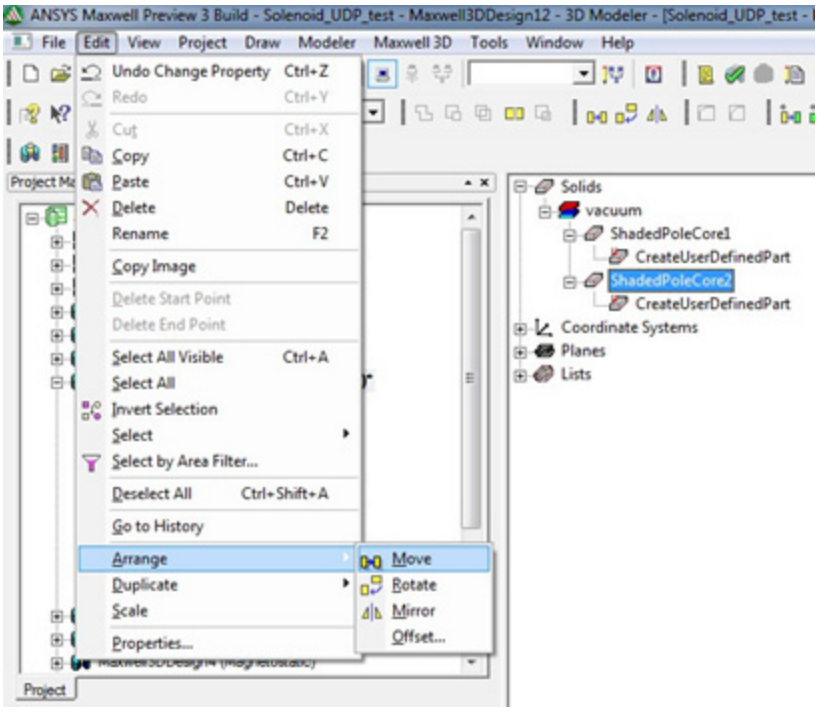
   Click on **CreateUserDefinedPart** of ShadedPoleCore2 object, and set **InfoCore** property to **5** in the pop-up window. The objects in the coordinate system window are shown in the following figure:



6. Move the hole filler to another position for subtraction.
7. Select the hole filler (ShadedPoleCore2) in the history tree.

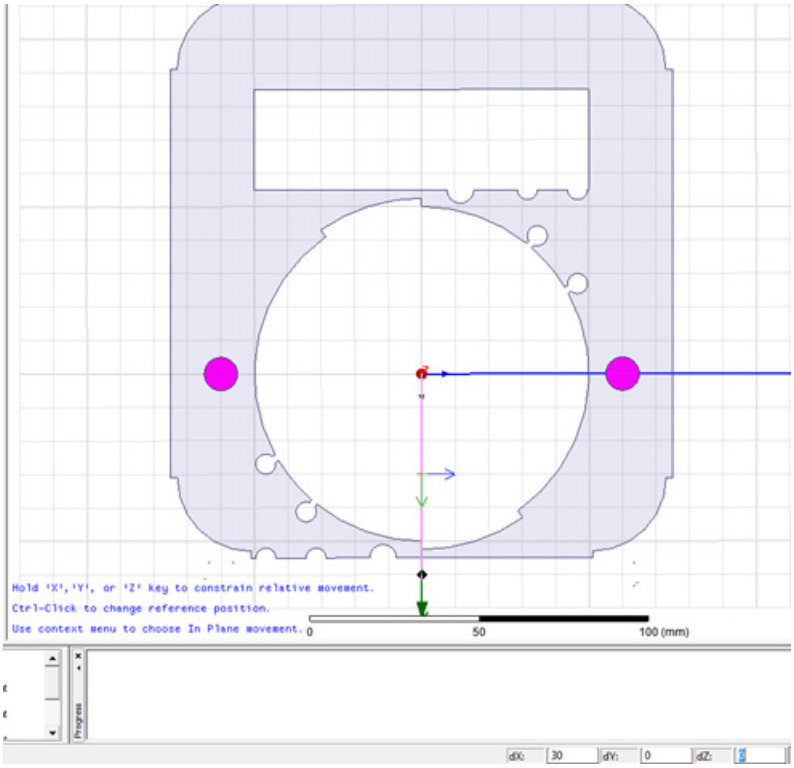
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Click **Edit** > **Arrange** > **Move** from the menu. The modeler switches to move mode and prompts you for a reference point in the status bar.



9. Enter a reference point by clicking at the origin. Another option is to enter **(0, 0, 0)** in the keyboard entry area of the status bar, and press **Enter**.
10. Enter a target point along the Z-axis by clicking the mouse along the axis. Another option is to enter **(30, 0, 0)** in the keyboard entry area, and press **Enter**, which is shown in the following figure.

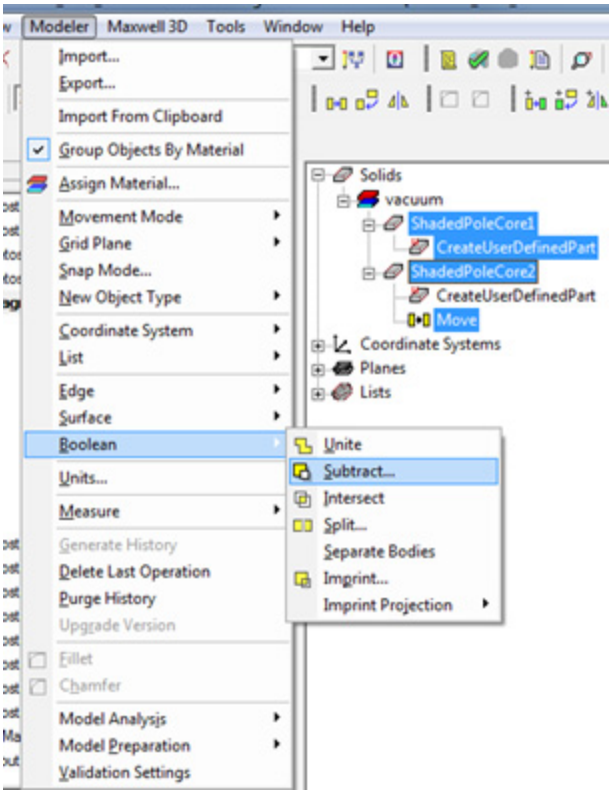
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



11. Choose the core (ShadedPoleCore1) and the hole filler (ShadedPoleCore2) to make a subtraction.
12. Select both the core and hole filler by pressing **Ctrl** on the keyboard.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13. Click **Modeler** > **Boolean** > **Subtract** to subtract the two selected objects.



14. In the **Subtract** dialog box, put core (ShadedPoleCore1) in the **Blank Parts** and hole filler (ShadedPoleCore2) in **Tool Parts**.

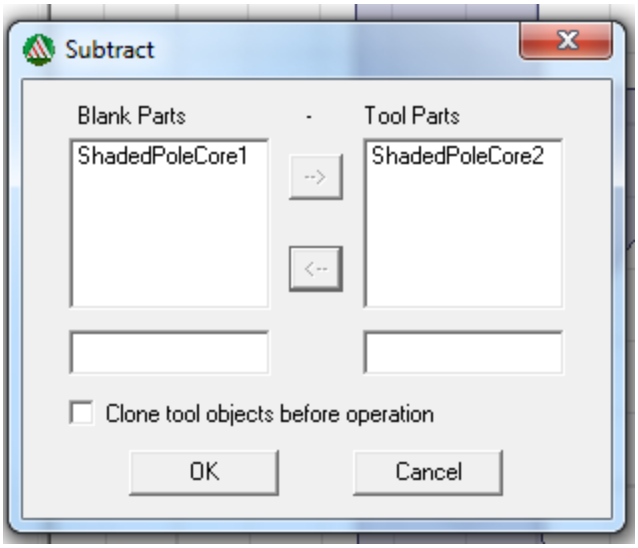
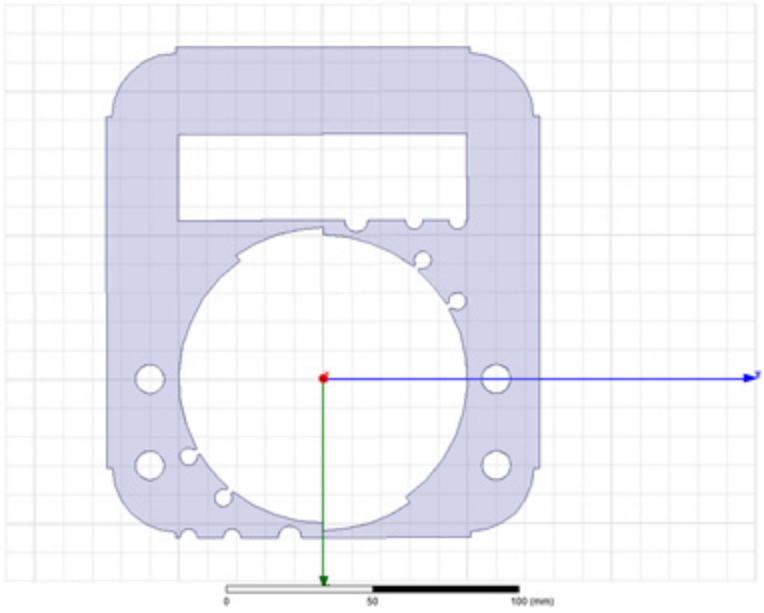


Figure 9-29 Select the Blank Parts and Tool Parts

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15. Click **OK** to finish subtraction. The core look likes the following figure:



Another option is to copy and paste multiple cores, change them to hole fillers and move them to different positions to create a core with more than 2 pairs of holes. The following figure shows a sample hole:

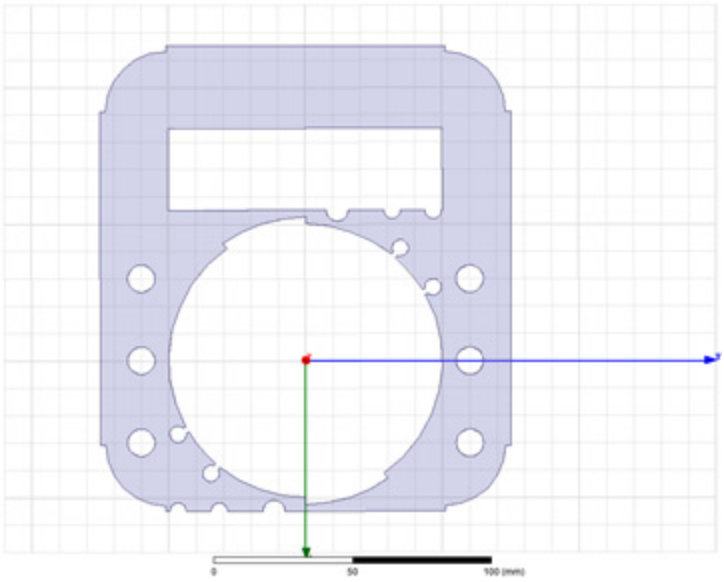


Figure 9-30 ShadedPoleCore with Multiple Holes

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UnivMCore UDP

The UnivMCore UDP is used to create the stator core and the field winding of a universal motor. It can also create a coil terminal for current assignment.

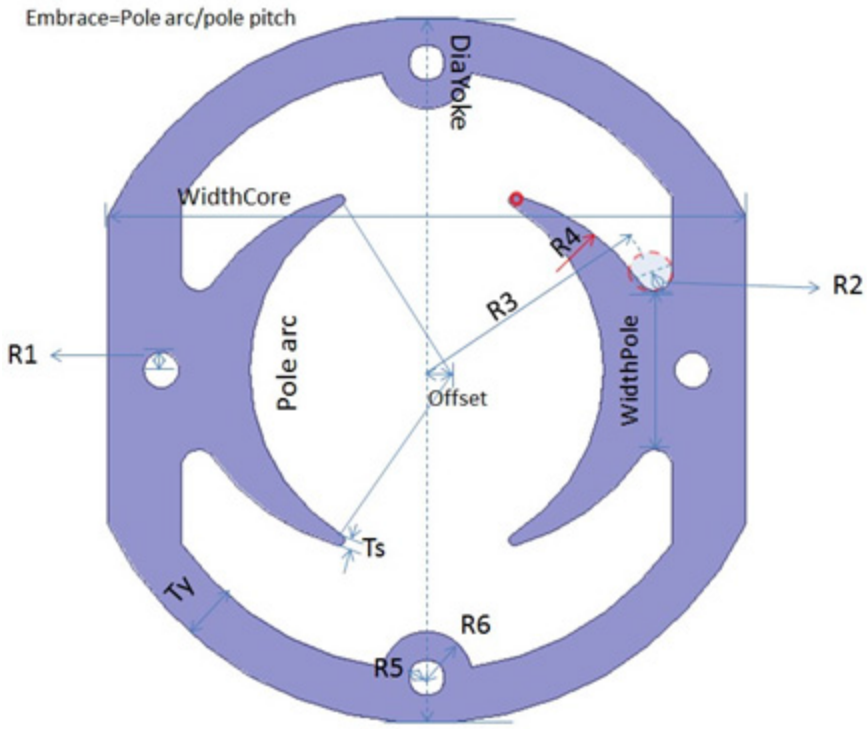
Property	Description
DiaGap	Core diameter on gap side, or inner diameter
DiaYoke	Core diameter on yoke side, or outer diameter
Length	Core length
Skew	Skew angle in core length range
Poles	Number of poles
PoleType	Pole type: 1 to 2
WidthCore	Overall width of a racetrack core
Embrace	Pole embrace (the ratio of pole arc to pole pitch)
Offset	Pole arc offset
WidthPole	Minimum width at pole root
Ty	Yoke thickness
Ts	Shoe-tip thickness
R1	Radius of the screw holes in pole center
R2	Radius of the side fillet arc at pole root
R3	Radius of the side fillet arc center layout circle
R4	Radius of the shoe connecting arc, 0 for auto-design
R5	Inner radius of the screw holes in between two poles
R6	Outer radius of the screw holes in between two poles.
EndExt	Coil one-side end extended length
LenRegion	Region length
InfoCore	0: core; 1: coils; 2: one coil; 3: terminal1; 4: terminal2; 100: region

These parameters are used in the following figures:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- In figures with both gray and red objects, gray objects are used as references.
- Universal motor core is used as outer core, which means the value of DiaYoke is larger than that of DiaGap.
- Although R4 is drawn in "Parameters in UnivM Core" below, it is not easy to be understood. It is the radius of a circle which is externally tangential to the circle of radius R2 and internally tangential to the circle of diameter of Ts on the same side of a pole. R4 and the two circles that it is tangential to are marked in red. When the value of R4 is 0, the length of it is automatically designed.



Parameters in UnivM Core

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

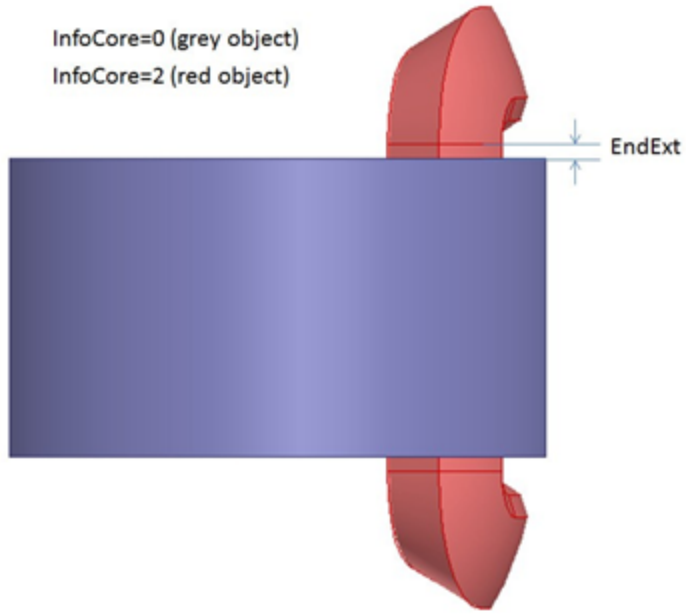


Figure 9-31 Two UnivM Cores with Different InfoCore Values

Creating an Outer Universal Motor Core

This core is used only as an outer core. To create an outer universal motor core, set the value of **DiaYoke** and **DiaGap** as **DiaYoke** > **DiaGap**. Set the **InfoCore** value to 0.

You can set the values either when creating the core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

The following figure shows the results:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

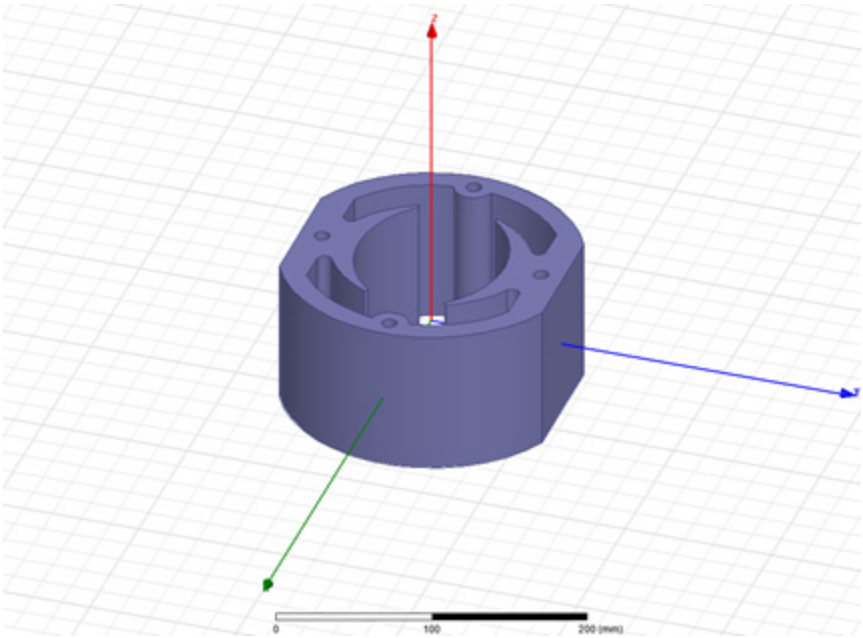


Figure 9-32 Outer Universal-Motor Core

Creating All Coils (UnivMCore)

To create all coils of a universal-motor core, manually set the property of **InfoCoil** to 1.

You can assign the value of this property either when creating a universal-motor core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

The following figure gives an example of all coils for this UDP:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

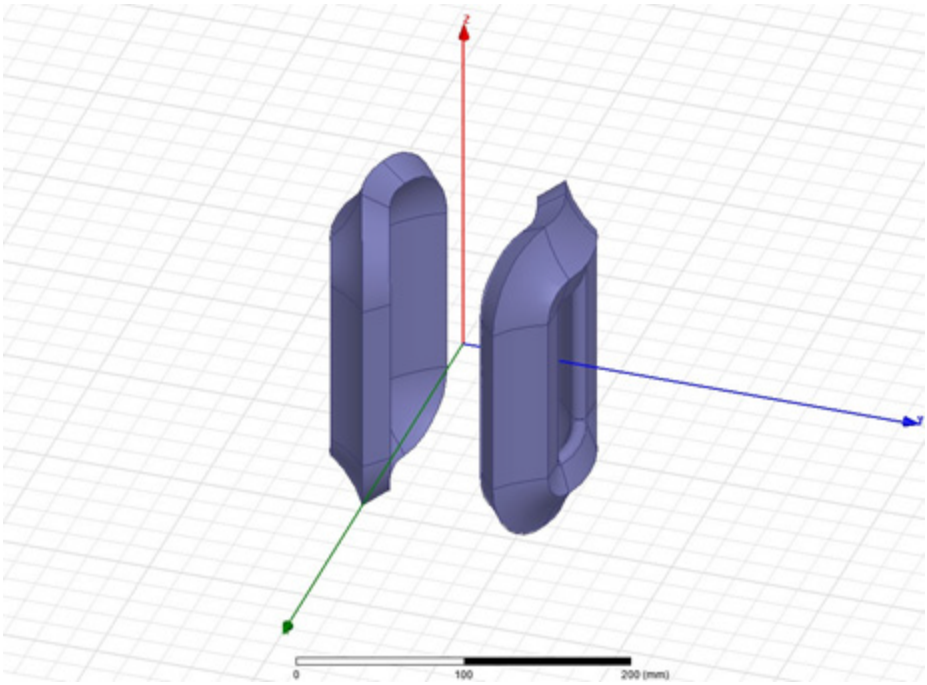


Figure 9-33 Coils of Universal-Motor Core

Creating a Coil and its Terminal (UnivMCore)

To create a coil of a universal-motor core, manually set the property of **InfoCoil** to 2, shown in " Two UnivM Cores with Different InfoCore Values" on page 9-118 as a red object.

To create its terminals, set the property of **InfoCoil** to 3 or 4 (for terminal1 and terminal2 respectively)." Terminal of a Coil for a Universal-Motor Core" on the facing page shows what a terminal looks like.

You can assign the value of this property either when creating a universal-motor core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

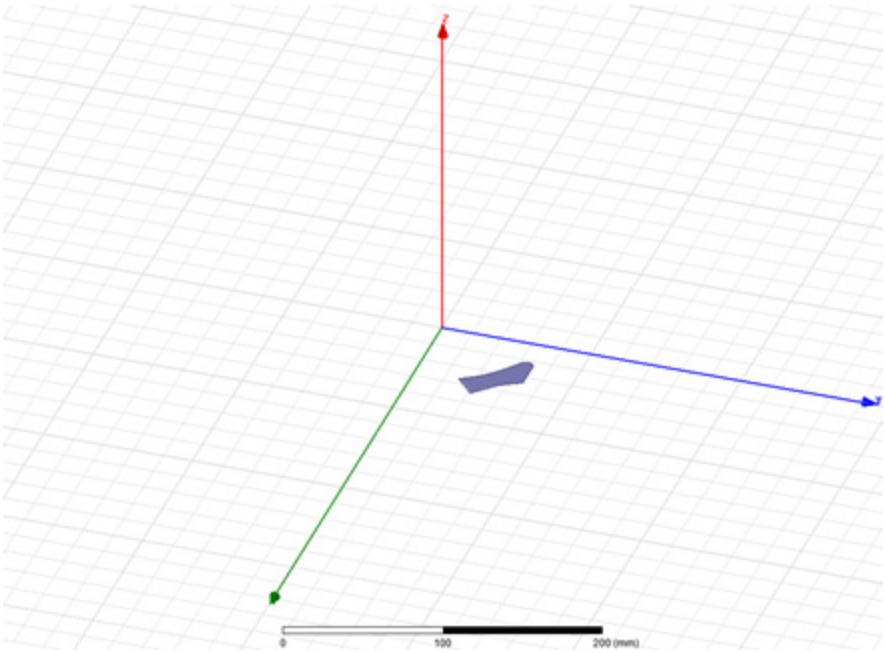


Figure 9-34 Terminal of a Coil for a Universal-Motor Core

SRMCore UDP

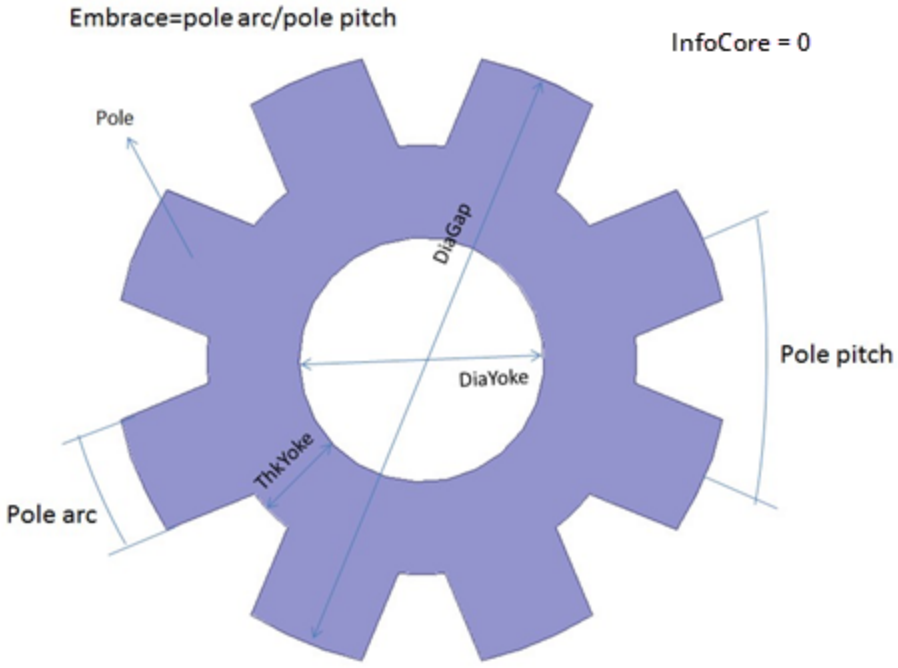
The SRMCore UDP is used to create the stator core, a stator winding, and the rotor core for switched reluctance motors. It can also create a coil terminal for current assignment.

Property	Description
DiaGap	Core diameter on gap side, DiaGap<DiaYoke for outer cores
DiaYoke	Core diameter on yoke side, DiaYoke<DiaGap for inner cores
Length	Core length
Poles	Number of poles
ThkYoke	Yoke thickness
Embrace	Pole embrace (the ratio of pole arc to pole pitch)
Delta	One side pole-edge angle, 0 for parallel pole edges
RFillet	RFillet > 0 for SRM cores with fillet corner; RFillet < 0, its absolute value defines the height of the V-shape slot bottom.
EndExt	Coil one-side end extended length
LenRegion	Region length
InfoCore	0: core; 1: core & coils; 2: coil; 3: terminal1; 4: terminal2; 100: region

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These parameters are used in the following figures.

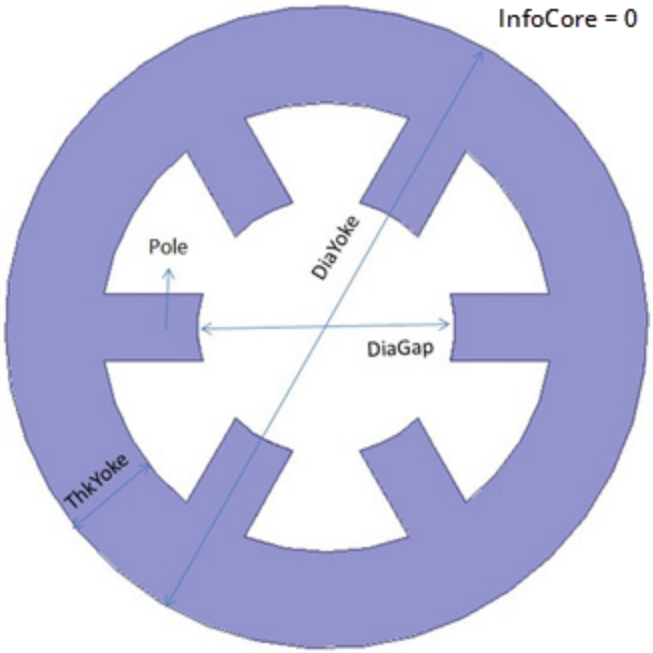
" Core and Coils of SRM Core" on the facing page shows the parameter **EndExt**, which is a parameter of a coil. The **InfoCore** value of **EndExt** is 1 (core & coils) to better describe the parameter.



Inner Core of SRM Core

Figure 9-35

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Outer Core of SRM Core

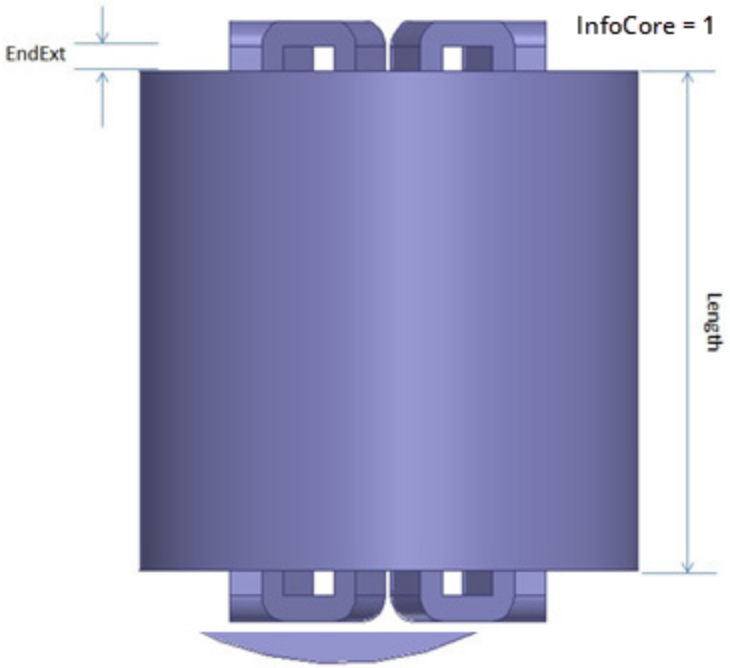


Figure 9-36

Figure 9-37 Core and Coils of SRM Core

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating an Inner or Outer Core for Switched Reluctance Machines

To create an inner core, set the value of **DiaYoke** and **DiaGap** as **DiaYoke** < **DiaGap.** To create an outer core, set the value of **DiaYoke** and **DiaGap** as **DiaGap** < **DiaYoke**. Set the **InfoCore** value to 0.

You can set the values either when creating the core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Examples of inner SRM core are shown in " Inner Core of SRM Core" on page 9-122 and " 3D Inner Core of SRM Core" below. For outer core, refer to " Outer Core of SRM Core" on the previous page and " 3D Outer Core of SRM Core" on the facing page.

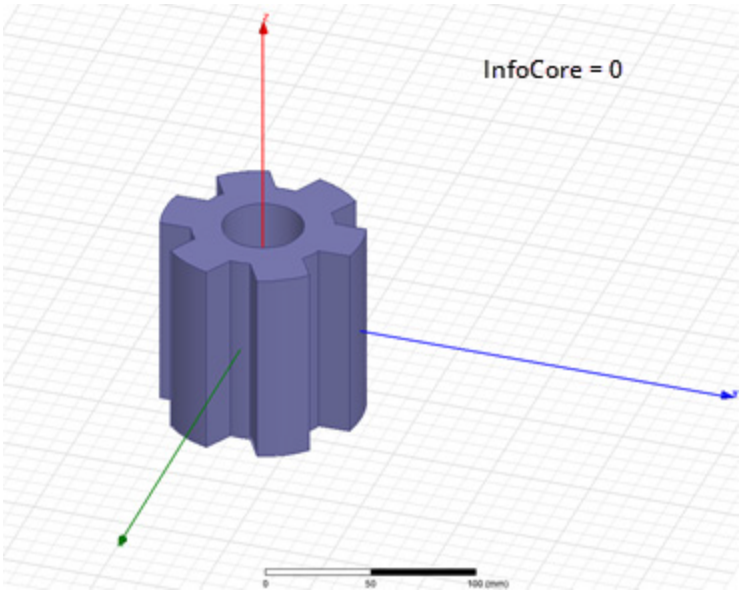


Figure 9-38 3D Inner Core of SRM Core

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

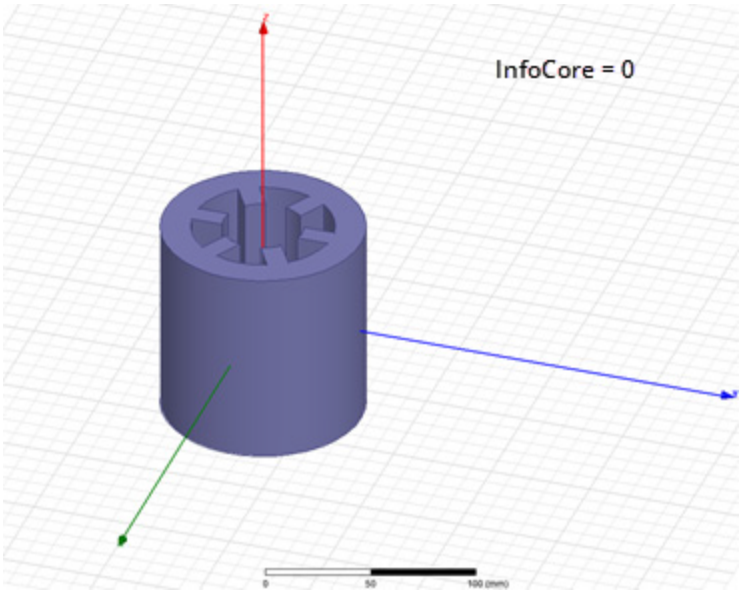


Figure 9-39 3D Outer Core of SRM Core

Creating Core and All Coils (SRMCore)

To create a core and all coils of an SRM Core, manually set the value of **InfoCoil** to 1.

You can assign the value of this property either when creating an SRM core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Core and all coils can be created only when the core is an outer core (value of **DiaYoke** > **DiaGap**).

Examples of this core are shown in " Core and Coils of SRM Core" on page 9-123 and " 3D Core and Coils of SRM Core" on the next page.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

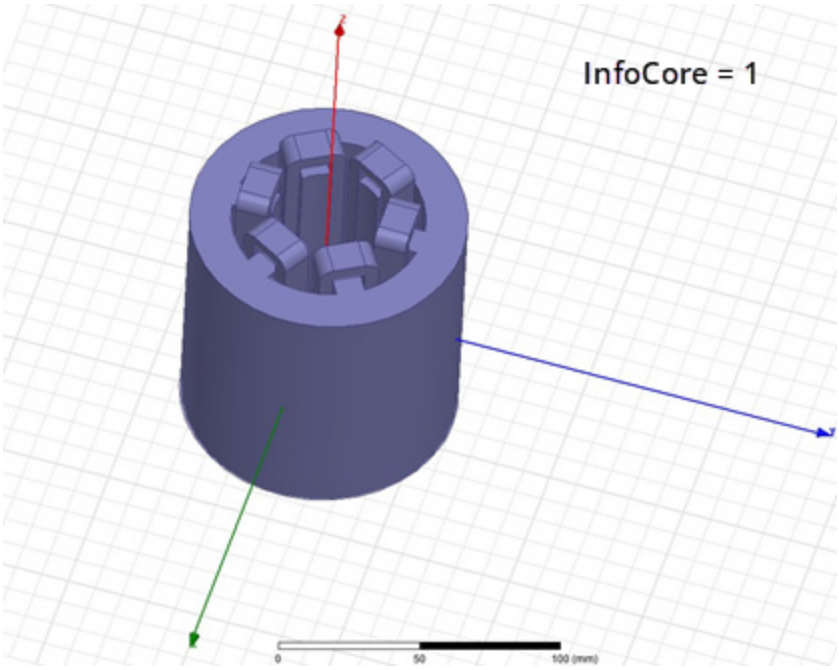


Figure 9-40 3D Core and Coils of SRM Core

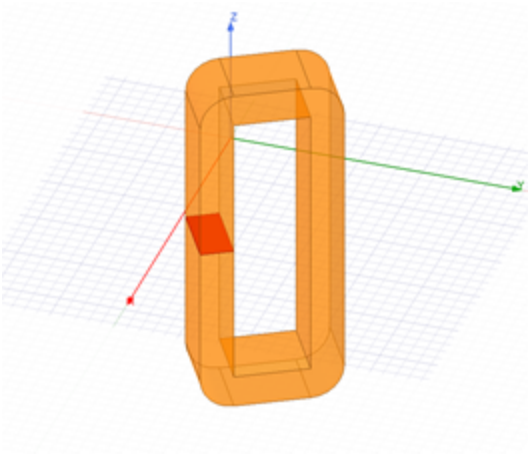
Creating a Coil and its Terminal (SRMCore)

To create a coil of an SRM core, manually set the value of **InfoCoil** to 2. An example is shown below.

To create a terminal, manually sett the property of **InfoCoil** to 3 or 4, for terminal1 or terminal2 respectively. An example is shown by the red sheet in the figure.

You can assign the value of this property either when creating an SRM core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

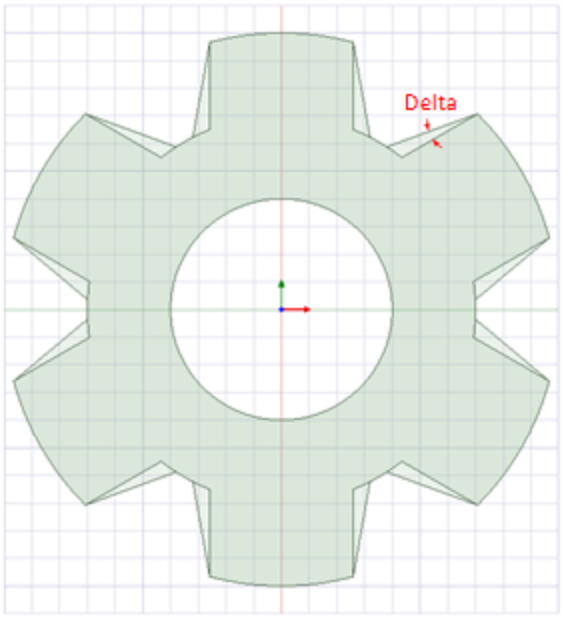
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Single Coil and its Terminal Created by SRMCore

Creating Core with Unparallel Pole Edges (SRMCore)

Parameter **Delta** is introduced to define one side pole-edge angle, as shown below. When **Delta** = 0, the two pole-edge sides are parallel.

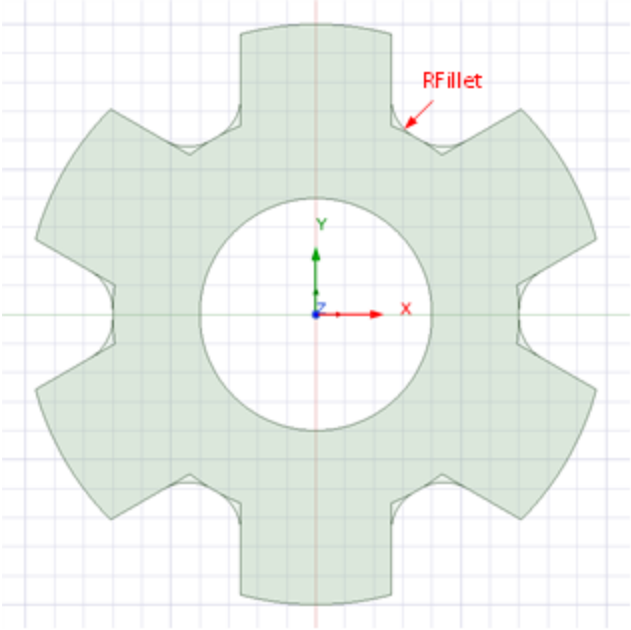


Parameter Delta for Unparallel Pole Edges

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

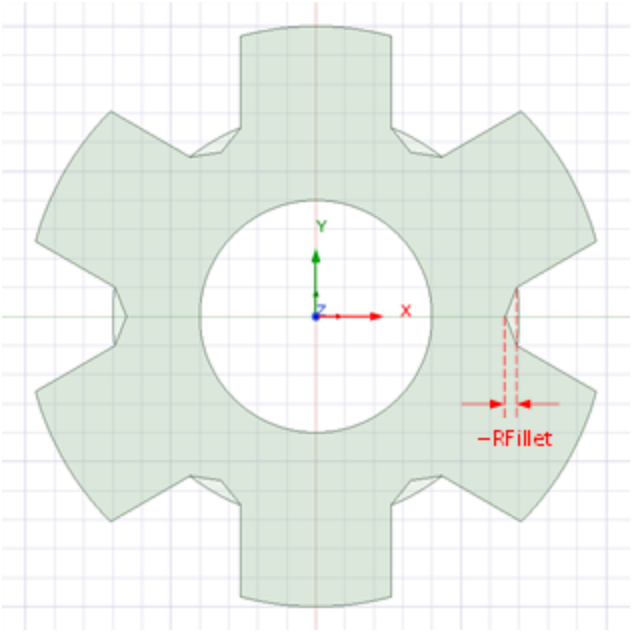
SRM Core with Slot Corner Fillet and V-Shape Bottom

Parameter **RFillet** is introduced to define corner fillet radius. **RFillet** > 0 for SRM cores with fillet corner as shown below.



When parameter **RFillet** < 0, its absolute value defines the height of the V-shape slot bottom as shown below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



SynRMCore UDP

The SynRMCore UDP is used to create the rotor core for synchronous reluctance machines.

Property	Description
DiaGap	Core diameter on gap side, or outer diameter
DiaYoke	Core diameter on yoke side, or inner diameter
Length	Core length
Poles	Number of poles
PoleType	1: ALA; 2: arc; 3: hyperbolic; 4: hyperbolic line
Barriers	Barriers per Pole, for PoleType 2 & 3 only
H	Bridge thickness, for PoleType 2 & 3 only
W	Rib width, for PoleType 2 & 3 only
R	Barrier fillet radius, for PoleType 2 & 3 only
R0	Radius of the bottom barrier arch, for PoleType 2 only
Rb	Barrier bottom minimum radius
Y0	Yoke bottom thickness
B0	Barrier bottom thickness, for PoleType of 2 & 3 only
LenRegion	Region length
InfoCore	0: core; 1: one barrier; 100: region

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The SynRM UDP supports four pole types:

- **Type 1** – axially laminated core
- **Type 2** – barriers with concentric arcs and lines
- **Type 3** – barriers with hyperbolic curves
- **Type 4** – barriers with hyperbolic polylines

Type 1

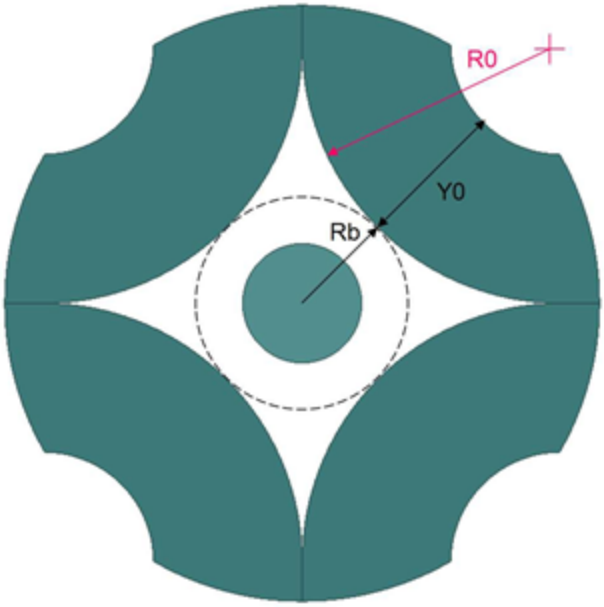
Type 2

Type 3

Type 4

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameters for SynRM core with PoleType = 1:



In the figure, *R0* is a derived parameter, as given below:

$$R0 = Rb\sin\theta/(1 - \sin\theta)$$

where $\theta = \pi/p$, and p, the number of poles, should be ≥ 4.

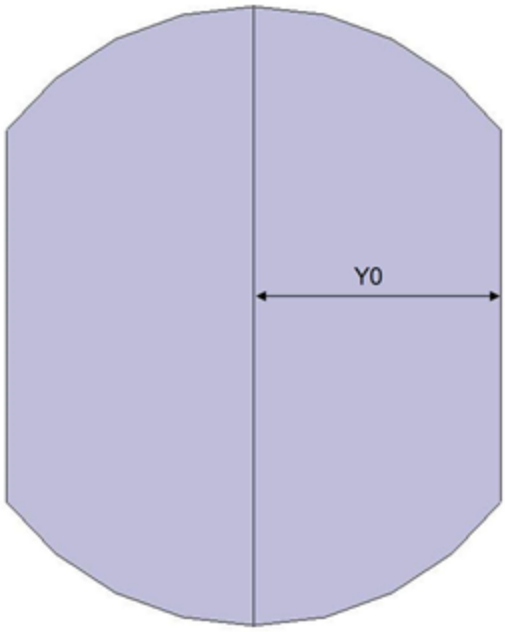
The center point (x, y) of the *R0* circle is:

$$x = (Rb + R0)\cos\theta$$

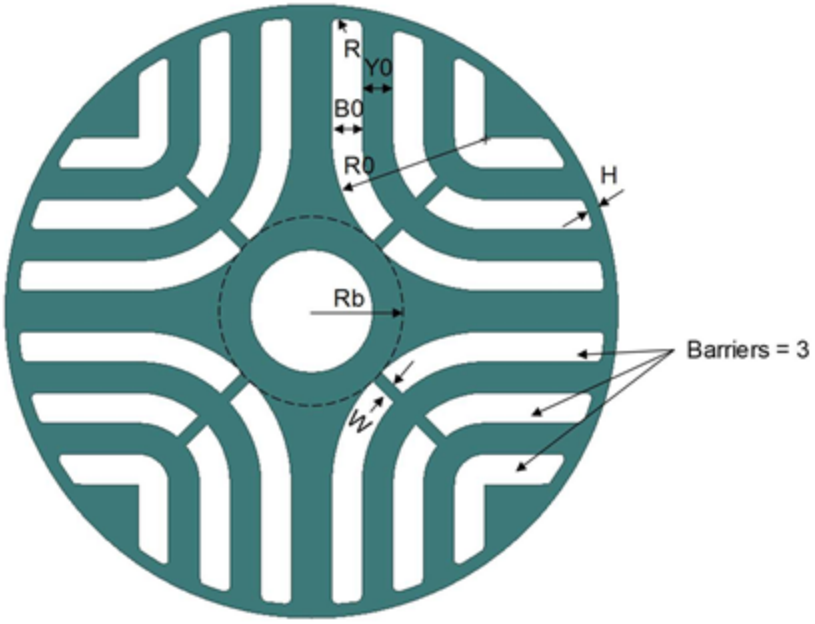
$$y = (Rb + R0)\sin\theta$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameters for a two-pole SynRM core with PoleType = 1:

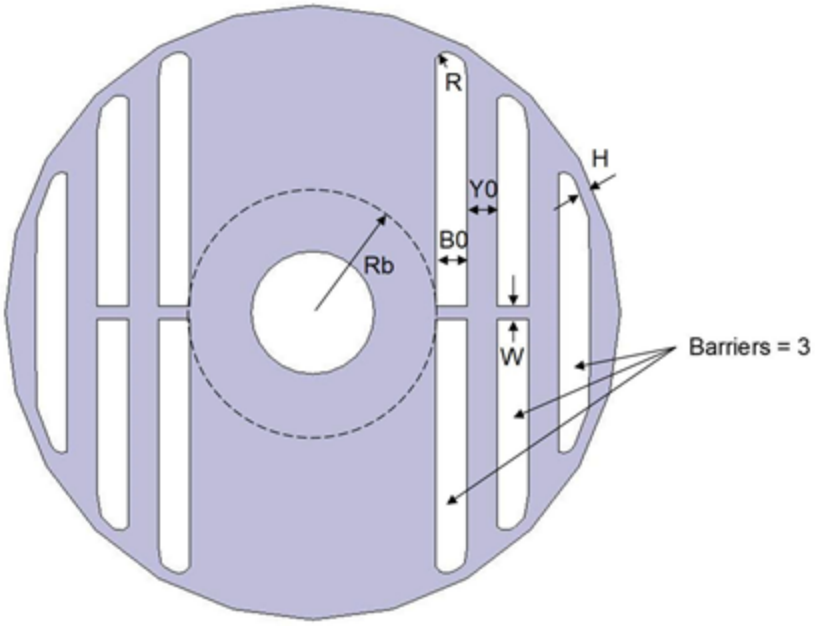


Parameters for SynRM core with PoleType = 2:

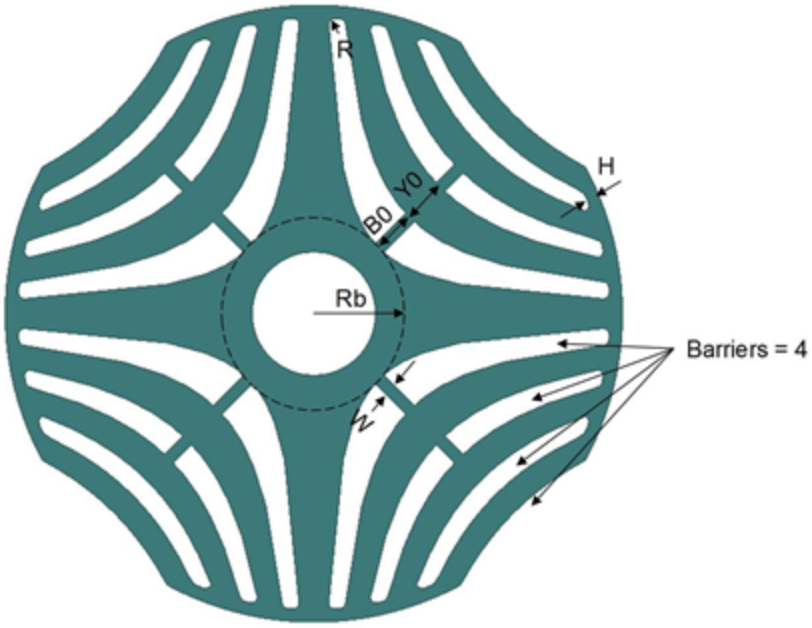


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameters for a two-pole SynRM core with PoleType = 2:



Parameters for SynRM core with PoleType = 3:



The main edges of barriers for Type 3 are hyperbolic curves defined by

$r^2 \sin(pp \bullet \theta) = R_m^2$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$x = r \bullet \cos\theta$

$y = r \bullet \sin\theta$

where *pp* is the number of pole pairs, $\pi/pp > \theta > 0$, and R_m is the minimum radius of a hyperbolic line. For different R_m values, you will get different hyperbolic curves.

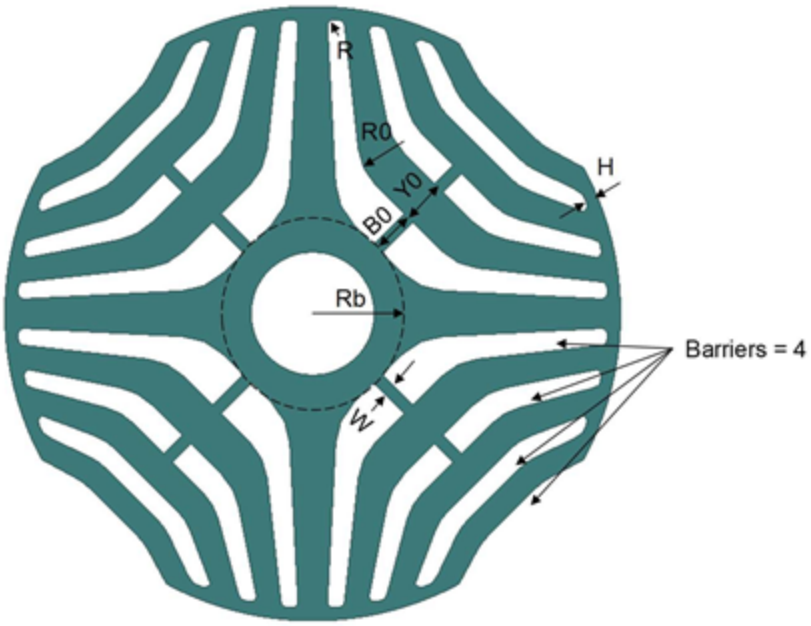
The first barrier is created based on the following two hyperbolic curves:

- for the bottom hyperbolic curve, *Rm2* = *Rb*
- for the top hyperbolic curve, *Rm1* = *Rb* +*B0*

The second and all other barriers are created based on a constant minimum barrier width and minimum tooth width at maximum limit radius which is derived from outer radius minus the value of bridge thickness parameter H.

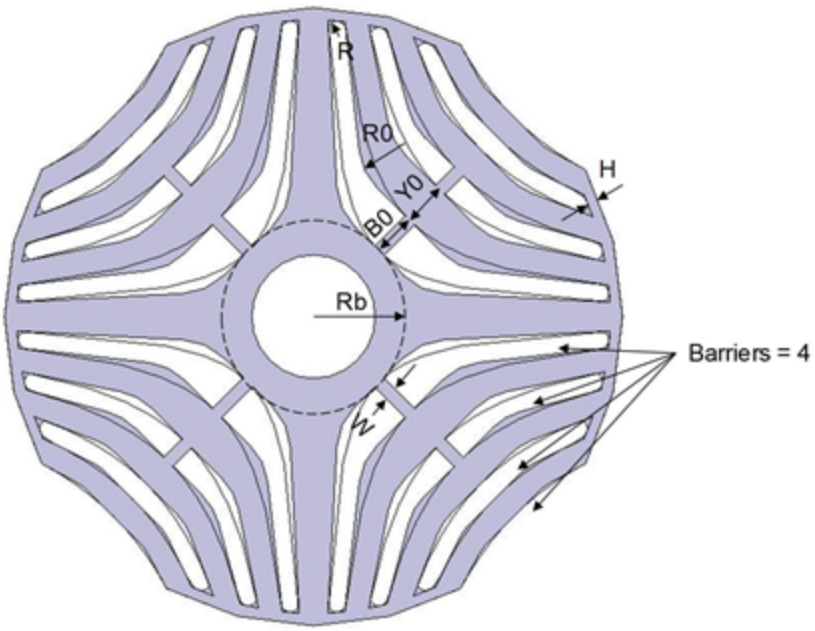
The minimum barrier width and minimum tooth width can be derived from *B0* and *Y0*, respectively. If the top and bottom hyperbolic curves are defined by *Rm1* and *Rm2*, respectively, and let *r* to be the maximum limit radius, you can obtain two points (*x1*, *y1*) and (*x2*, *y2*) at the maximum limit circle intersected with the top and bottom hyperbolic curves, respectively, based on the above hyperbolic curve equation. Note that the distance between these two intersection points is not the minimum width because the connection line is not perpendicular to the center hyperbolic curve. The angle between the connection line and normal line is the angle between the *r* line and the tangent line which can be computed from *dy*/*dx* and θ of the center hyperbolic curve at the maximum limit radius.

Parameters for SynRM core with PoleType = 4:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The main edges of barriers for Type 4 are hyperbolic polylines. A hyperbolic polyline is derived from a hyperbolic curve. It consists of one bottom line (tangent to the bottom points of the hyperbolic curve), two side lines (tangent to the point on the hyperbolic curve at the maximum limit radius), and fillet arcs defined *R0*. The value of fillet radius *R0* is the same for all hyperbolic polylines. All parameters for hyperbolic curves to derive hyperbolic polylines are defined in the same way as those for pole Type 3. The following figure shows how barriers of Type 4 are derived from those of Type 3.



Note: Two-pole SynRM cores for Types 3 and 4 will be the same as those of Type 2.

Creating an Inner Core for a Synchronous Reluctance Machine

To create an inner core, set the value of **DiaYoke** and **DiaGap** as **DiaYoke** < **DiaGap**. You can set the value either when creating the core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Set the **InfoCore** value to 0.

" 3D SynRM Core" on the next page is an example of an inner core for SynRM.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

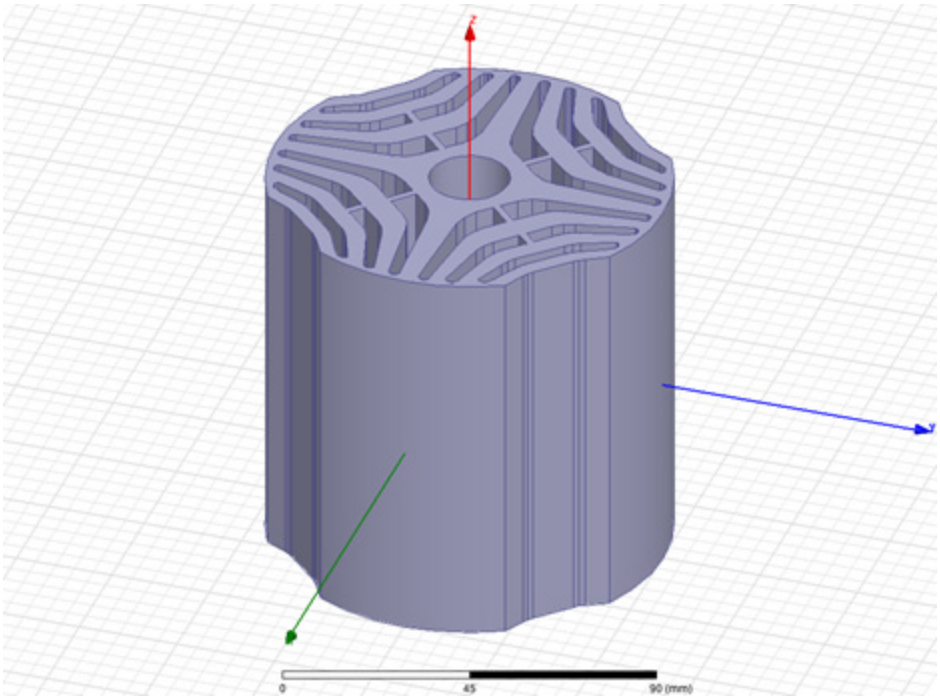


Figure 9-41 3D SynRM Core

Creating a SynRMCore with Different Barriers

You can create a SynRMCore with different barriers through several steps, as follows.

1. Create a SynRMCore as shown in Creating an Inner Core for a Synchronous Reluctance Machine.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Change the value of Barriers to 1. The result is shown in the following figure.

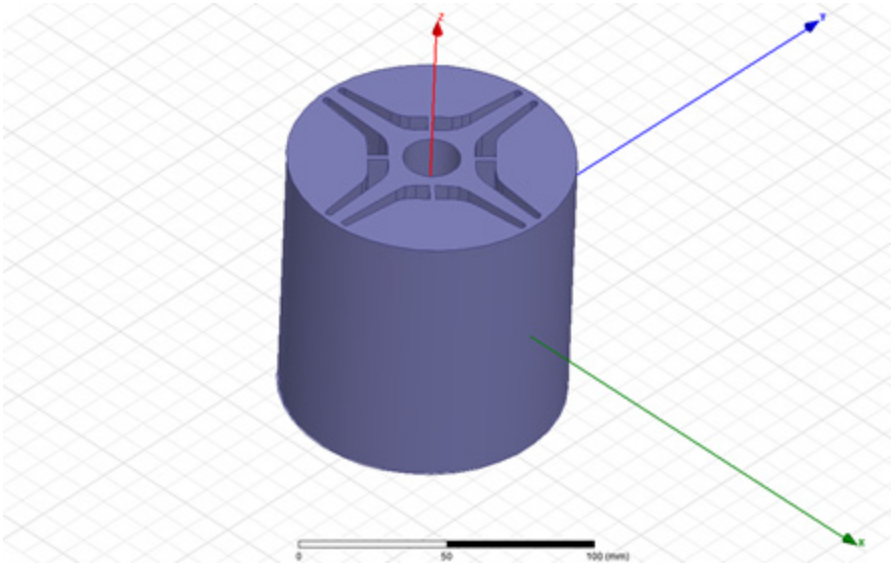
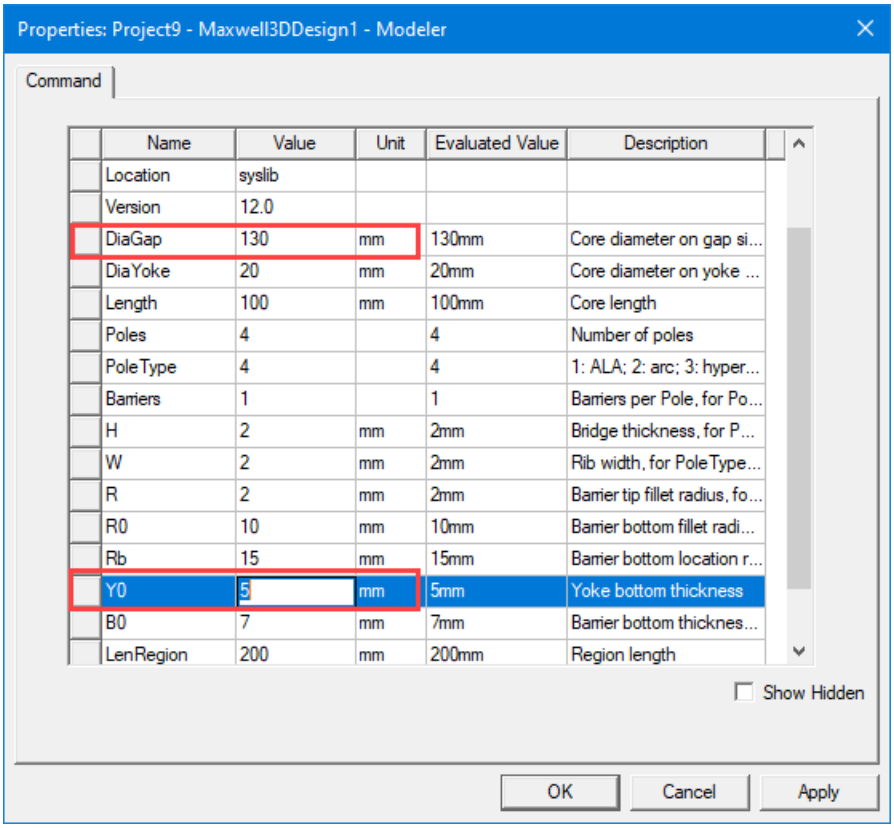


Figure 9-42 A SynRMCore with One Barrier

3. Change the value of **DiaGap** to be larger and make the value of **Y0** smaller to make enough space for other barriers. You can modify these values by clicking on **CreateUserDefinedPart** in the history tree under the SynRMCore, as shown in the following figure.

   As an example, the values of **DiaGap** and **Y0** are changed to 130 and 5, respectively.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

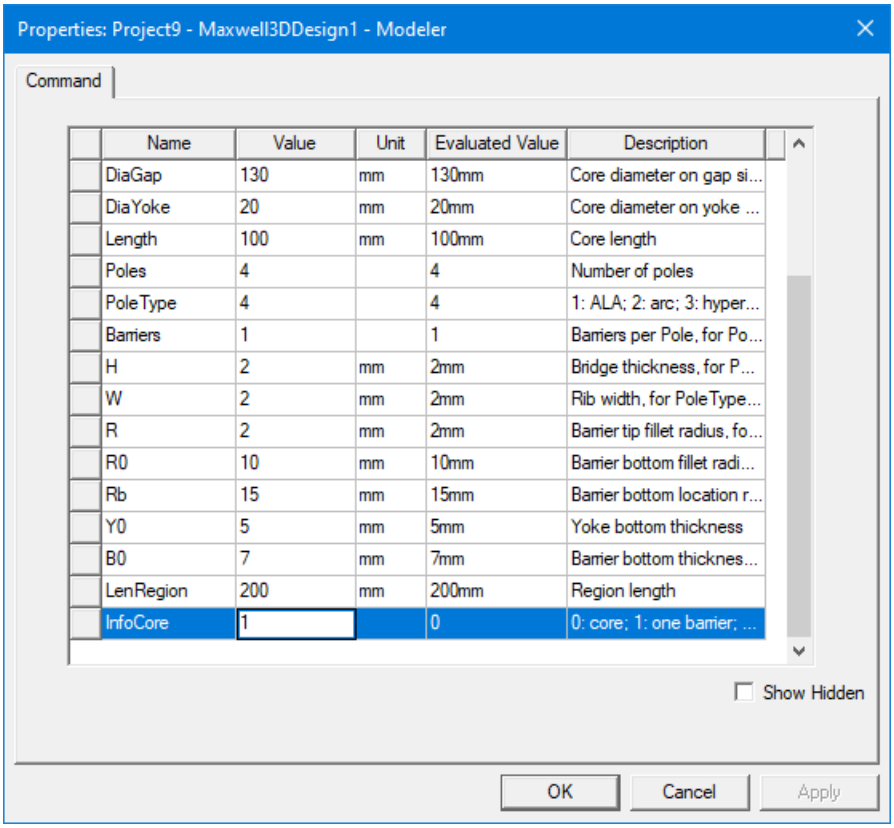


4. Make a copy (or multiple copies if more than 1 barrier is needed) of the current SynRMCore object.

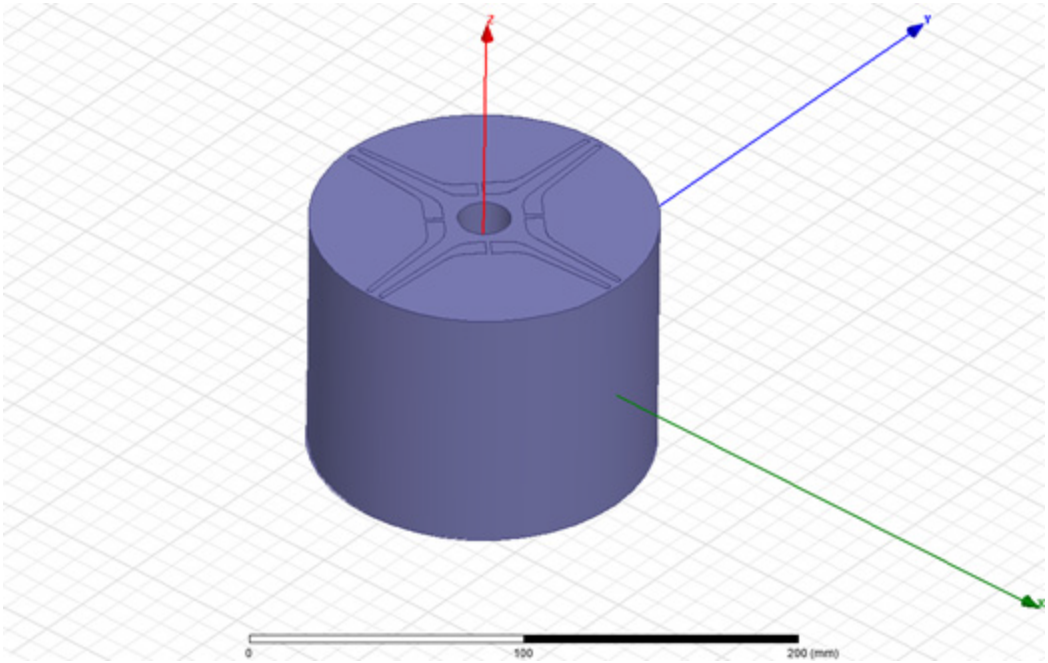
   In the history tree, click **SynRMCore1**, press **Ctrl**+**C**, and then press **Ctrl**+**V** to duplicate the SynRMCore.

5. Change the value of **InfoCore** of SynRMCore2 to 1 to create a barrier.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The result of this change is shown in the following figure.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 9-43 Result of InfoCore of SynRMCore2 Being Changed

6. Change the value of **Rb** of SynRMCore2 (barrier) to make it larger, and thus make the barrier away from the center of the core.

   As an example, the value of **Rb** of SynRMCore2 is changed from 15 to 25 by clicking on **CreateUserDefinedPart** in the history tree under the SynRMCore2, as shown in the following figure.

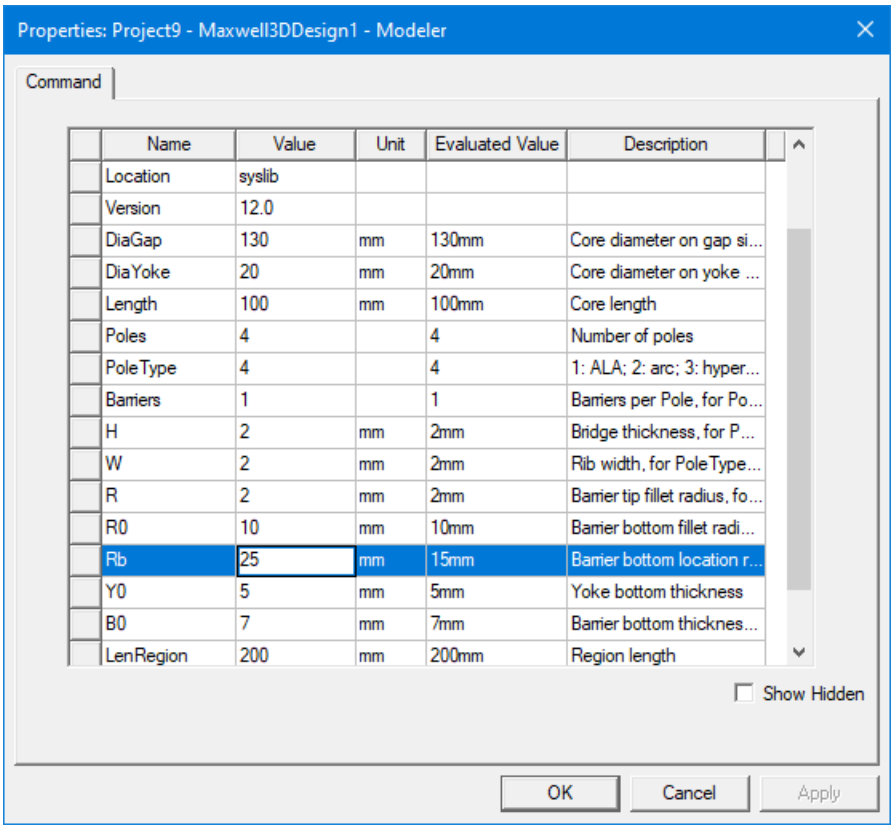


Figure 9-44 Changing the Value of Rb of SynRMCore2

   Another option is to change other values of SynRMCore2 (barrier) according to different needs, such as set the value of **W** to 0.

7. Then rotate the coordinate system. Objects in it should look like those in the following figure:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

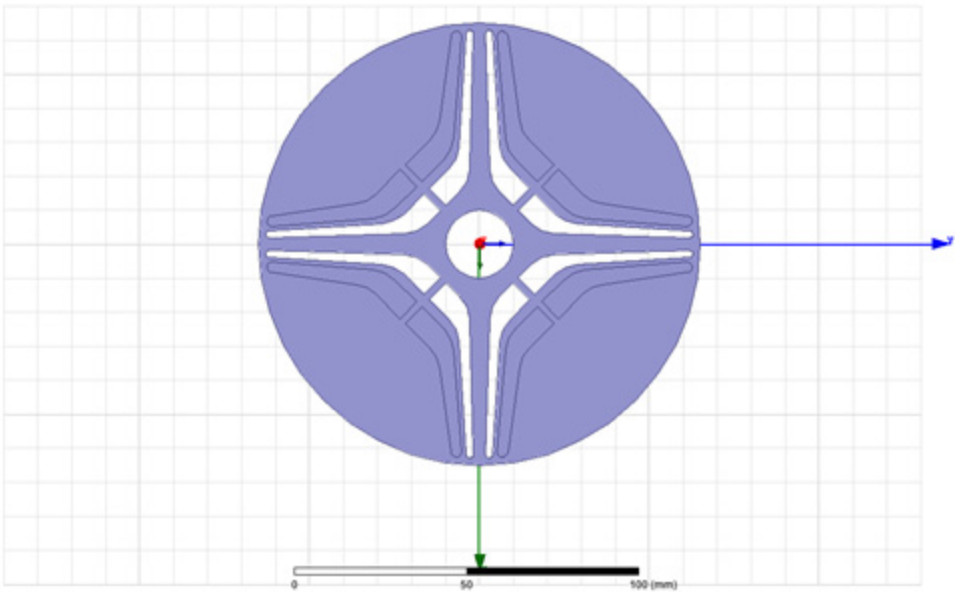


Figure 9-45 Before Subtraction of Two Objects

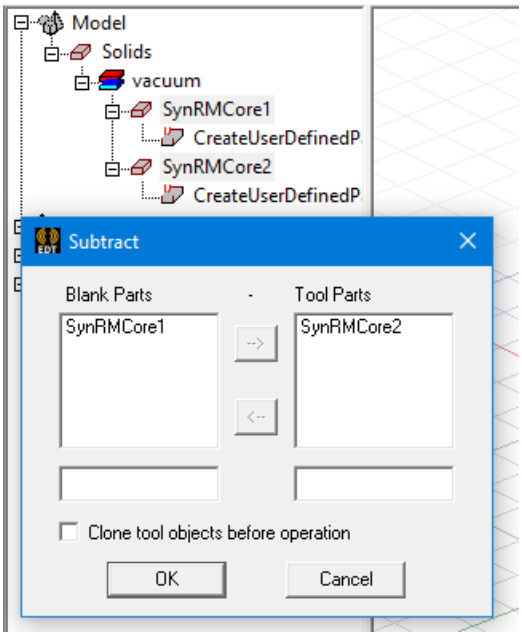
8. Perform a subtraction on two objects:

   Choose both objects simultaneously by pressing **Ctrl** and clicking on two objects in the history tree on the screen.

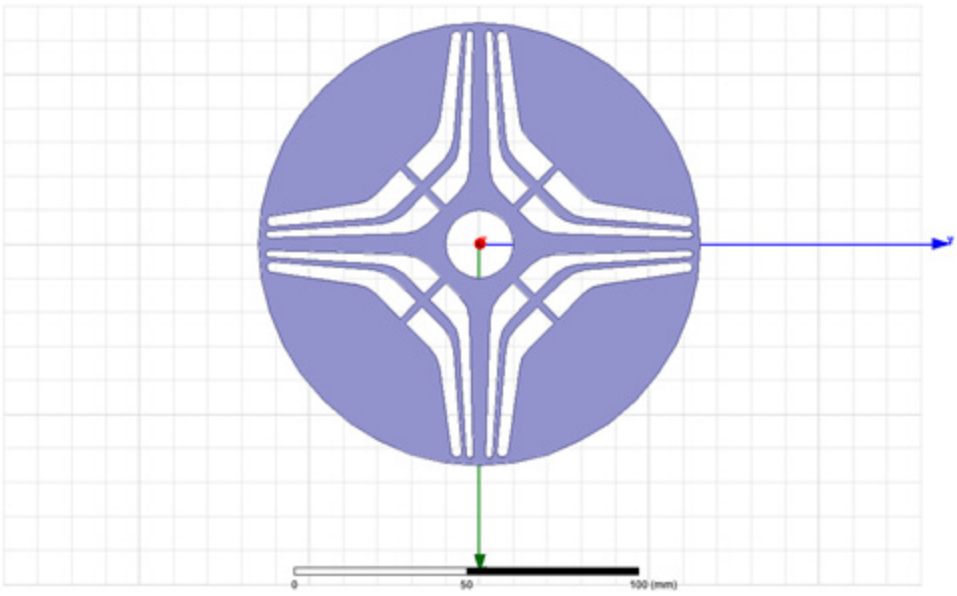
   Then click **Modeler** > **Boolean** > **Subtract** to do the subtraction.

   The SynRMCore1 should be in **Blank Parts** whereas SynRMCore2 is in **Tool Parts**, as shown in the following figure:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



9. Click **OK** to finish subtraction. A SynRMCore with different barriers is shown in the following figure:



To make the barriers look different, you can change the parameters of barriers like **W** and **B0**. Also, repeating the processes can make multiple barriers.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ClawPoleCore UDP

The ClawPoleCore UDP is used to create the claw-pole rotor core, a field winding and/or a PM for claw-pole alternators. It can also create a coil terminal for current assignment.

Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaShaft for outer cores
DiaShaft	Core diameter on shaft side, DiaShaft < DiaGap for inner cores
Length	Core length
Poles	Number of poles
EmbraceTip	Embrace of pole tip
EmbraceRoot	Embrace of pole root
ThickTip	Thickness of pole tip
ThickRoot	Thickness of pole root
ThickShoe	Thickness of side shoes
DepthSlot	Depth of slot between two poles
ThickYoke	Thickness of yoke
LengthPole	Length of pole from tip to tip
LengthMag	Length of magnet or the second air-gap
SegAngle	Deviation angle for skewed pole sides (1~10, <1 for true surface)
LenRegion	Region length
InfoCore	0: core; 1: core & coil; 2: coil; 3: magnet; 4: terminal; 100: region

These parameters are used in the following figures. " Transparent Claw Pole Core (InfoCore set to 0)" on the next page is in transparent to show parameters **EmbraceTip** and **EmbraceRoot** which otherwise are hard to show. For **LengthMag** parameter, refer to Creating a Magnet (Claw Pole) .

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

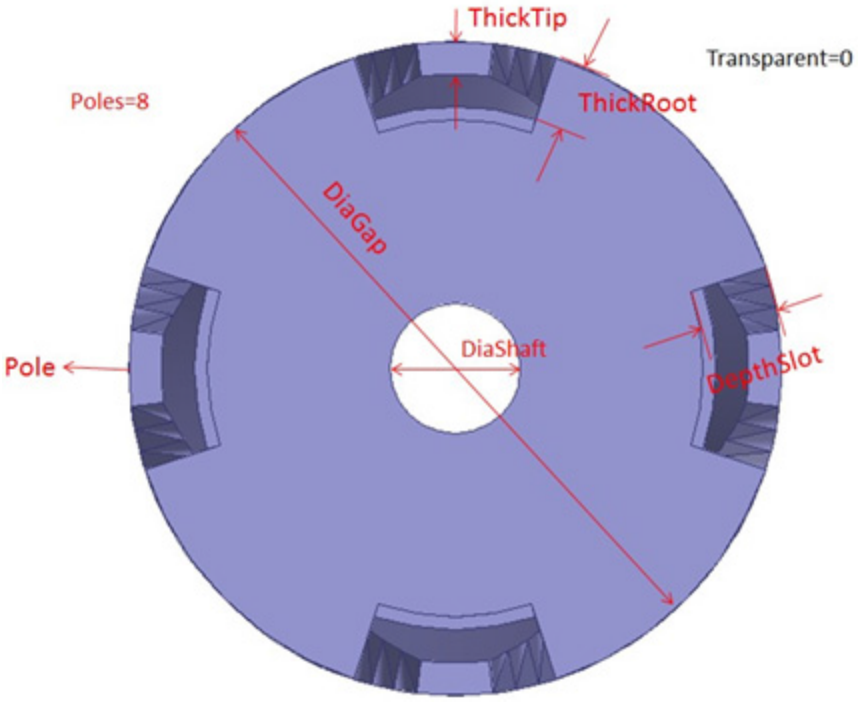


Figure 9-46 Claw Pole Core (InfoCore set to 0)

Figure 9-47

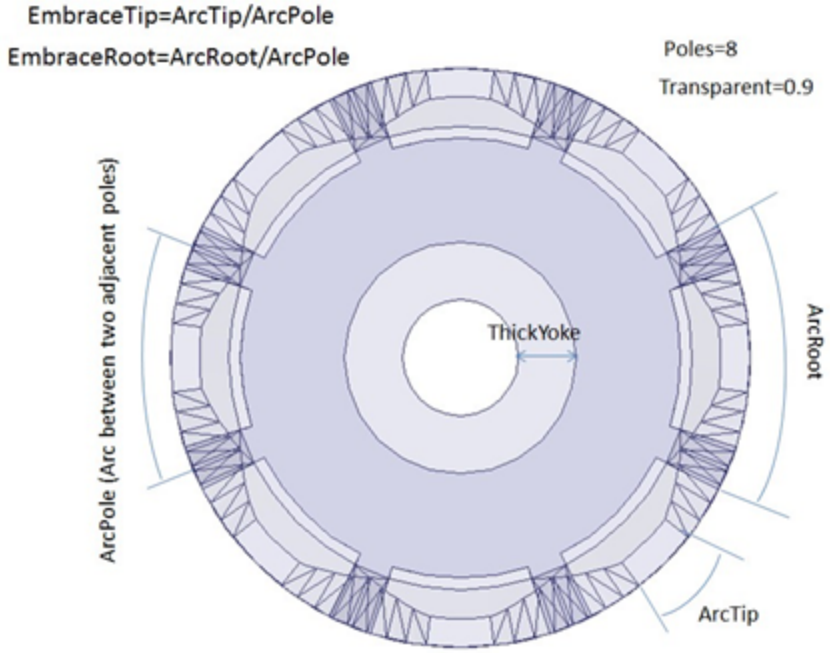


Figure 9-48 Transparent Claw Pole Core (InfoCore set to 0)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 9-49

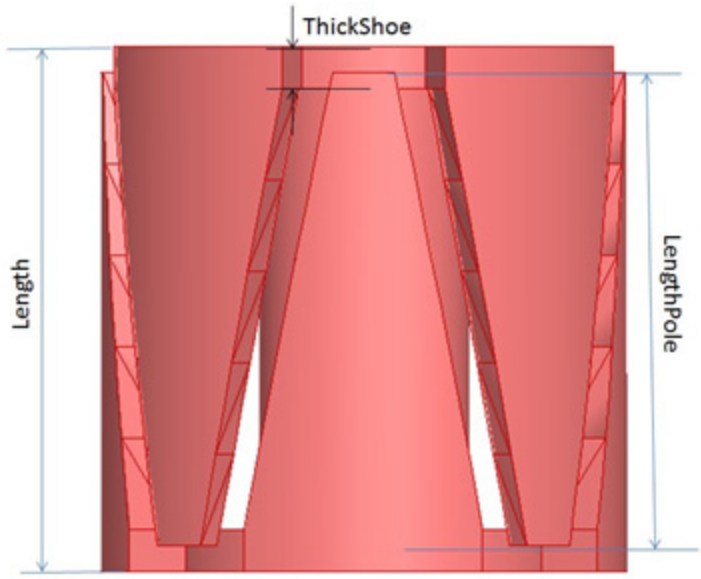


Figure 9-50 3D Claw Pole Core (InfoCore set to 0)

Creating an Inner Claw Pole Core

Set the value of **DiaShaft** and **DiaGap** as **DiaShaft** < **DiaGap** to create an inner claw pole core. You can set the value either when creating the core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Set the **InfoCore** value 0.

Examples of inner claw-pole core are shown in Figure 1-1, Figure 1-2, and Figure 1-3.

Creating a Coil and its Terminal (Claw Pole)

You can create a coil of a claw pole core manually by setting the value of **InfoCoil** to 2. An example is shown in" Coil of a Claw-Pole Core" on the next page.

You can create a terminal of a claw pole core manually by setting the value of **InfoCoil** to 4. " Terminal of Coil in this Claw-Pole Core" on the next page gives an example of this terminal.

You can assign the value of this property either when creating a claw-pole core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

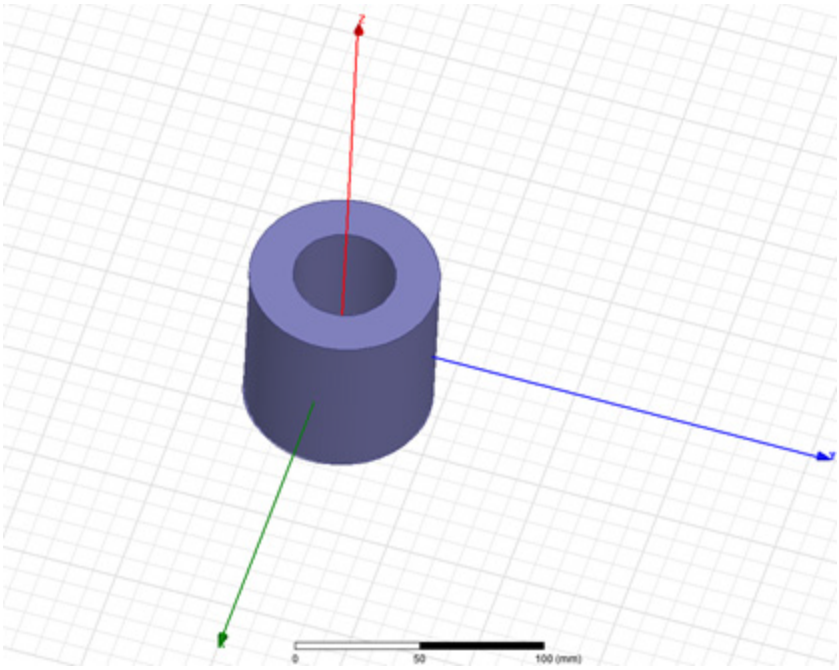


Figure 9-51 Coil of a Claw-Pole Core

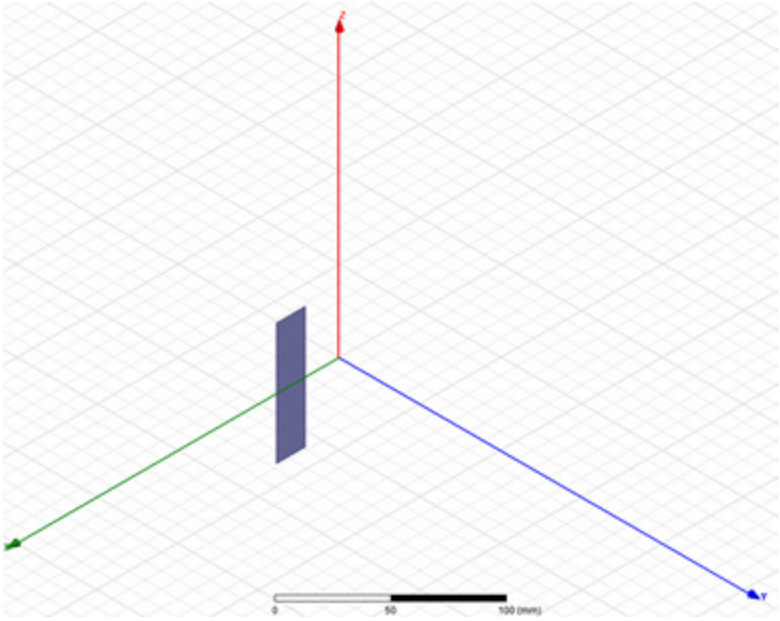
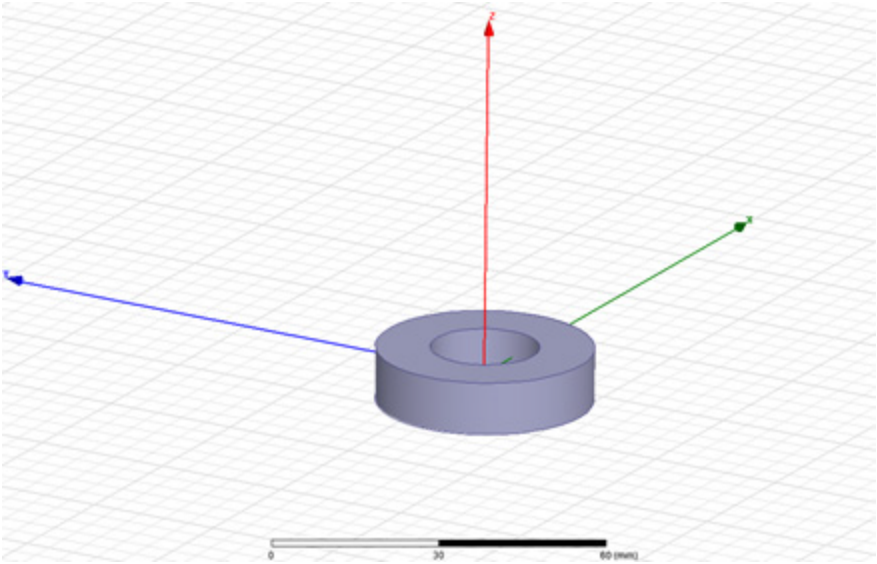


Figure 9-52 Terminal of Coil in this Claw-Pole Core

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating a Magnet (Claw Pole)

To create a magnet, you need to set the value of **LengthMag** larger than 0 (by default, it is 0) and value of **InfoCore** to be 3. An example is shown in the following figure.



Magnet of ClawPoleCore

The following figure visualizes the **LengthMag** parameter.

Figure 9-53 LengthMag of a Magnet of ClawPoleCore

You can assign the value of these properties either when creating a claw-pole core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

PMCore UDP

The PMCore UDP is used to create a PM stator/rotor core and a PM for PM commutating DC machines, PM adjustable-speed synchronous machines, or PM brushless DC motors.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaYoke for outer cores
DiaYoke	Core diameter on yoke side, DiaYoke < DiaGap for inner cores
Length	Core length
Poles	Number of poles
PoleType	Pole type: 1 to 5
Embrace	Pole embrace (not for type 4)
ThickMag	Max thickness of magnets
WidthMag	Magnet width (for types 4 & 5)
Offset	Pole arc offset (for types 1, 2 & 3)
Bridge	Bridge thickness (for type 5 only)
Rib	Rib width (for type 5 only), Rib=0 for rectangle ducts
LenRegion	Region length.
InfoCore	0: core; 1: magnets; 2: magnet; 100: region

These parameters are used in the figures that follow.

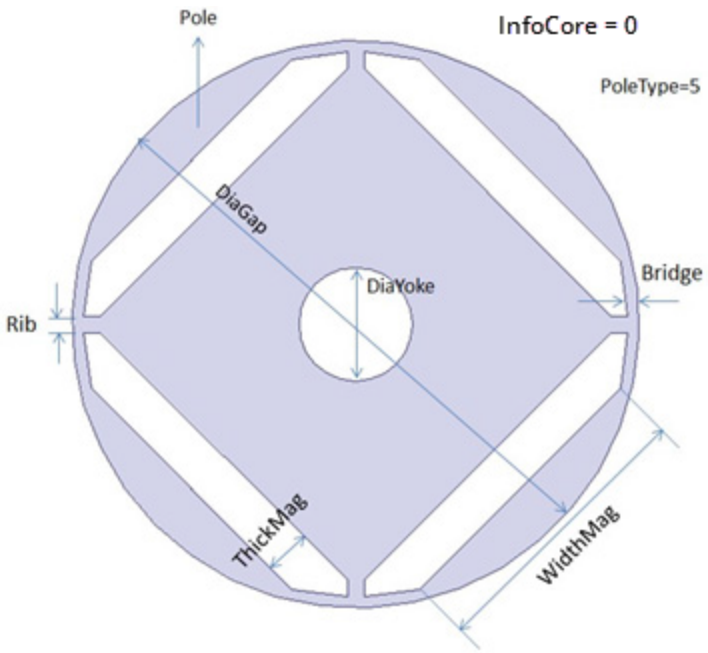


Figure 9-54 PM Core with PoleType Set to 5

Figure 9-55

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

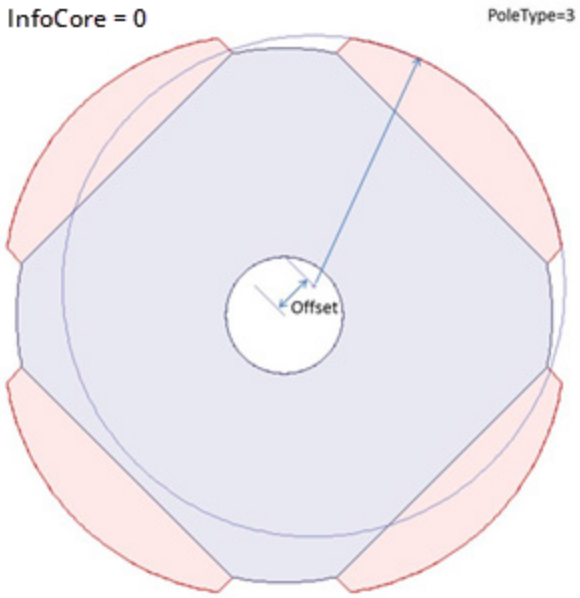


Figure 9-56 PM Core with PoleType Set to 3

Creating an Inner or Outer PM Core

Set the value of **DiaYoke** and **DiaGap** as **DiaYoke** < **DiaGap** to create inner cores or **DiaGap** < **DiaYoke** for outer cores. In addition, r to create an outer core, the **Pole Type** must be 1 or 2. You can set the values either when creating the core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Set the **InfoCore** value to 0.

An inner PM core and an outer PM core are shown in Figure 1-1 and Figure 1-2 , respectively.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

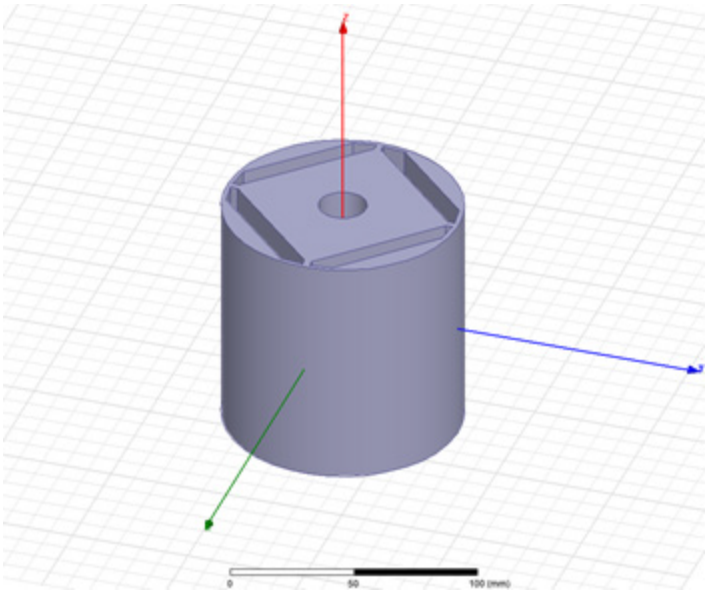


Figure 9-57 Inner PM Core with PoleType Set to 5

Figure 9-58

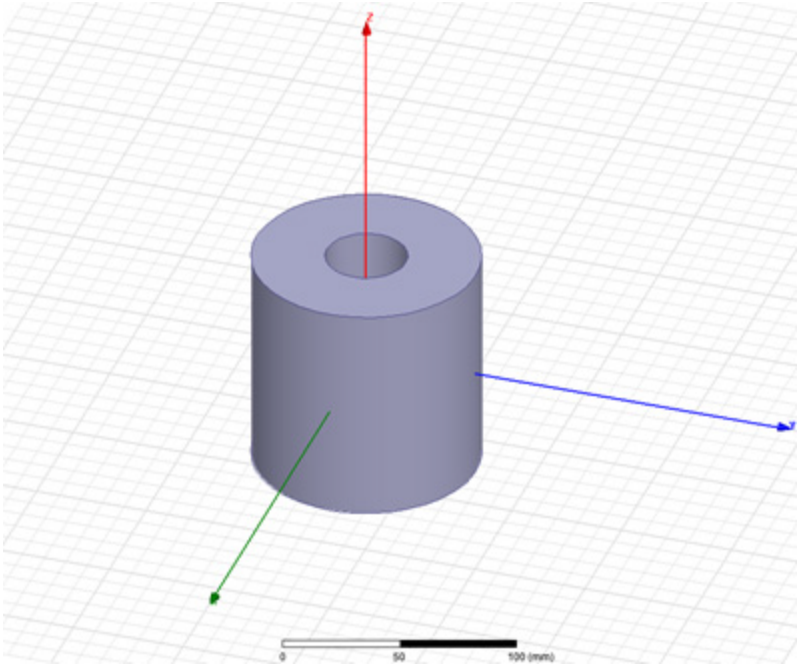


Figure 9-59 Outer PM Core with PoleType Set to 2

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating All Magnets (PMCore)

You can create all magnets of a PM core manually by setting the value of **InfoCore** to 1. An example is shown by the four inner blue objects in the following figure, for which InfoCore has been set to 0 allowing the OuterPM core (gray object) and all magnets for PoleType=2 to be shown.

You can assign the value of this property either when creating a PM Core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

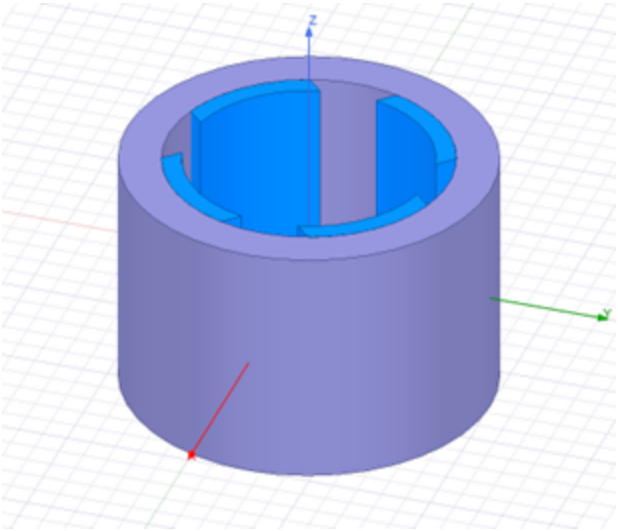


Figure 9-60 All Magnets of a PM Core

Creating a Magnet (PM Core)

You can create a magnet of a PM core manually by setting the value of **InfoCore** to 2. An example is shown in the following figure.

You can assign the value of this property either when creating a PM core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

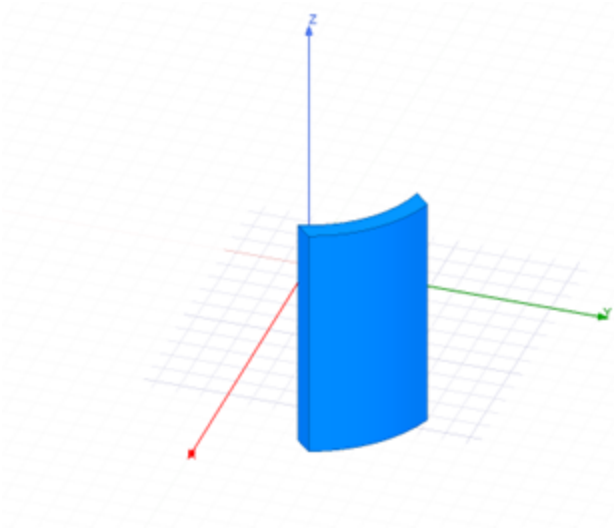
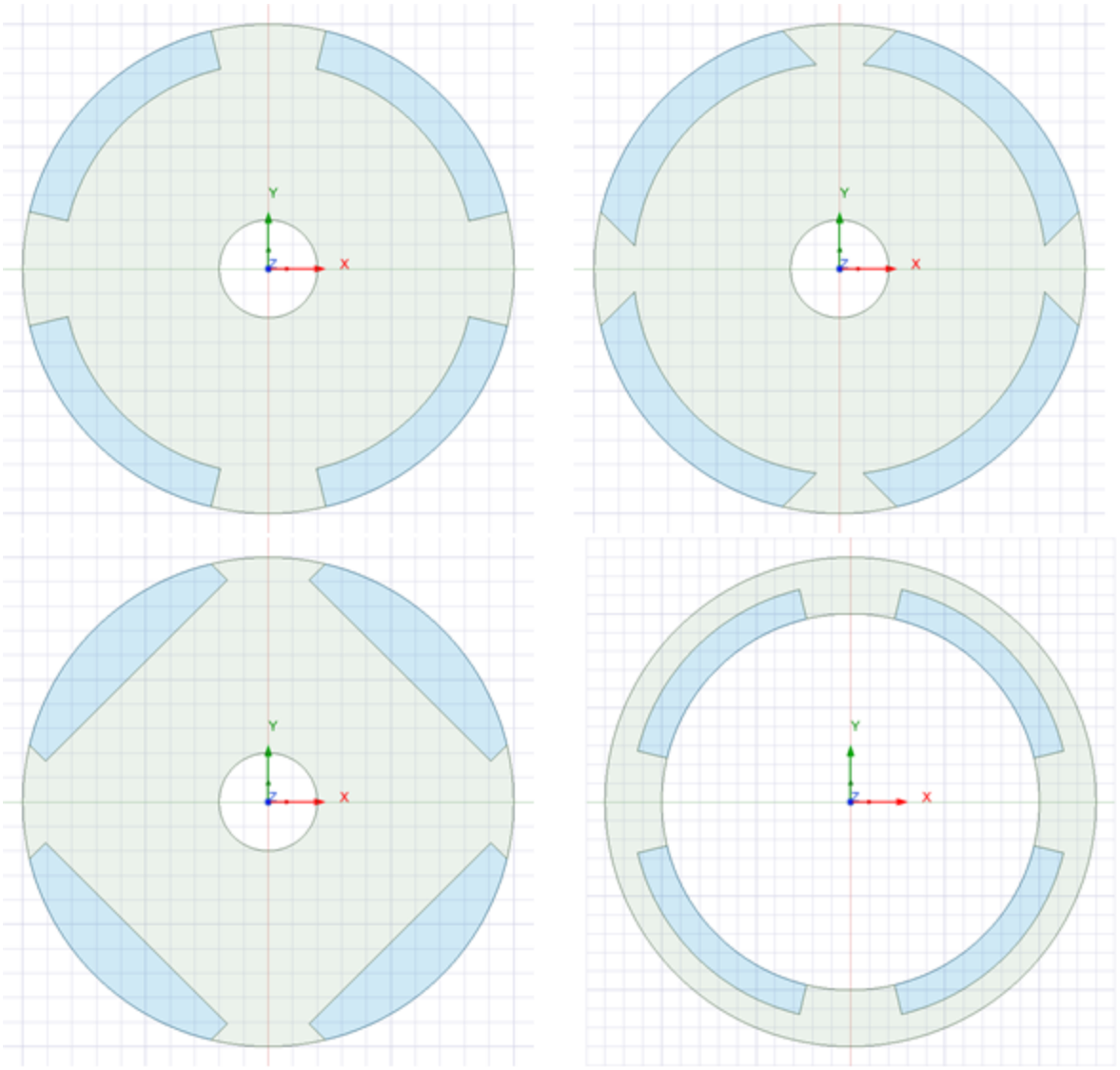


Figure 9-61 A Magnet of PM Core

Creating an Insert Surface PM Core

When the value of **PoleType** is 1 to 3, you can create an insert surface PM core by setting **Embrace** to a negative value as shown in the examples below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



IPMCore UDP

The IPMCore UDP is used to create an IPM core and a PM for IPM synchronous machines or IPM brushless DC motors.

Property	Description
DiaGap	Core diameter on gap side, or outer diameter
DiaYoke	Core diameter on yoke side, or inner diameter
Length	Core length
Poles	Number of poles

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Property	Description
PoleType	Pole type: 1 to 6
D1	Limited diameter of PM ducts
O1	Bottom width for separate or flat-bottom duct (for types 3~6)
O2	Distance from duct bottom to shaft surface (for types 2~6)
B1	Duct thickness
Rib	Rib width
HRib	Rib height (for types 3~5)
DminMag	Minimum distance between side magnets (for types 3~5)
ThickMag	Magnet thickness
WidthMag	Total width of all magnet per pole
LenRegion	Region length
InfoCore	0: core; 1: magnets; 2: ducts; 3: one-pole magnet; 100: region

Most parameters of IPMCore UDP are shown in the following figure except those common parameters like **DiaGap**, **DiaYoke** and **Length**.

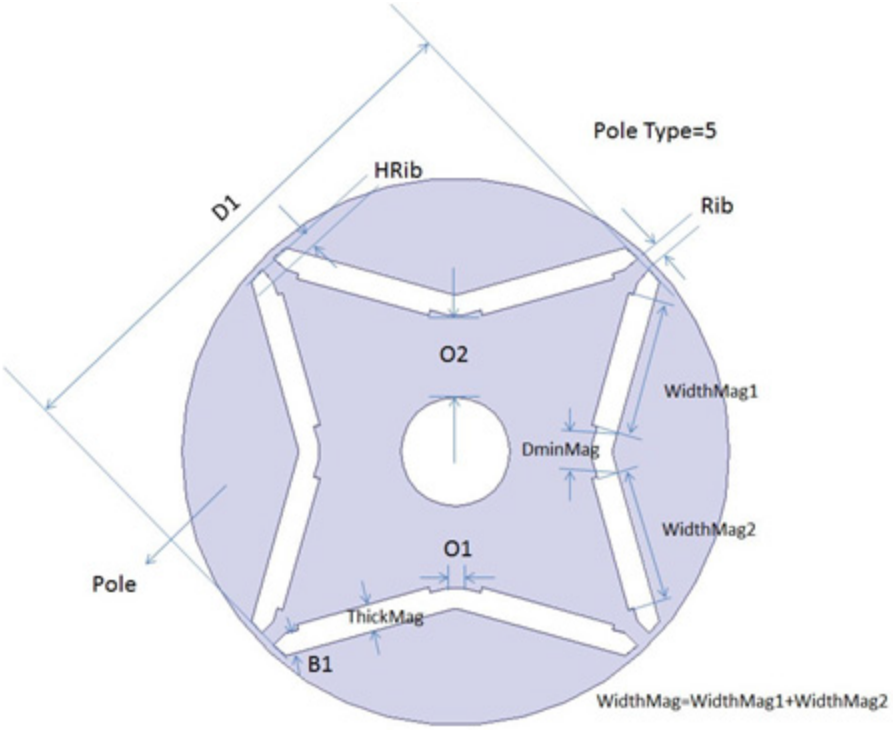


Figure 9-62 Parameters of IPM Core

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating an Inner IPM Core

Set the value of **DiaYoke** and **DiaGap** as **DiaYoke** < **DiaGap** to create an inner IPM core. You can set the value either when creating the core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Set the **InfoCore** value to 0.

An example is shown in the following figure.

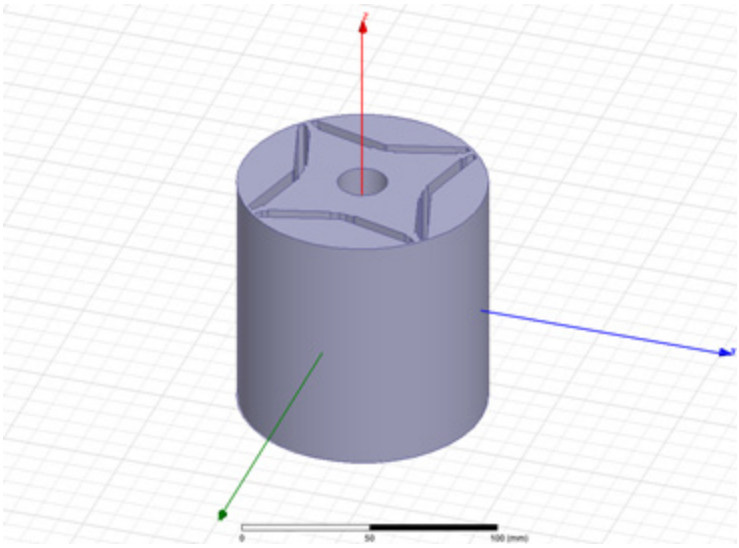


Figure 9-63 Inner IPM Core

Creating All Magnets (IPM Core)

You can create all magnets of an IPM core manually by setting the value of **InfoCoil** to 1.

You can assign the value of this property either when creating an IPM Core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

An example is shown in the following figure.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

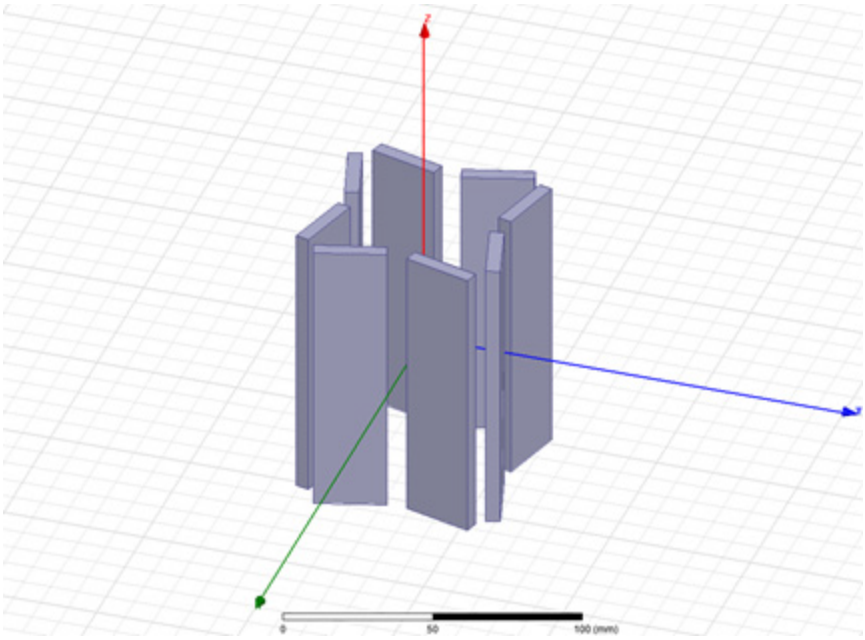


Figure 9-64 All Magnets of IPM Core

Creating One-Pole Magnets

You can create one-pole magnets of an IPM core manually by setting the property of **InfoCoil** to 3.

You can assign the value of this property either when creating an IPM core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

The following figure shows an example of one-pole magnets.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

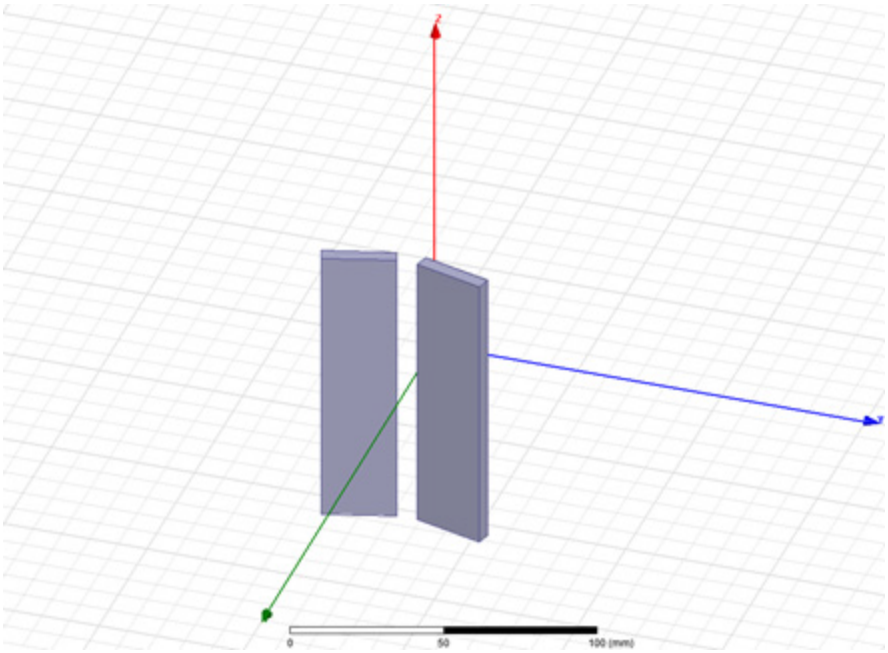


Figure 9-65 One-Pole Magnets of an IPM Core

PMDamperCore UDP

The PMDamperCore UDP is used to create the PM rotor core with damper for line-start PM synchronous machines.

Property	Description
DiaGap	Core diameter on gap side, or outer diameter
DiaYoke	Core diameter on yoke side, or inner diameter
Length	Core length
Skew	Skew angle in core length range
Slots	Number of damper slots per pole
SlotType	Slot type: 1 to 4
Hs0	Slot opening height
Hs01	Slot closed bridge height
Hs1	Slot wedge height
Hs2	Slot body height
Bs0	Slot opening width
Bs1	Slot wedge maximum width
Bs2	Slot body bottom width, 0 for parallel teeth.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Property	Description
Rs	Slot body bottom fillet
FilletType	0: a quarter circle; 1: tangent connection; 2&3: arc bottom.
Poles	Number of poles
PoleType	Pole type: 1 to 8
D1	Limited diameter of PM ducts
O1	A locating dimension of PM ducts, for pole types 3, 4, 7 & 8.
O2	A locating dimension of PM ducts, for pole types 1, 3, 5, 6, 7 & 8
B1	Barrier width; for pole types 1, 2 & 3
Rib	Rib to hold PM ducts, for pole types 1 to 7.
ThickMag	Magnet thickness
WidthMag	Total width of all magnet per pole
BarEndExt	One-side damper bar end extended length
RingLength	One-side axial ring length
RingHeight	Radial ring height
SegAngle	Deviation angle for slot arches (10~30, <10 for true surface)
LenRegion	Region length
InfoCore	0: core; 1: magnets; 2: damper; 100: region.

These parameters are used in the figures that follow.

Note:

- Parameters O1 and O2 are locating dimensions of PM ducts. They may have different meanings in different pole types. For example, " PMDamperCore with Pole Type Set to 3 " on the facing page shows what O1 and O2 refer to in PMDamper Core with Pole Type 3.
- In " Damper of PMDamperCore (red) " on the facing page and " 3D Damper of PMDamperCore (red)" on page 9-160 , the gray object (a PMDamperCore with InfoCore set to 0) is used as reference to show the red object (a PMDamperCore with InfoCore set to 2)
- For parameters such as **DiaGap, DiaYoke, Length,Skew**, **FilletType**, **SegAngle** and parameters related to slots, refer to SlotCore UDP parameters

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

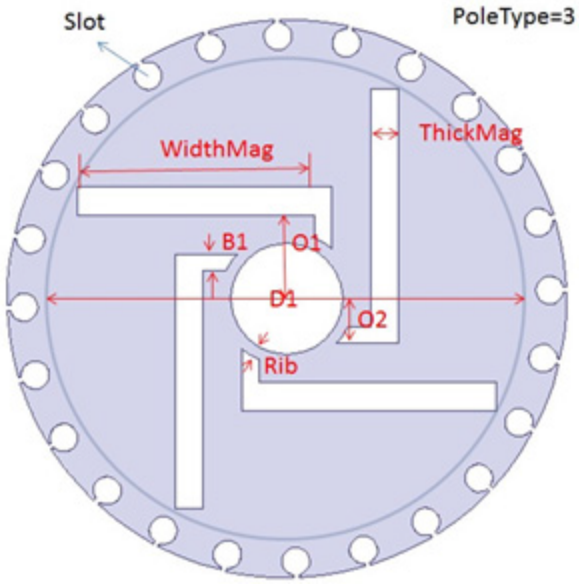


Figure 9-66 PMDamperCore with Pole Type Set to 3

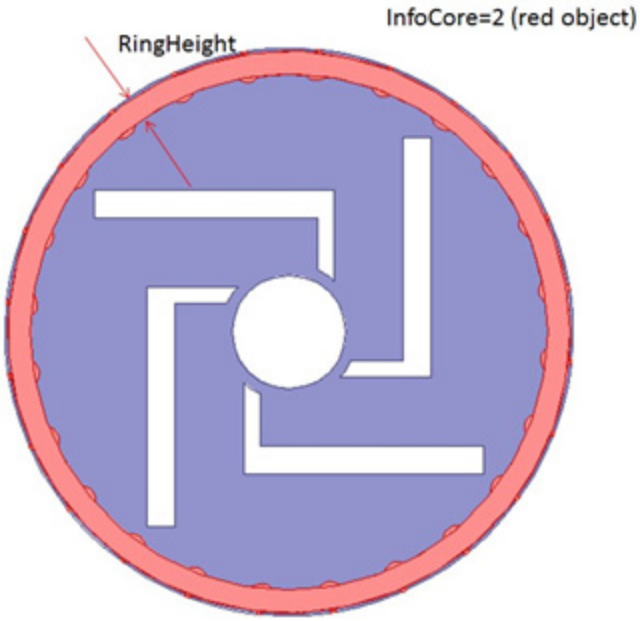


Figure 9-67 Damper of PMDamperCore (red)

Figure 9-68

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

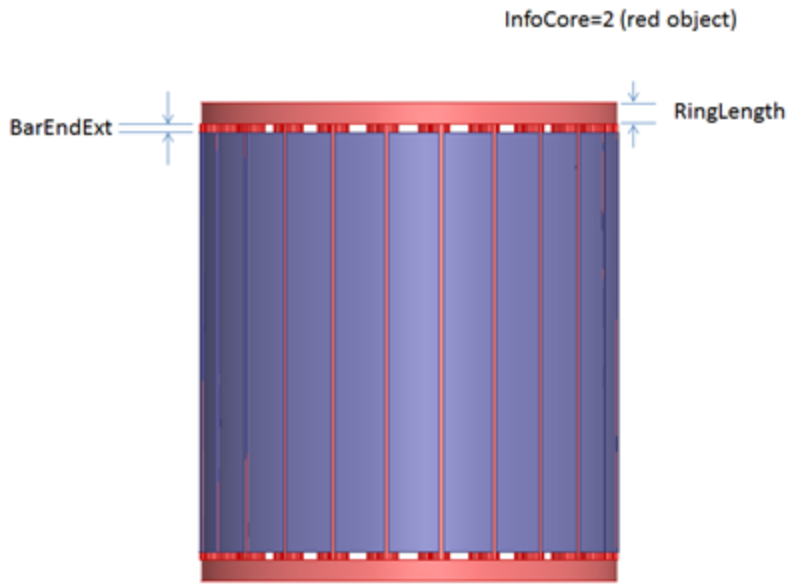


Figure 9-69 3D Damper of PMDamperCore (red)

Creating an Inner PM Damper Core

Set the value of **DiaYoke** and **DiaGap** as **DiaYoke** < **DiaGap** to create an inner PM damper core. You can set the value either when creating the core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Set the **InfoCore** value to 0.

An example is given in the following figure.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

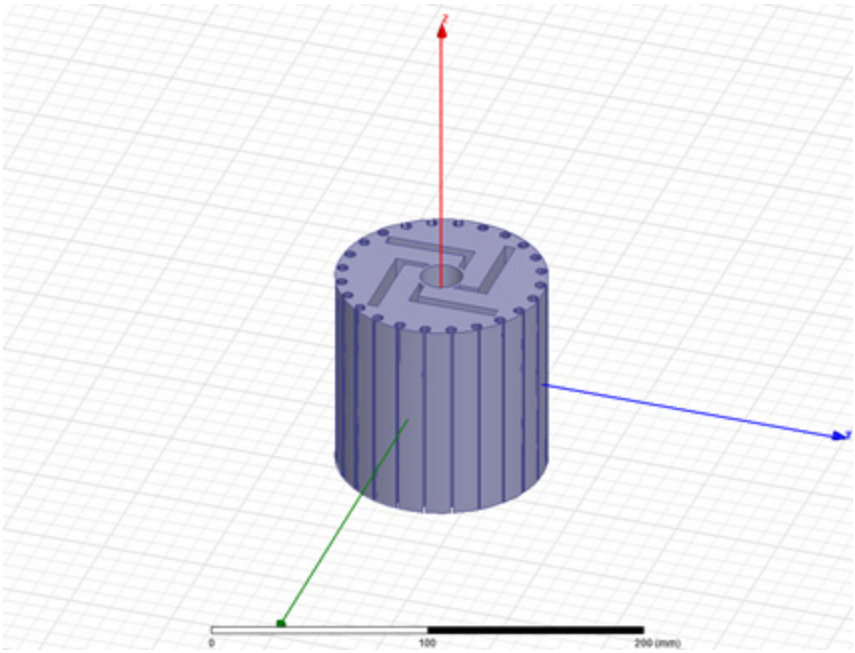


Figure 9-70 Inner PM Damper Core

Creating a Damper (PM Damper Core)

You create a damper of a PM damper core manually by setting the value of **InfoCoil** to 2.

You can assign the value of this property either when creating a PM damper core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Examples are shown in " 3D Damper of PMDamperCore (red)" on the previous page as a red object and "Damper" on the next page.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

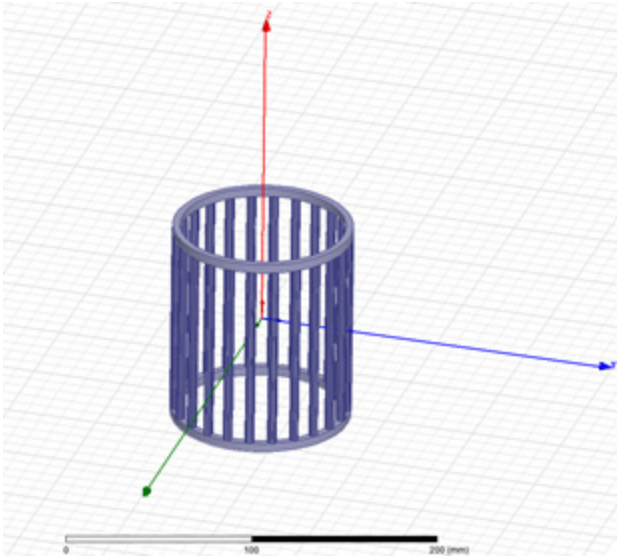


Figure 9-71 Damper

Creating All Magnets (PM Damper Core)

You can create all magnets of a PM damper core manually by setting the value of **InfoCoil** to 1.

You can assign the value of this property either when creating a PM damper core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

An example is shown in the following figure. Note that this example is realized when the value of **PoleType** is 3.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

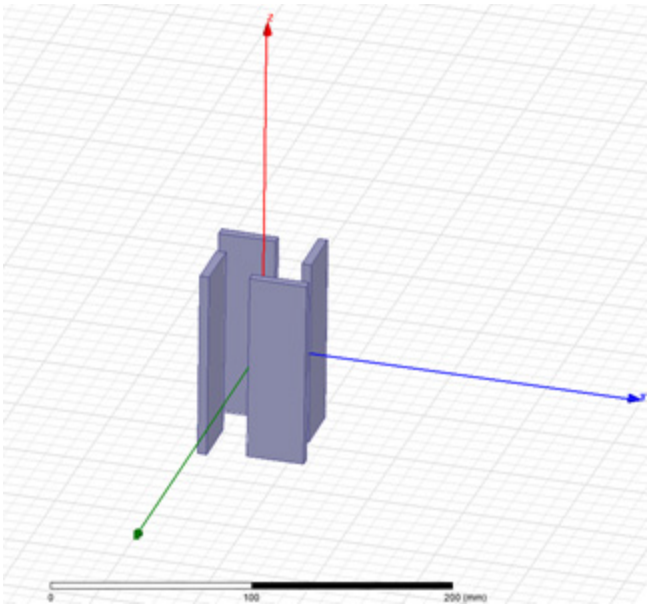


Figure 9-72 All Magnets

Creating a Magnet (PM Damper Core)

You can create a magnet of a PM damper core by setting the **InfoCore** value to -1, -2 or -3 according to the number of magnets for each pole of the core.

For instance, when **Pole Type** is equal to 3 (which is shown in " All Magnets" above), there is only one magnet for a pole. In this situation, setting the value of **InfoCore** to be -1 can create a magnet. An example is shown in the following figure.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

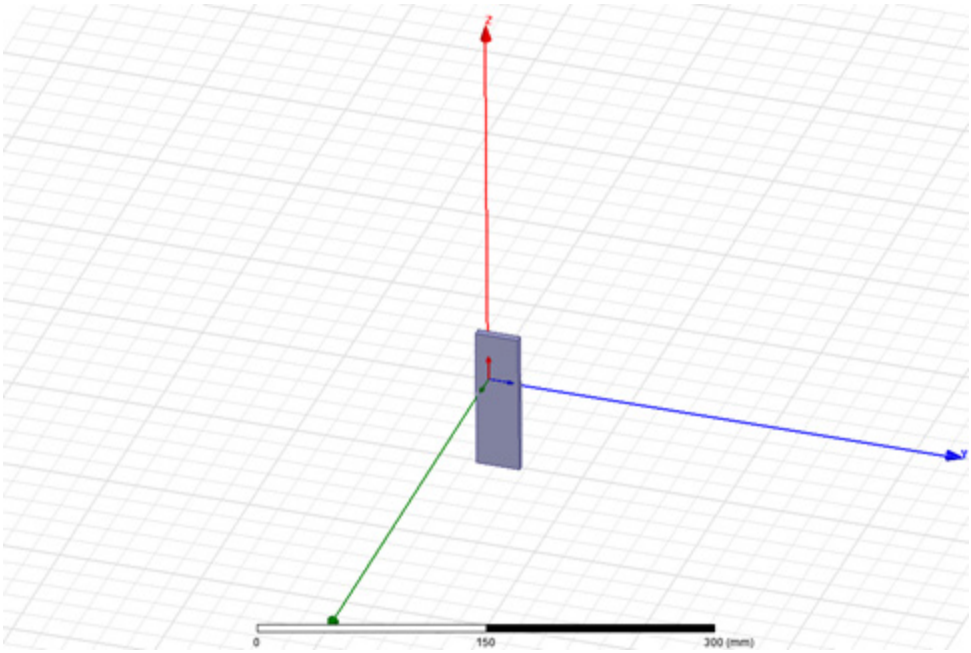


Figure 9-73 Magnet with Pole Type = 3

When there are two magnets for a pole of a PM damper core (for example, when **Pole Type** is 5), setting the value of **InfoCore** to be -1 can create one of these two magnets while setting the value of **InfoCore** to be -2 can create another one.

For instance, " PM Damper Core with Pole Type = 5, InfoCore = 1" on the facing page shows all magnets of a PM damper core when **Pole Type** is 5, whereas " PM Damper Core with Pole Type = 5, InfoCore = -1" on the facing page and " PM Damper Core with Pole Type = 5, InfoCore = -2" on page 9-166 present one of two magnets for a pole, by setting **InfoCore** value to be -1 and -2 respectively.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

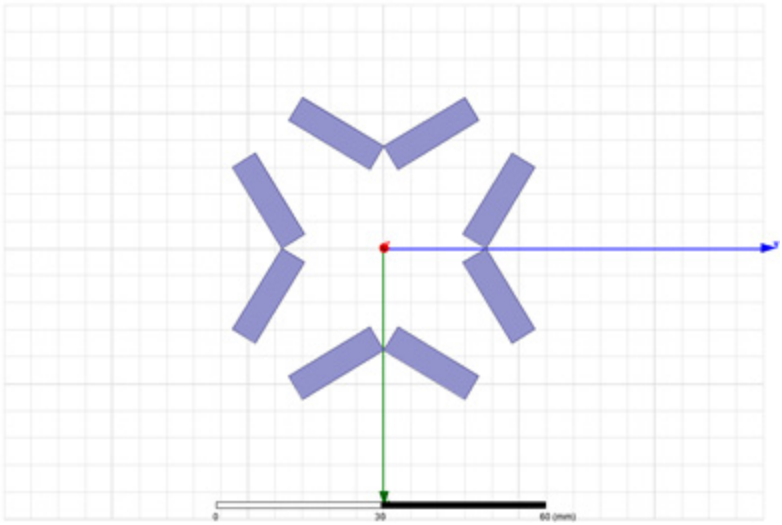


Figure 9-74 PM Damper Core with Pole Type = 5, InfoCore = 1

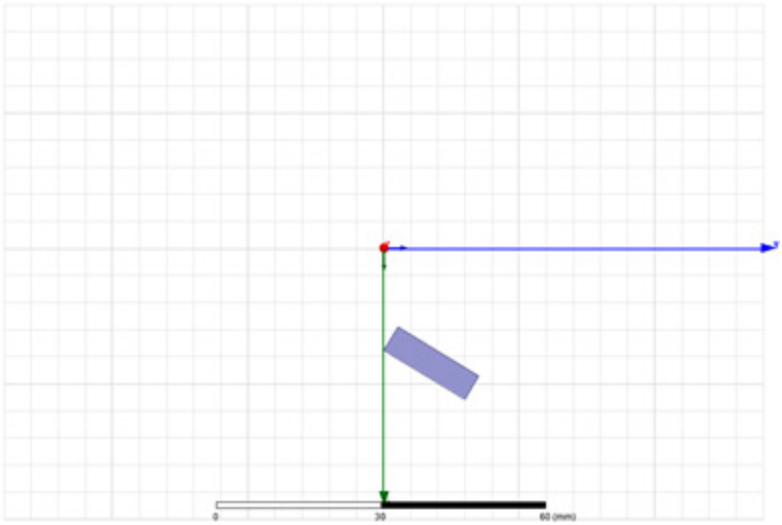


Figure 9-75 PM Damper Core with Pole Type = 5, InfoCore = -1

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

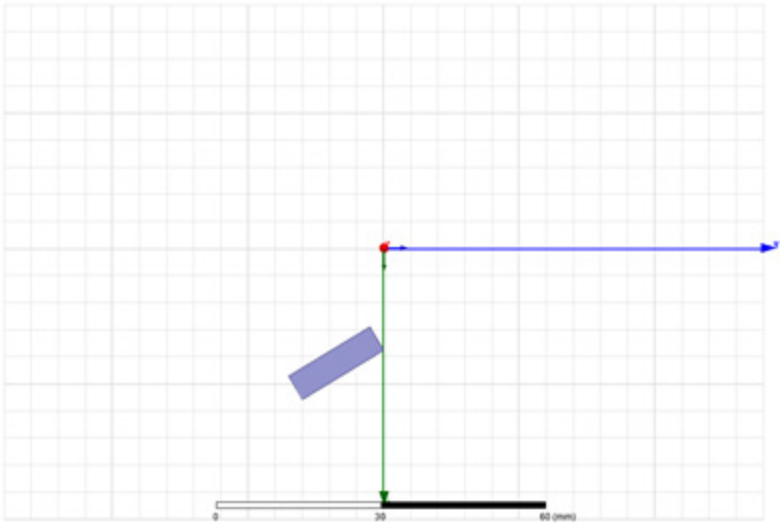


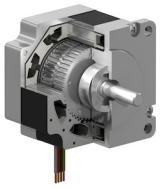
Figure 9-76 PM Damper Core with Pole Type = 5, InfoCore = -2

Similarly, when a pole of PM damper core has 3 or more magnets (such as when **InfoCore** is 6), you can create a single magnet by setting the value of InfoCore to -1, -2, -3 and so on.

You can assign the value of these properties either when creating a PM damper core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

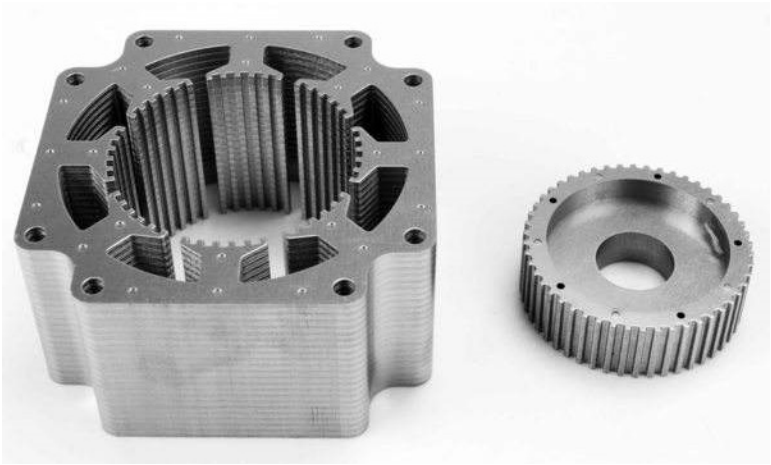
StepMCore UDP

The StepMCore UDP is used to create stator and rotor cores for Variable Reluctance (VR), or Hybrid, stepper motors, as shown as in the first figure for a stepper motor and in the second figure for the stator and rotor.



Hybrid Stepper Motor

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Stator and Rotor of a Stepper Motor

Parameter Definitions

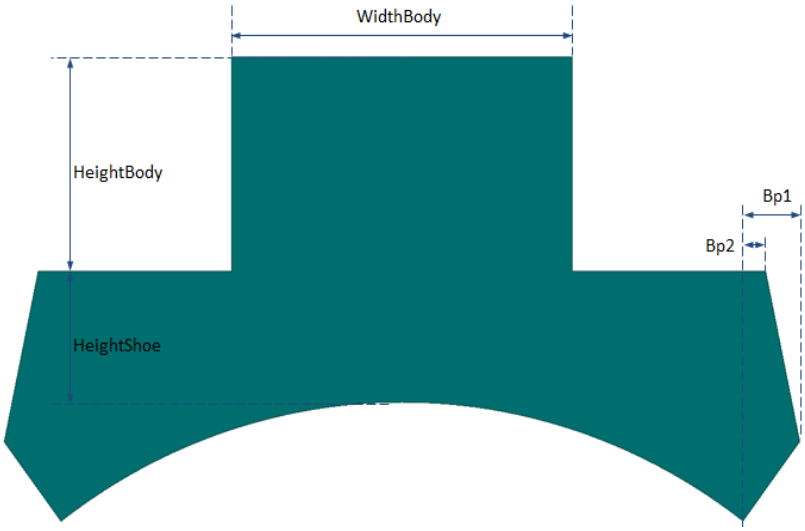
Property	Description
DiaGap	Core diameter on gap side, DiaGap < DiaYoke for outer cores
DiaYoke	Core diameter on yoke side, DiaYoke < DiaGap for inner cores
Length	Core Length
Poles	Number of poles
PoleType	Pole type: 0, 1, or 2 (not valid for inner cores)
EmbraceShoe	Pole shoe embrace related to pole pitch for WidthShoe computation
HeightShoe	Pole shoe height at pole center
WidthBody	Pole body width
HeightBody	Pole body height
Bp1/Rp1	Bp1 for pole type 1; or Rp1 for pole type 2; not valid for type 0
Bp2/Rp2	Bp2 for pole type 1; or Rp2 for pole type 2; not valid for type 0
SurfaceTeeth	Number of surface teeth per pole (>=1)
IndexingTeeth	Number of indexing teeth for tooth pitch computation, not valid for SurfaceTeeth=1
EmbraceTooth	Tooth embrace related to tooth pitch, not valid for SurfaceTeeth=1
HeightTooth	Surface tooth height, or slot depth, not valid for SurfaceTeeth=1
FilletSlot	Fillet radius at slot bottom
GapCore	Gap of two core sections in radial direction (>0 for Hybrid core)
ThickYoke	Thickness of solid yoke in radial direction for Hybrid core
ThickMag	Magnet thickness in radial direction for Hybrid core
LengthMag	Magnet length in axial direction for Hybrid core (>=GapCore)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Property	Description
CoilEndExt	One-side coil end extended Length
LenRegion	Region length
InfoCore	0: core; 1: core & coils; 2: coil; 3: magnet; 4: terminal1; 5: terminal2; 6: poles; 7: yoke; 100: region

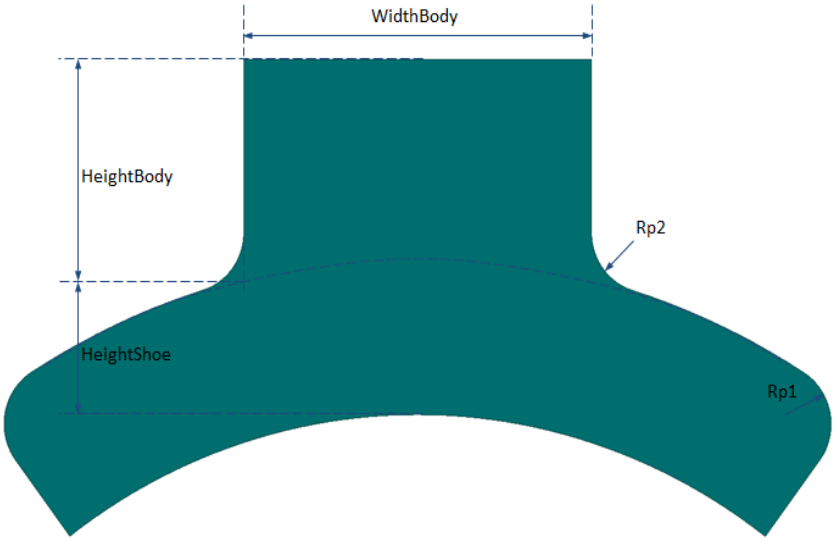
Main parameters described in the table above for different pole types are shown in the figures below.

Pole Parameters for PoleType = 0

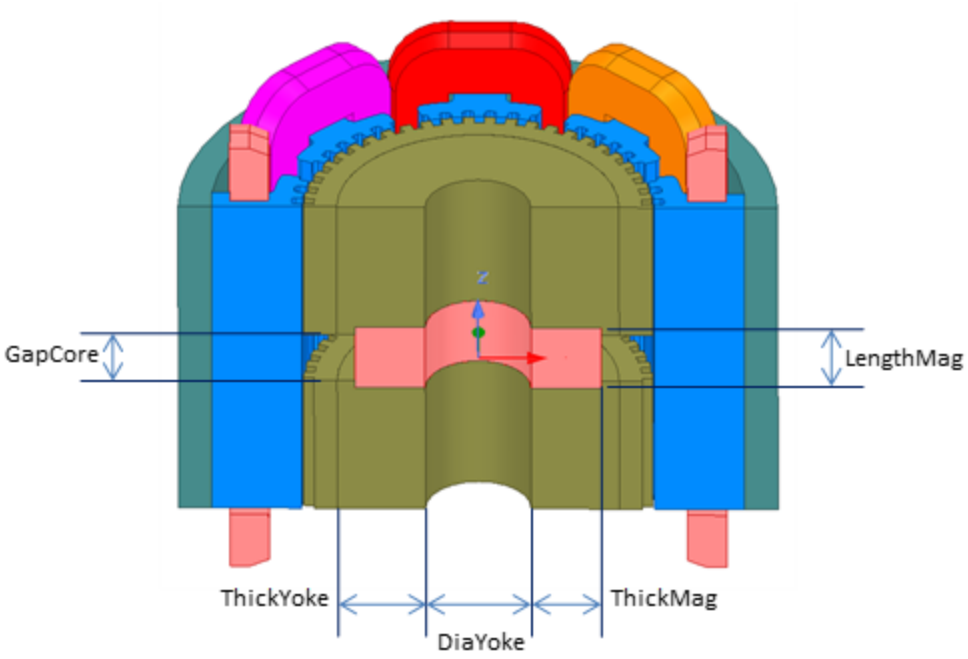


Pole Parameters for PoleType = 1

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Pole Parameters for PoleType = 2



Pole Parameters for Hybrid Rotor

EmbraceShoe is the ratio of the pole shoe angle to pole pitch, and **EmbraceTooth** defines the ratio of a tooth angle to tooth pitch is: ToothPitch = 360°/**IndexingTeeth**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

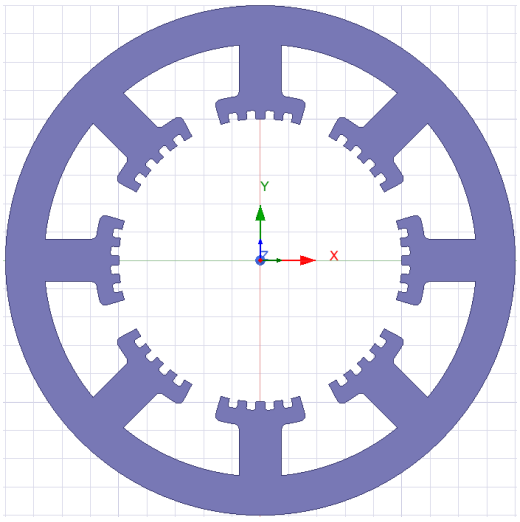
When **SurfaceTeeth** = 1, which means there are no slots on the shoe surface, the shoe width, noted as WidthShoe, will be directly obtained from **EmbraceShoe**. When **SurfaceTeeth** > 1, the minimum shoe angle AngShoeMin will be obtained from the minimum angle to be able to create the specified number of surface teeth, that is

AngShoeMin = (**SurfaceTeeth** – 1 + **EmbraceTooth**) x ToothPitch

and the maximum shoe angle is defined to add a half slot in each shoe side, that is

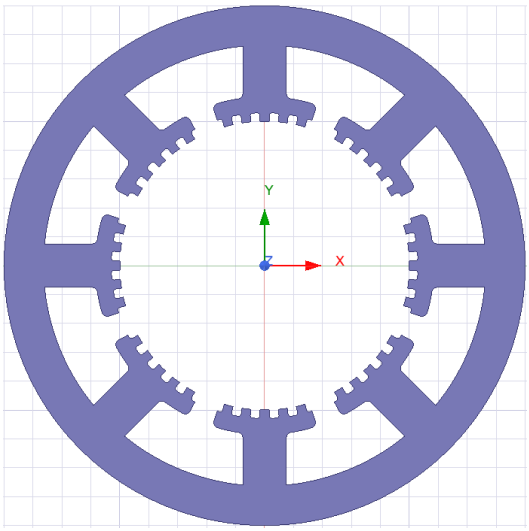
AngShoeMax = AngShoeMin + (1 – **EmbraceTooth**) x ToothPitch = **SurfaceTeeth** x ToothPitch.

When the shoe angle corresponding to the specified **EmbraceShoe** is out of the range of (AngShoeMin, AngShoeMax), the UDP will automatically adjust the value of **EmbraceShoe**, as shown below.



EmbraceShoe = 0.7

(Automatically adjust to 0.72)



EmbraceShoe = 0.9

(Automatically adjust to 0.8)

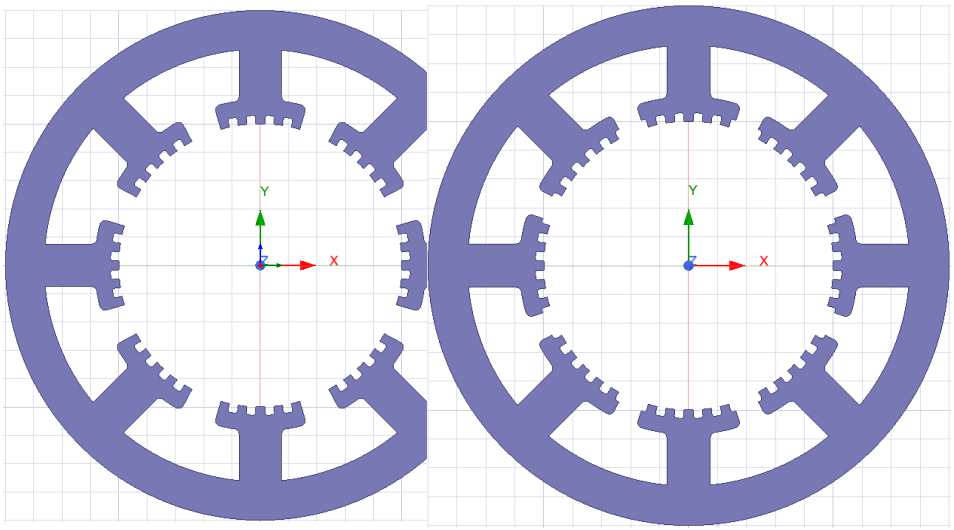
In this example, an 8-pole stepper motor stator has **SurfaceTeeth** = 5, **IndexingTeeth** = 50 and **EmbraceTooth** = 0.5. Therefore, ToothPitch = 7.2°, and AngShoeMin and AngShoeMax will be 32.4° and 36°, corresponding to minimum and maximum shoe embrace of 0.72 and 0.8, respectively. The value of **EmbraceShoe** will automatically be adjusted to 0.72, or 0.8, if the input value is out of the range.

Creating an Inner or Outer Core for a Variable Reluctance Stepper Motor

You can set the value of **DiaYoke** and **DiaGap** as **DiaYoke** < **DiaGap** to create inner cores or **DiaGap** < **DiaYoke** for outer cores.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Keeping **GapCore** = 0 and **InfoCore** = 0, outer stator cores of a variable reluctance stepper motor with different **EmbraceShoe** are created, as shown in the figures below:

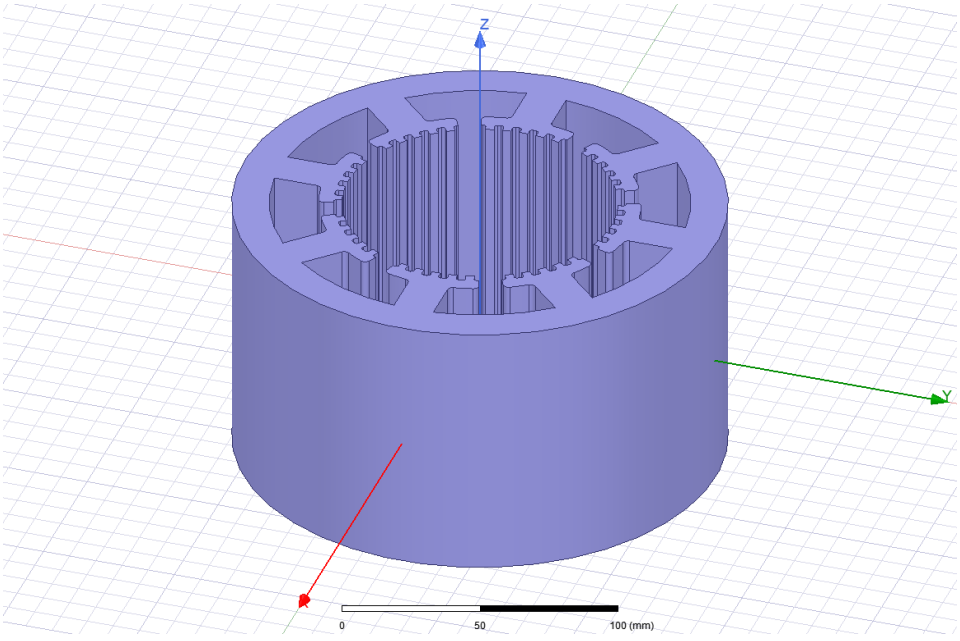


EmbraceShoe = 0.7

(Automatically adjust to 0.72)

EmbraceShoe = 0.9

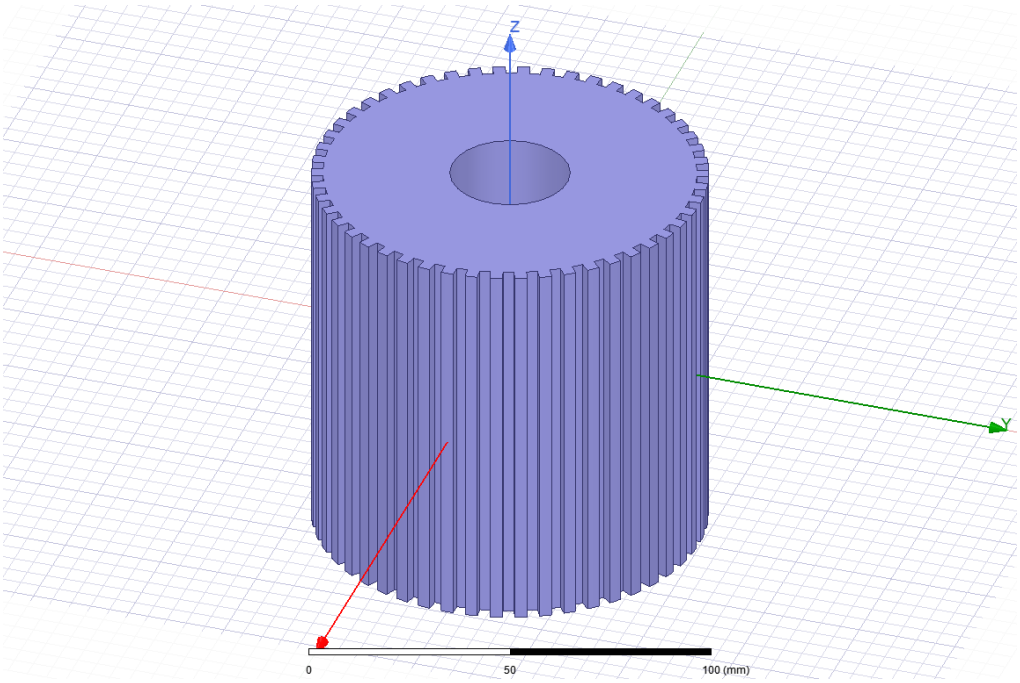
(Automatically adjust to 0.8)



Outer Stator Core in 3D (PoleType = 2)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A 3D inner rotor core with 50 poles, which is the same number as the stator indexing teeth, is shown below:



Inner Rotor Core in 3D (PoleType = 0)

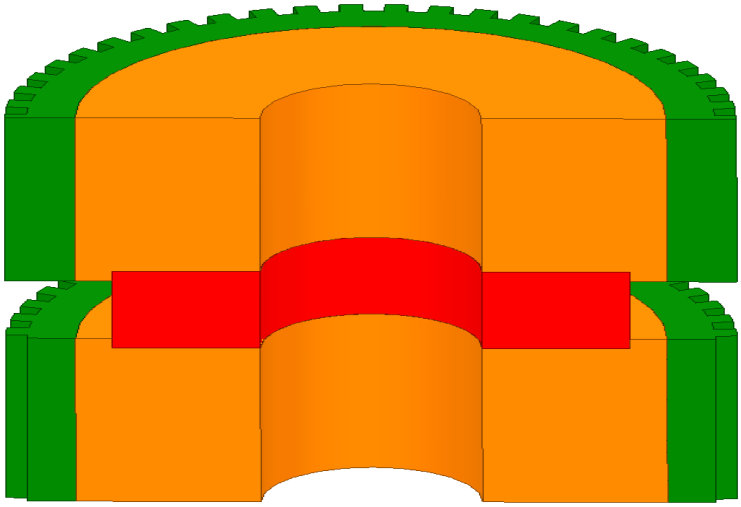
Creating an Inner Rotor Core for a Hybrid Stepper Motor

The stator structure of a hybrid stepper motor is similar to that of a variable reluctance stepper motor, but the rotor structure is quite different. The rotor core is divided into two separate parts. The two parts are shifted by a half tooth pitch in circumferential direction, as shown below. The top pole part, in green color created with **InfoCore** = 6, is cut by the symmetry plan through the pole center, or the tooth center, however, the bottom pole part is cut through the slot center.

The rotor core could be laminated, solid, or laminated for poles and solid for yoke. In the example, the poles, in green color, are laminated, and the yoke, in golden color created with **InfoCore** = 7, is solid. The magnet, in red color created with **InfoCore** = 3, is magnetized in vertical direction. The fluxes in both top and bottom parts are in the radial direction, one from rotor to stator, and the other from stator to rotor.

The hybrid rotor can be created only in 3D.

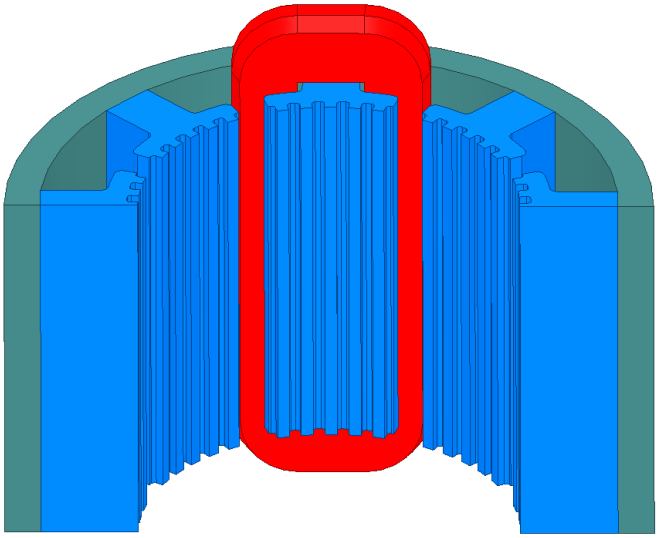
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Inner Rotor Core in 3D (PoleType = 0)

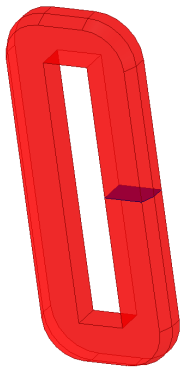
Creating a Coil and Coil Terminals for a Stepper Motor

A coil can be created by setting **InfoCoil** = 2, as shown below,



and a coil terminal can be created by setting **InfoCoil** = 4 or 5. (Coil terminal created by **InfoCore** = 4 shown below.)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

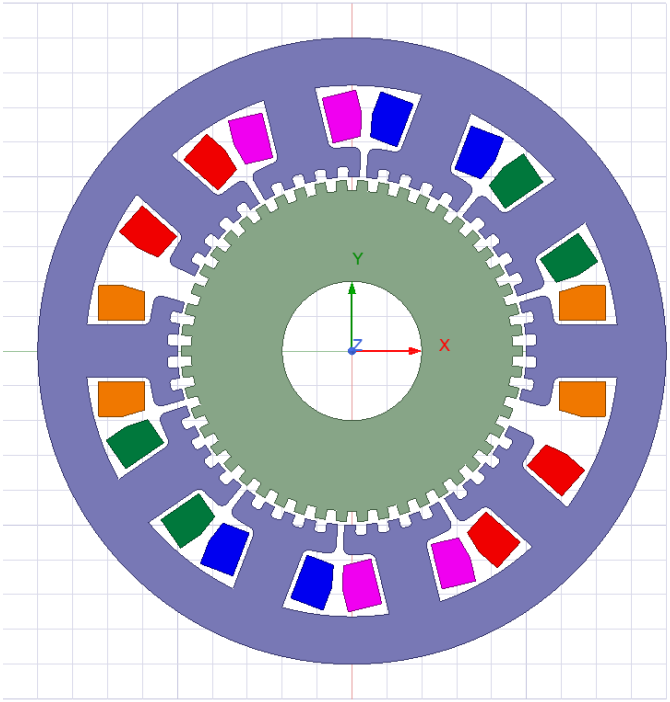


In 2D, you can create a coil by setting **InfoCoil** = 2 with two united terminals. You can also create two separate terminals by setting **InfoCoil** = 4 and 5 for convenient coil setup.

Creating a Stator Core with Non-Uniform Pole Pitch

To be able to create torque when stator windings are excited phase by phase (a phase winding normally consists of two coils in two poles shifted by 180 degrees), the number of rotor poles should be the same as the number of indexing teeth for the stator pole surface teeth, and the number of indexing teeth should not be integer multiples of the number of stator poles. However, in some applications, the number of indexing teeth may be integer multiples of the number of stator poles. For example, in widely used 5-phase stepper motors, the stator contains 10 poles, while the rotor has 50 poles, or teeth. The tooth pitch in both stator and rotor is 7.2°. In order to be able to produce continuous torque, the 5 stator pole pairs should be shifted from their uniform positions by 0, 1.44, 2.88, 4.32, 5.76, successively.

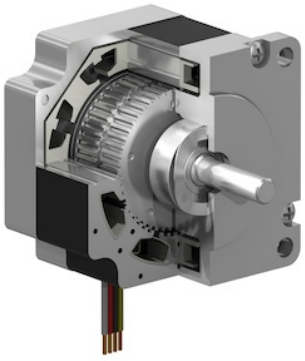
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

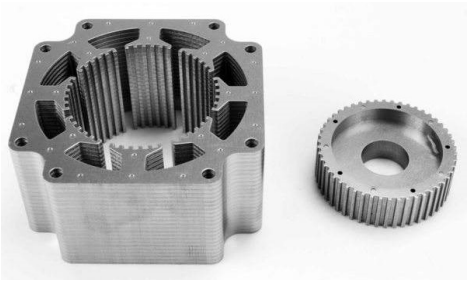


2D Stator Core and Coils with Non-Uniform Pole Pitch

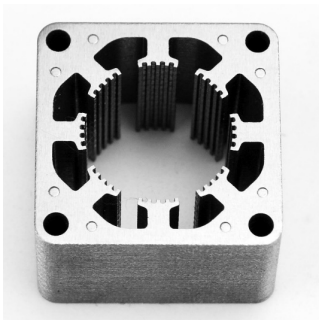
Square Yoke UDP

The SquareYoke UDP is used to create outer square yokes, which can be combined with various stator types to be able to directly mount with end caps for small-size machines, as shown in the examples below.





Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

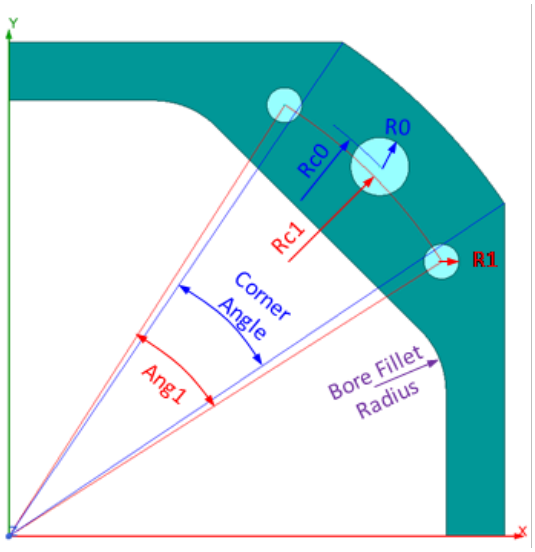


Parameter Definitions

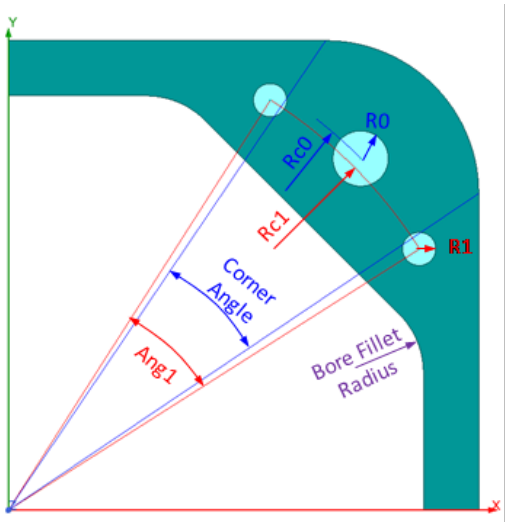
Property	Description
WidOuter	Outer width of square profile
DiaInner	Inner diameter or minimum width
Length	Yoke Length
YokeType	0: Circle; 1: Square Circle; 2: Square Bulged; 3: Square Flat; 4: Square Dented
AngCorner	Corner angle related to bore center, not valid for CornerType = 0
BoreType	1: Circle; 2: Square; 3: Octagon; BoreType is always 1 for CornerType = 0
BoreFillet	Inner bore fillet radius, for square or octagon bore type only
R0	Radius of mounting hole or dented fillet, not valid for CornerType = 0
Rc0	Radius of mounting hole location or bulged fillet, not valid for CornerType = 0
R1	Radius of two corner holes, not valid for CornerType = 0
Rc1	Radius of corner hole location, not valid for CornerType = 0
Ang1	Angle of two corner-hole centers related to bore center, not valid for CornerType = 0
LenRegion	Region length
InfoCore	0: core; 1: holes; 2: 100: region

Main parameters described in the table above for the four yoke corner types are shown below, where "Corner Angle" is **AngCorner**, and "Bore Fillet Radius" is **BoreFillet**.

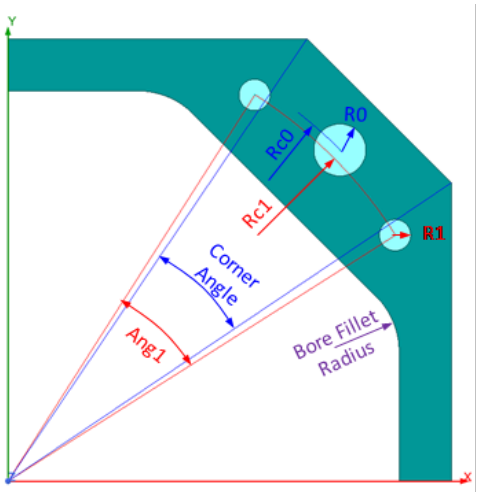
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



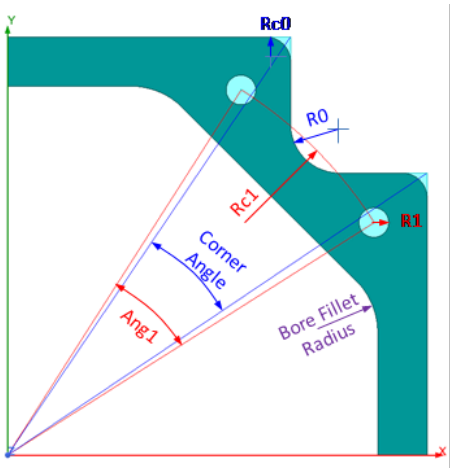
Yoke parameters for YokeType = 1



Yoke parameters for YokeType = 2



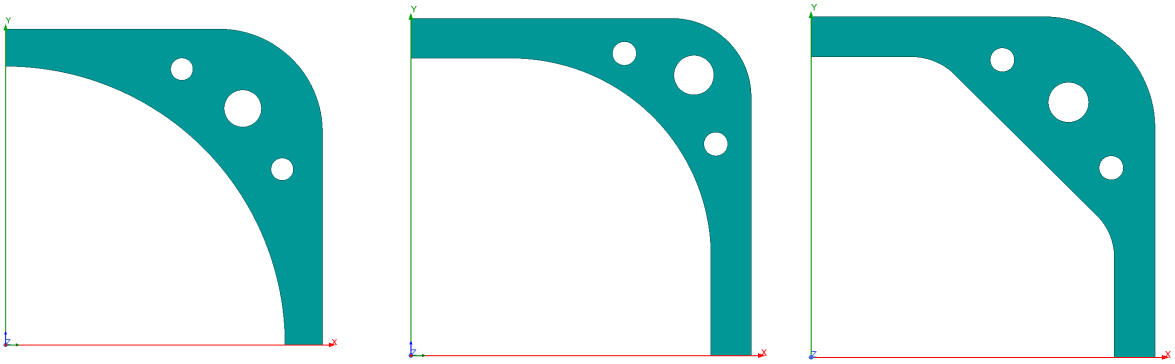
Yoke parameters for YokeType = 3



Yoke parameters for YokeType = 4

The three bore types are shown below.

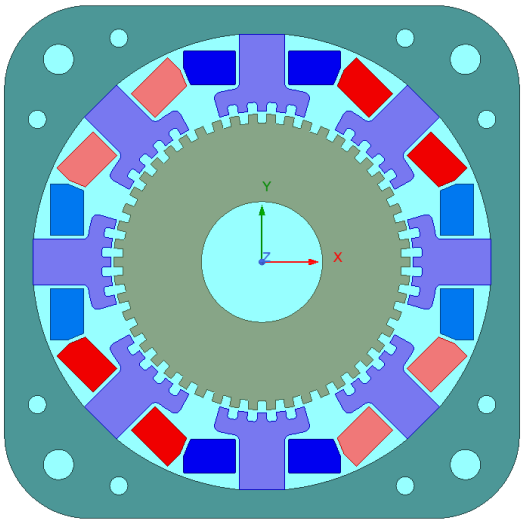
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



BoreType = 1 BoreType = 2 BoreType = 3

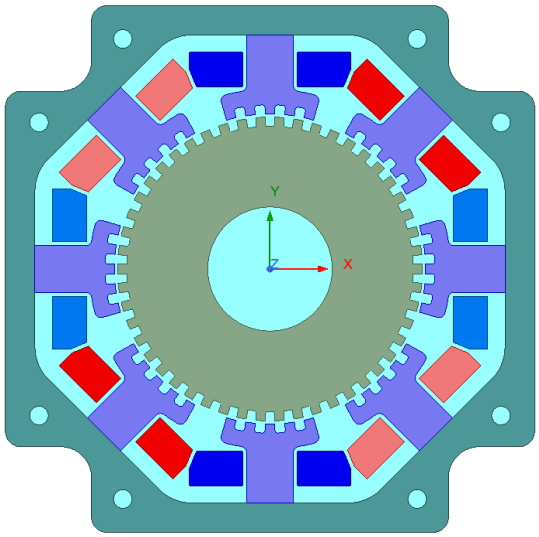
Creating a Stator Yoke to Match Stator Poles of Stepper Motors

The figure below shows an example for a variable reluctance stepper motor in which the stator poles are mounted to a square yoke with **YokeType**= 2 and **BoreType**= 1, where **BoreFillet** is set as **WidthBody** in StepMCore UDP to fit the pole bottom.

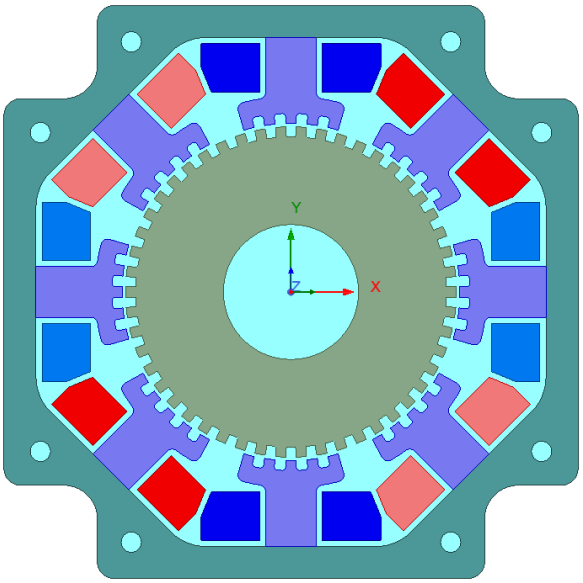


The figure below shows a stepper motor in which the stator poles are mounted to a square yoke with **YokeType**= 4 and octagon **BoreType**= 3 in which the coil area is that of circular bore type (**BoreType**= 1) as shown in the previous figure.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For bore type (**BoreType**= 3), the coil area can be larger. To do this, you can set a negative value for StepMCore UDP parameter **CoilEndExt** to create the coils. The modified coils are shown below.



Coil area modified by setting StepMCore UDP parameter **CoilEndExt** < 0.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

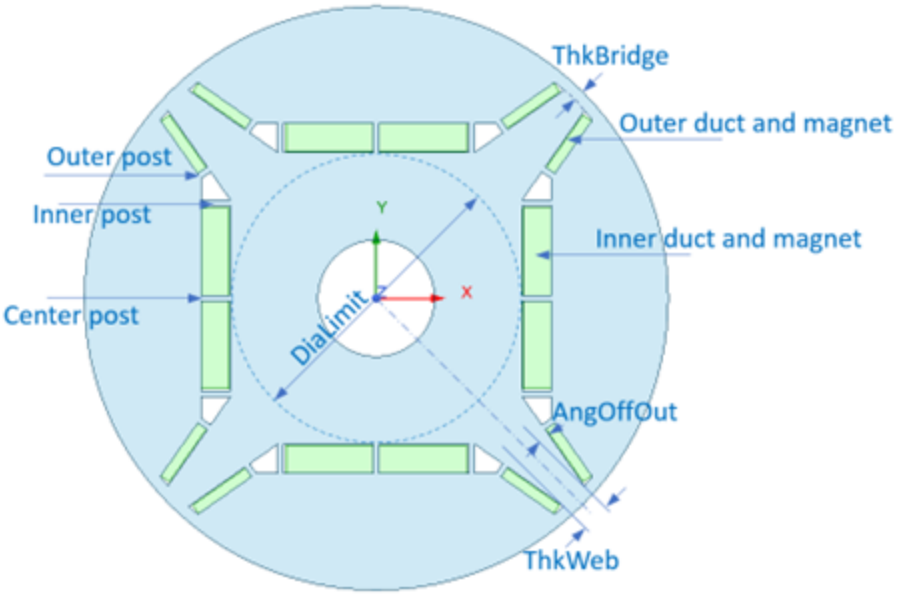
UPMCore UDP

The UPMCore UDP is used to create a single- or poly- layer U-type IPM core and PMs for IPM synchronous machines, or IPM brushless DC motors.

Property	Description
DiaGap	Core diameter on gap side, or outer diameter
DiaYoke	Core diameter on yoke side, or inner diameter
Length	Core length
Poles	Number of poles
DiaLimit	Inner limited diameter of PM ducts
ThkBridge	Bridge thickness
ThkWeb	Web thickness
AngOffOut	Outer (web) angle offset
ThkDuctOut	Outer duct thickness
ThkDuctIn	Inner duct thickness
ThkPostOut	Outer post thickness
ThkPostIn	Inner post thickness, <0 for duct bottom corner radius
ThkPostMid	Center post thickness
WidMagOut	One-side width of outer magnets
GapMagOut	Two-side gap of outer magnets (ThkMagOut = ThkDuctOut – GapMagOut)
WidMagIn	Total width of inner magnets per pole
GapMagIn	Two-side gap of inner magnets (ThkMagIn = ThkDuctIn – GapMagIn)
LenRegion	Region length
InfoCore	0: core; 1: magnets; 2: ducts; 3: inner magnet; 4: left magnet; 5: right magnet; 100: region

Most parameters for a UPMCore UDP are shown in the figure below except common parameters like **DiaGap**, **DiaYoke**, and **Length**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Parameters of a UPMCore UDP with AngOffOut = -10°.

Note that when **AngOffOut** = 0, the edges of outer PM ducts between two adjacent poles are parallel. When **AngOffOut** > 0, the inner web thickness is smaller than the outer web thickness which is labeled as **ThkWeb**. When **AngOffOut** < 0, the inner web thickness is larger than the outer web thickness, as shown above.

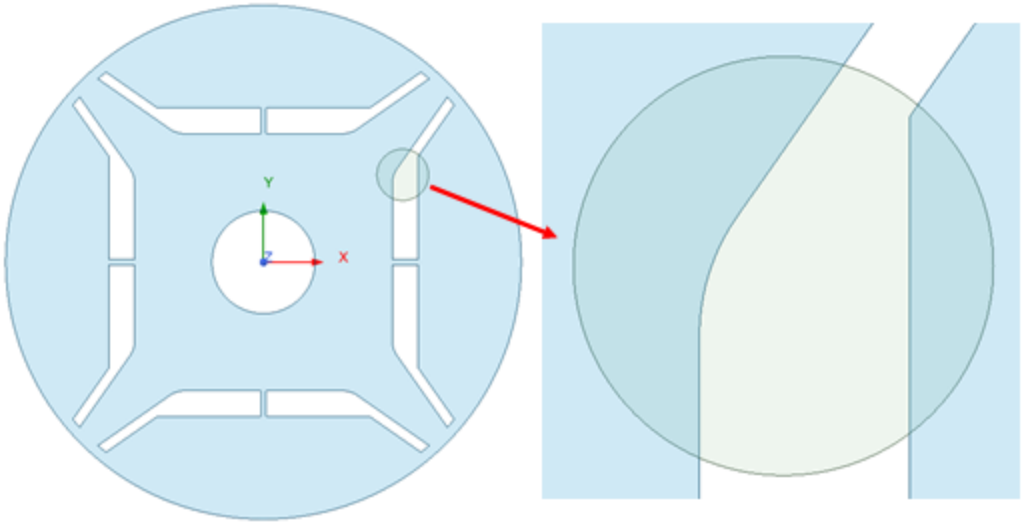
Creating a U-Type IPM Core

You can create a U-type IPM core with **InfoCore** = 0.

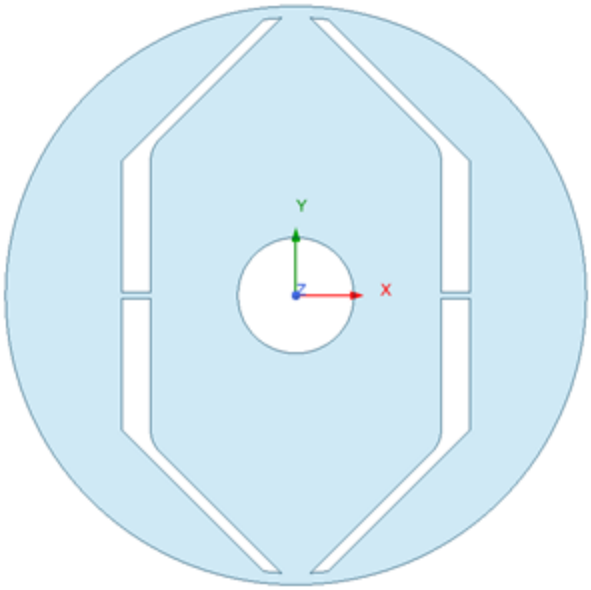
With the default settings of the UPMCore UDP, the created U-type IPM core is shown below.

You can input a negative value for **ThkPostIn**. When **ThkPostIn** < 0, its absolute value, Rc = abs(**ThkPostIn**), is the radius of a fillet at the bottom corner of the PM ducts. The upper limited value of Rc is the maximum value of thickness of the inner and outer ducts. That is, the center point of the fillet arc will always within PM ducts. When **ThkPostIn** < 0, there will be no outer post, that is, the value of **ThkPostOut** will be automatically changed to 0.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can also create a U-type IPM core of two poles. The following figure is an example of a two-pole U-type IPM core, where the setting value of **AngOffOut** is -45.

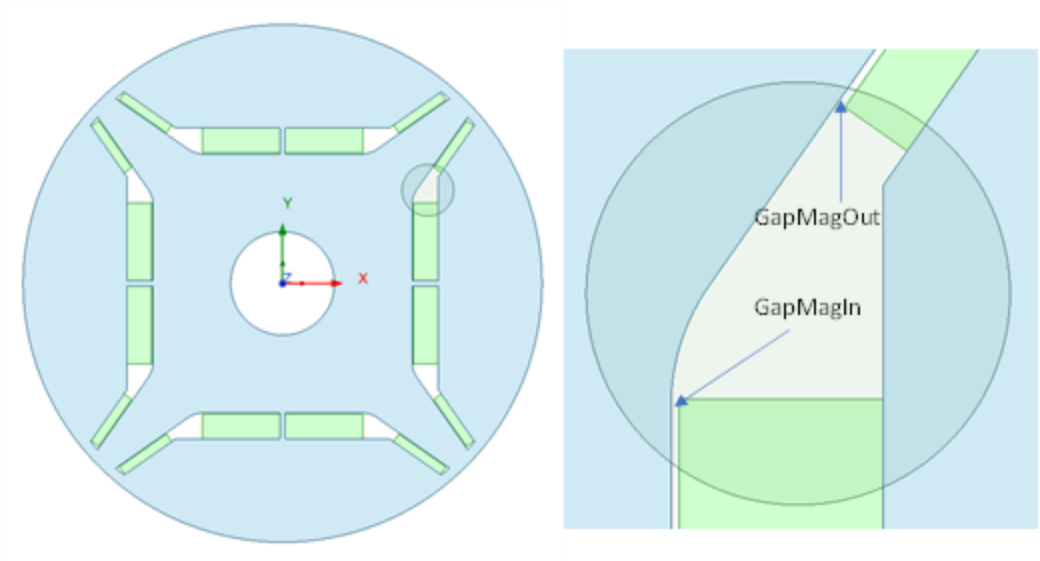


Creating Magnets (UPM Core)

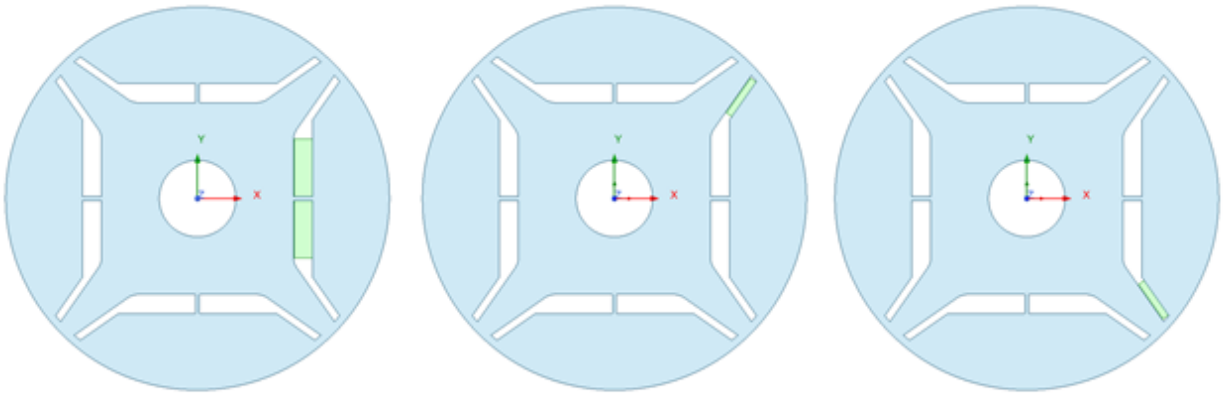
All magnets of a U-type IPM Core are created by setting the value of **InfoCore** to 1, as shown below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notice that **GapMagOut** and **GapMagIn** include all gaps of both top and bottom sides. If both top and bottom gaps are 0.1 mm, the created magnets will be located inside the PM ducts in such a way that the top gap is 0, and the bottom gap is 0.2 mm. This will not affect the performance solutions. However, this enhances the mesh quality greatly.



To easily assign the direction of magnet excitation, magnets can be created one-by-one by setting the value of **InfoCore** to 3, 4, or 5. The setting of **InfoCore** = 3 is used to create inner magnet(s) as one object no matter whether it is separated by the center post or not, and the value of **WidMagIn** stands for the total width of inner magnet(s). **InfoCore** = 4 is used to create the left-side magnet, and **InfoCore** = 5 to create the right-side magnet, as shown below.

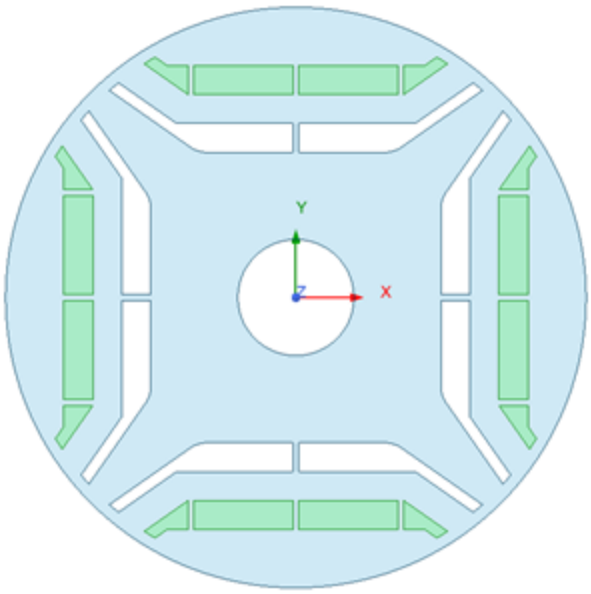


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating PM Ducts (UPM Core)

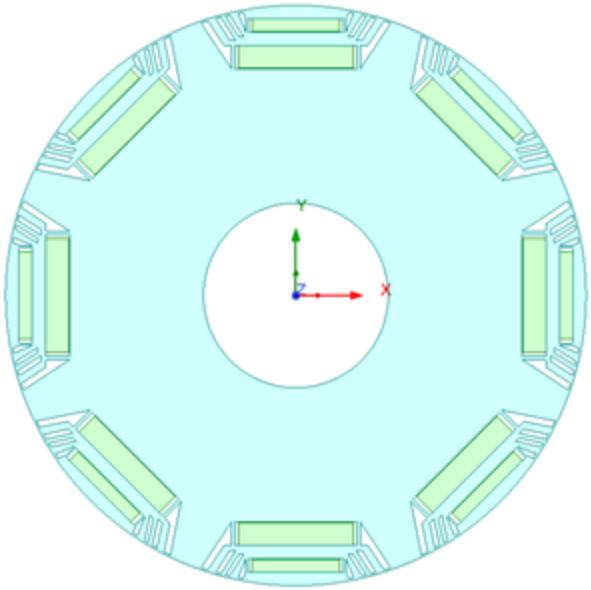
All PM ducts of a U-type IPM Core are created by setting the value of **InfoCore** to 2. Subtracting a PM duct set, the software can create a poly-layer U-type IPM core.

A U-type PM duct set, together with a U-type IPM core, is shown below. The parameters of the additional ducts can be totally different from those of the original core.



If you subtract the additional PM duct set from the core, you can get a core with two-layer PM ducts. Continuing to create and subtract additional PM duct sets, the software can create an IPM core with an arbitrary number of layers of PM ducts. The figure below shows an IPM core with four-layer PM ducts. Notice that when **ThkDuctIn** = 0, the PM ducts will have only the outer parts, and when **ThkDuctOut** = 0, the PM ducts will have only the inner parts.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



VPMCore UDP

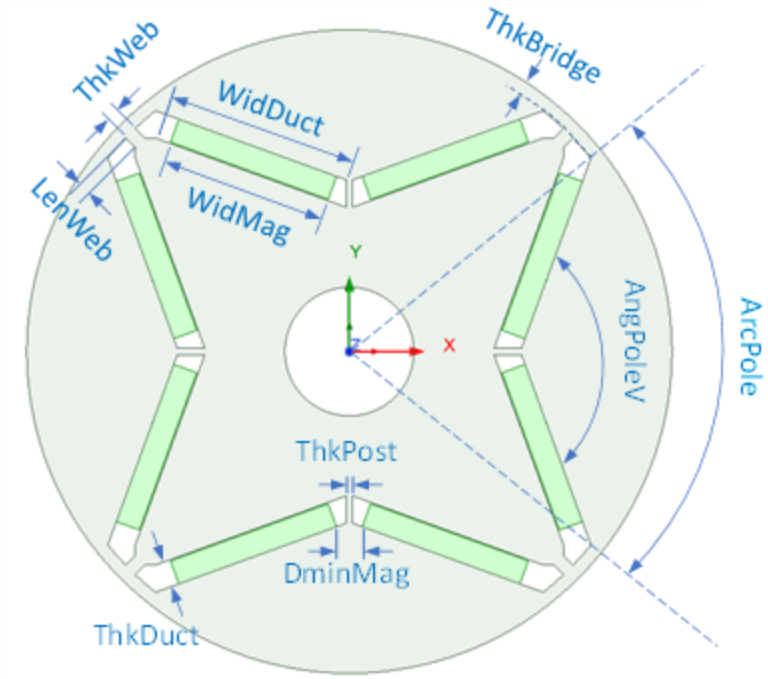
The VPMCore UDP is used to create a single- or poly-layer V-type IPM core and PMs for IPM synchronous machines, or IPM brushless DC motors.

Property	Description
DiaGap	Core diameter on gap side, or outer diameter
DiaYoke	Core diameter on yoke side, or inner diameter
Length	Core length
Poles	Number of poles
PoleType	0: web; 1: simple
ThkBridge	Bridge thickness
ThkDuct	Magnet duct thickness
WidDuct	Magnet duct width
ThkWeb/ExtOut	Web thickness (web type), or outer extension (simple type)
LenWeb/ExtIn	Web length (web type), or inner extension (simple type)
AngPoleV	Pole-V angle
ArcPole	Pole arc, in electric degrees, for web pole type only
ThkPost	Center post thickness, < 0 for duct bottom corner radius (web type)
DminMag	Minimum distance between two-side magnets
WidMag	One-side width of magnets

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

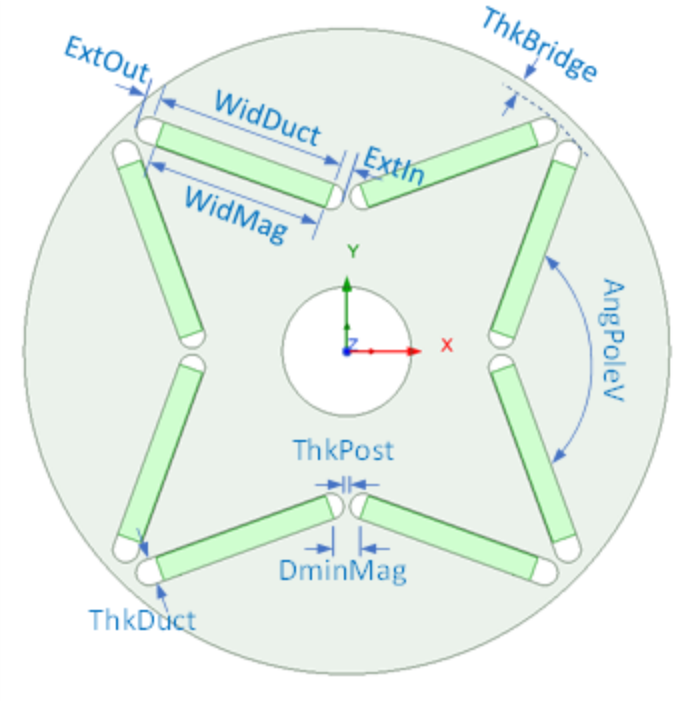
Property	Description
GapMag	Two-side gap of magnets (ThkMag = ThkDuct – GapMag)
LenRegion	Region length
InfoCore	0: core; 1: magnets; 2: ducts; 3: left magnet; 4: right magnet; 100: region

Most parameters for a VPMCore UDP are shown in the figures below except common parameters like **DiaGap**, **DiaYoke**, and **Length**.



Parameters of VPMCore UDP (PoleType = 0).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Parameters of VPMCore UDP (PoleType = 1).

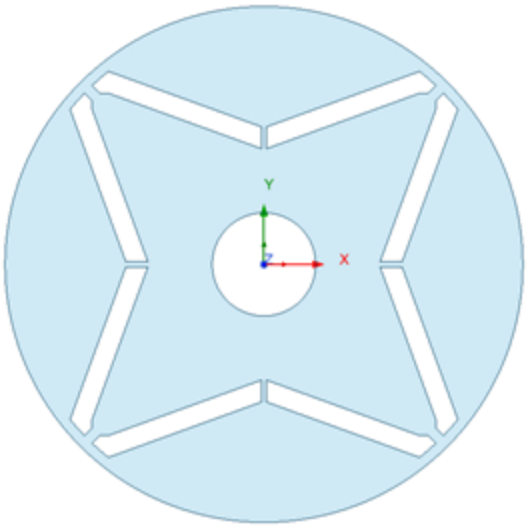
Creating a V-Type IPM Core

You can create a V-type IPM core by setting **InfoCore** = 0.

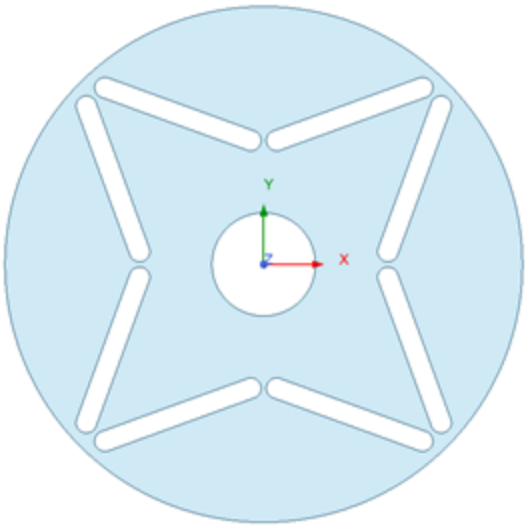
With the default settings of the VPMCore UDP, the created V-type IPM cores of web type (**PoleType** = 0) and simple type (**PoleType** =1) are shown below.

For a web-type core, you can input a negative value for **ThkPost**. When **ThkPost** < 0, its absolute value, Rc = abs(**ThkPost**), is the radius of a fillet at the bottom corner of the PM ducts. The upper limited value of Rc is the thickness of the PM ducts. For a simple-type core, the upper limit value for **ExtOut** and **ExtIn** is a half thickness of PM ducts.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



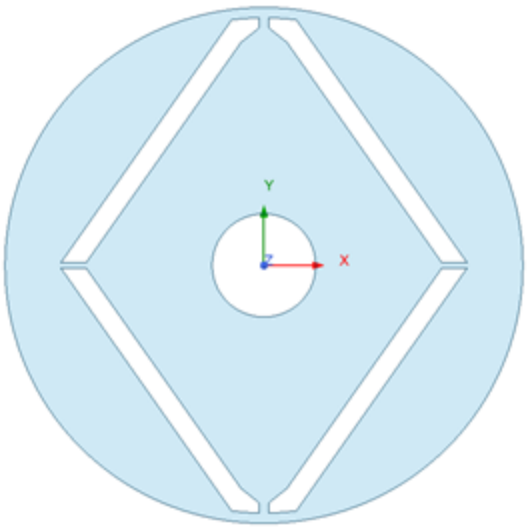
Web type (**PoleType** = 0)



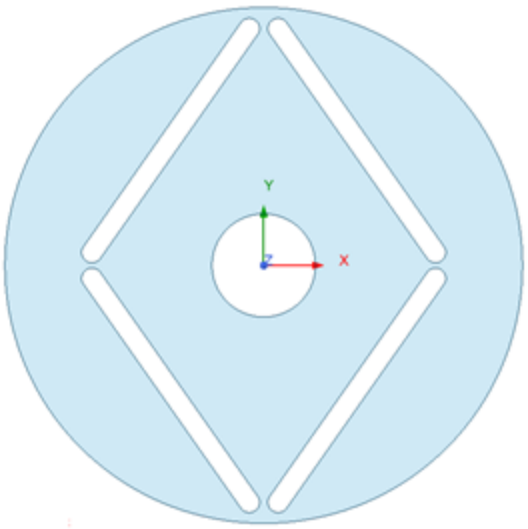
Simple type (**PoleType** = 1)

4-Pole V-Type IPM Cores

The software can create a V-type IPM core of two poles. The figure that follows is an example of 2-pole V-type IPM core, where the value of **AngPoleV** is 250.



Web type (**PoleType** = 0)



Simple type (**PoleType** = 1)

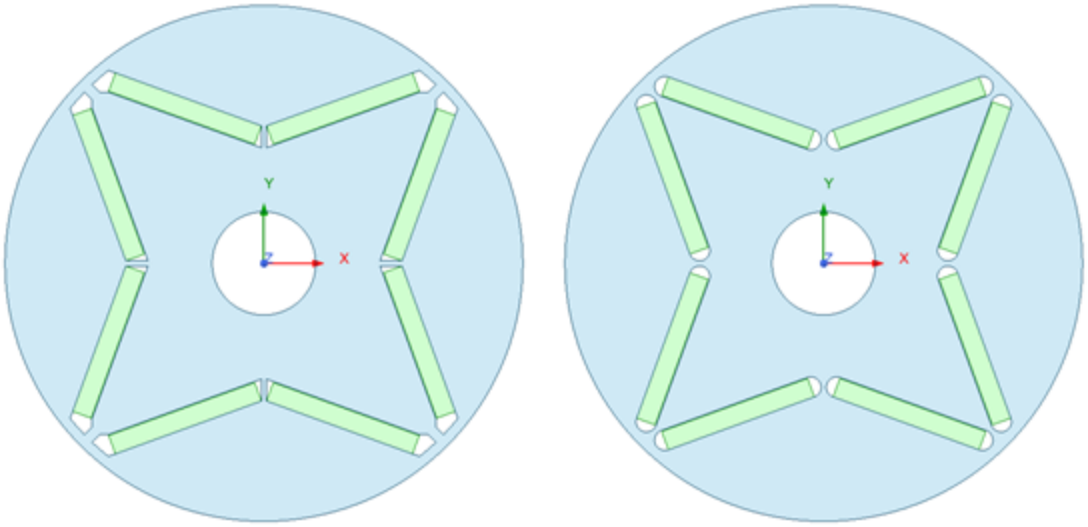
2-Pole V-Type IPM Cores

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating Magnets (VPM Core)

All magnets of a V-type IPM Core are created by setting the value of **InfoCore** to 1. V-type IPM cores of web type and simple type with all magnets are shown below.

Notice that **GapMag** includes total gaps of both top and bottom sides. If both top and bottom gaps are 0.1 mm, the created magnets will be located inside the PM ducts in such a way that the top gap is 0, and the bottom gap is 0.2 mm. This will not affect the performance solutions. However, this enhances the mesh quality greatly.



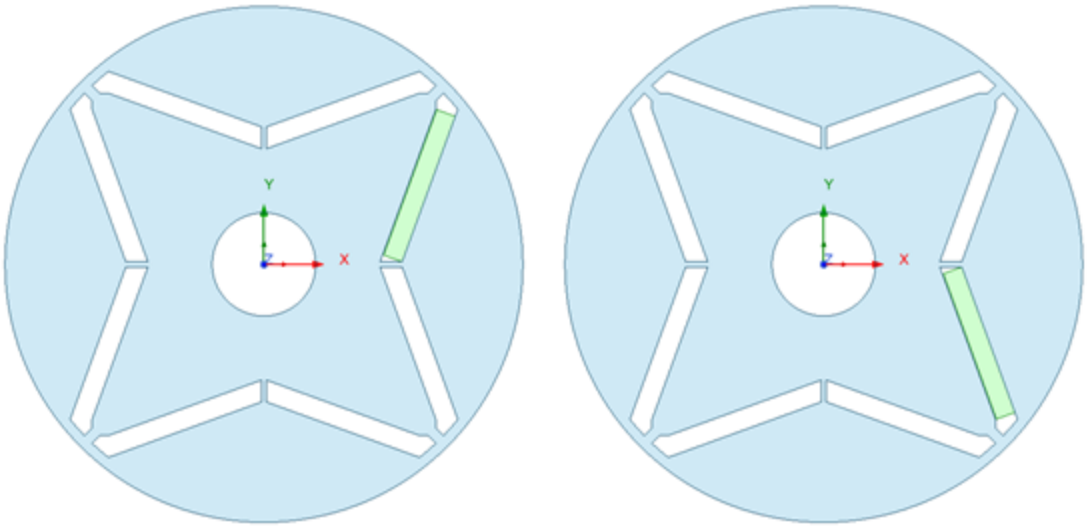
Web type (**PoleType** = 0)

Simple type (**PoleType** = 1)

V-Type IPM Cores with All Magnets

To easily assign the direction of magnet excitation, magnets can be created one-by-one by setting the value of **InfoCore** to 3, or 4. The setting of **InfoCore** = 3 is used to create the left-side magnet and **InfoCore** = 4 is used to create the right-side magnet as shown below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



InfoCore = 3 InfoCore = 4

V-Type IPM Core with a Single Magnet

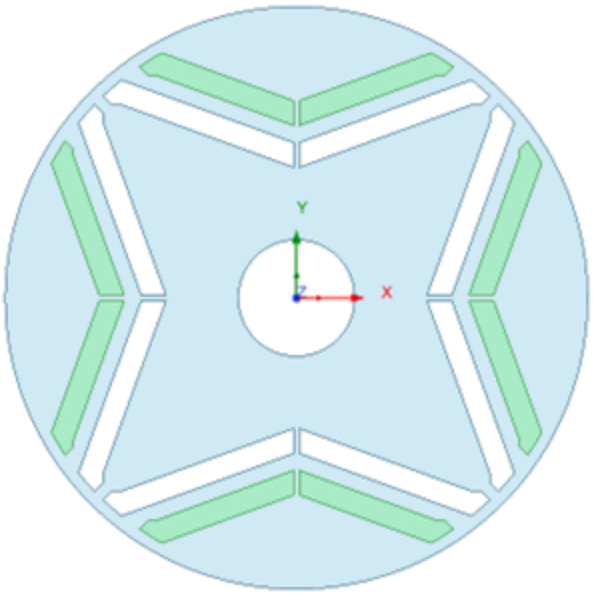
When the input values of magnet width are too large to fit in the PM ducts, they will be modified based on the maximum possible sizes.

Creating PM Ducts (VPM Core)

All PM ducts of a V-type IPM Core are created by setting the value of **InfoCore** to 2. Subtracting a PM duct set, you can create a poly-layer V-type IPM core

A V-type PM duct set, together with a V-type IPM core, is shown below. The parameters of the additional ducts can be totally different from those of the original core.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you subtract the additional PM duct set from the core, you can get a core with two-layer PM ducts. Continuing to create and subtract additional PM duct sets, the software can create an IPM core with an arbitrary number of layers of PM ducts.

VentCore UDP

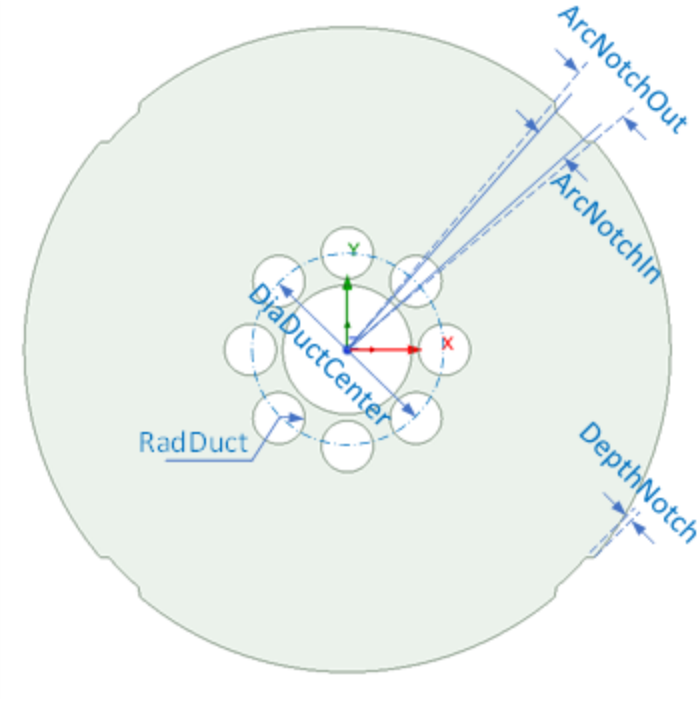
The VentCore UDP is used to create a vent core, a vent shaft, vent ducts, as well as notches, for various cores of electrical machines. A vent core can be used as a base object from which some tool objects can be subtracted to form various machine cores. The vent ducts and notches can also be used as tools to be subtracted from other machine cores.

Property	Description
DiaGap	Core diameter on gap side, or outer diameter
DiaYoke	Core diameter on yoke side, or inner diameter
Length	Core length
Poles	Number of poles for notches
DepthNotch	Notch depth
ThkBridge	Bridge thickness
ArcNotchOut	Notch outer arc, in electric degrees
ArcNotchIn	Notch inner arc, in electric degrees
Ducts	Number of axial vent ducts
DuctType	0: none; 1: circular; 2: shaft spoke; 3: arc; 4: rectangular

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

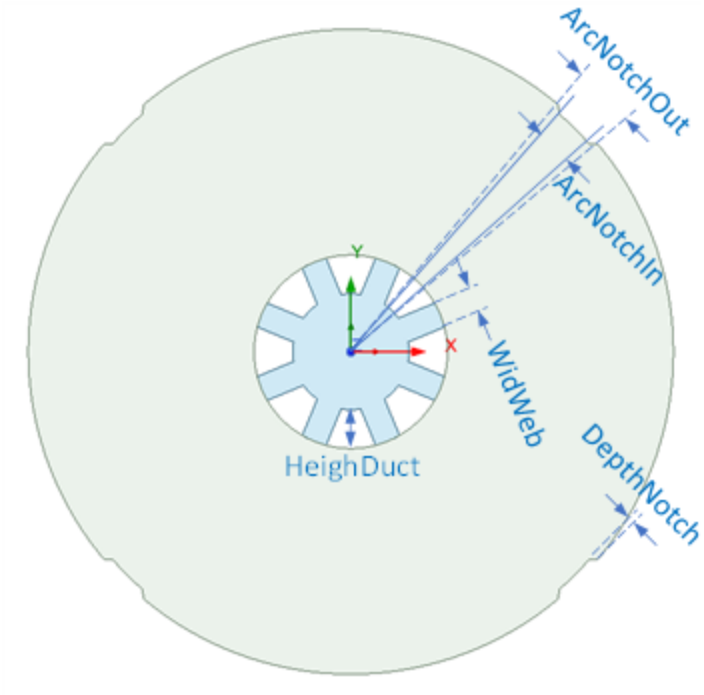
Property	Description
DiaDuctCenter	Diameter of duct center (not for shaft spoke)
HeightDuct	Duct height in radial direction
WidDuct/WidWeb	Duct width (rectangular) or web width (shaft spoke, arc).
RadDuct	Duct radius (circular) or corner radius (arc, rectangular)
AngDuctShift	Duct shift angle
LenRegion	Region length
InfoCore	0: core; 1: shaft; 2: ducts; 3: notches; 100: region

Most parameters for a VentCore UDP are shown in the figures below except common parameters like **DiaGap**, **DiaYoke**, and **Length**.



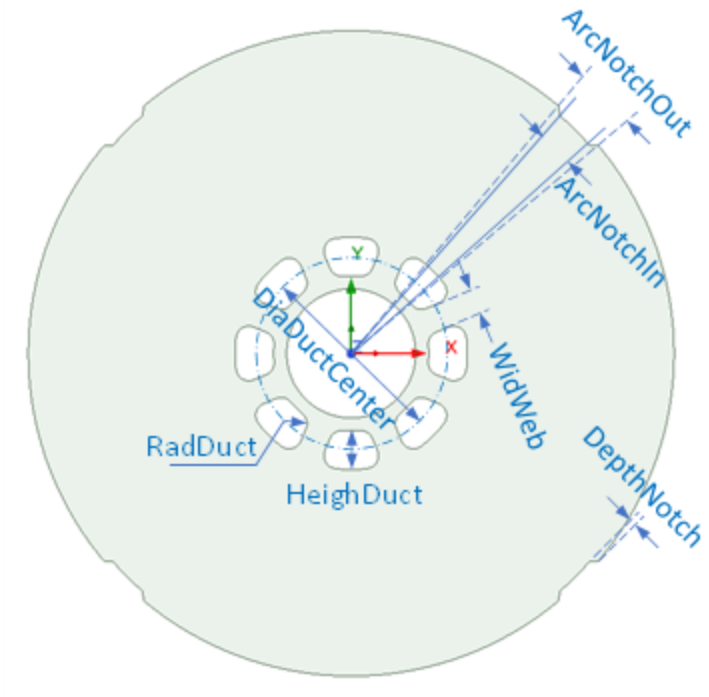
Parameters of VentCore UDP (DuctType = 1)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



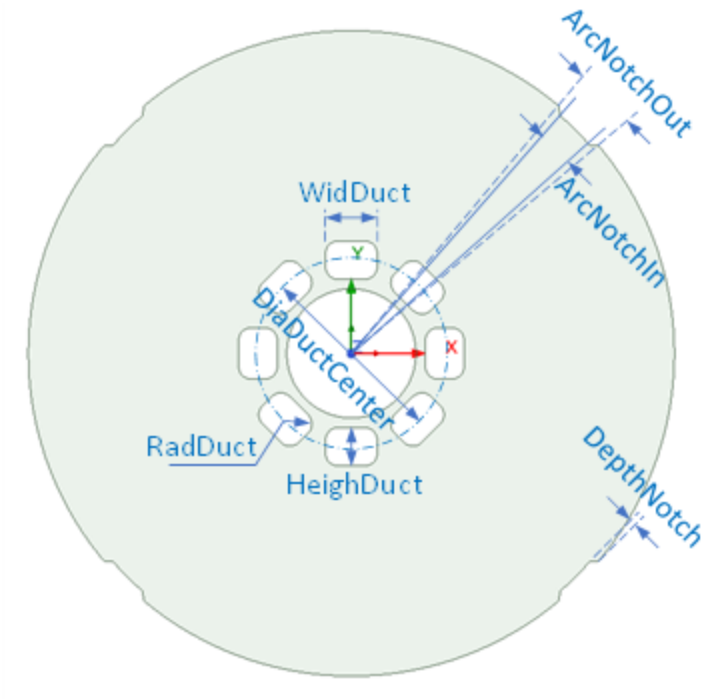
Parameters of VentCore UDP (DuctType = 2)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Parameters of VentCore UDP (DuctType = 3)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Parameters of VentCore UDP (DuctType = 4)

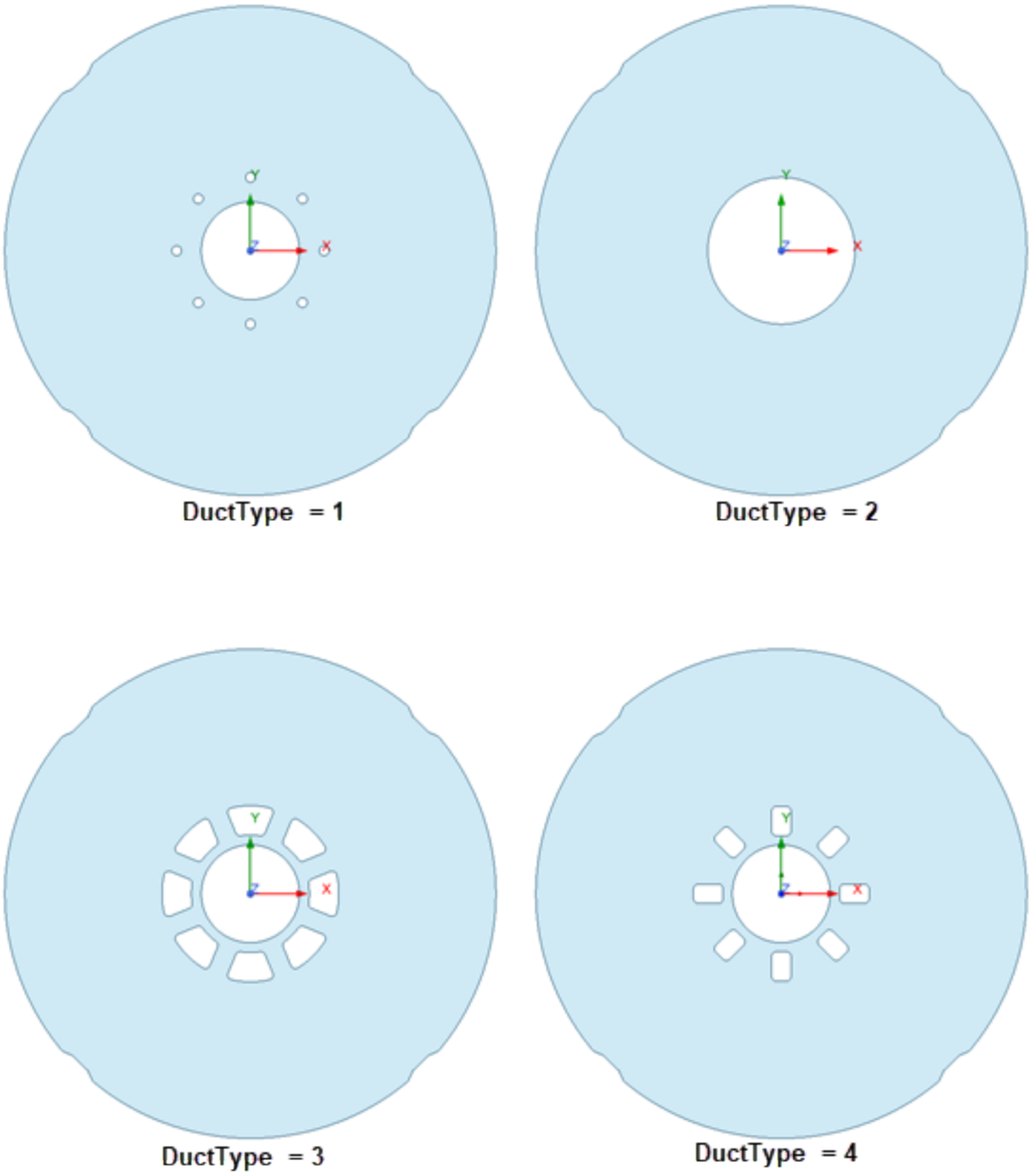
Creating a VentCore

You can create a vent core from VentCore UDP by setting **InfoCore** = 0.

With the default parameter settings of the VentCore UDP, the created vent cores of different duct types are shown in the figures below. Note that the value of **DiaYoke** for **DuctType** = 2 is modified to 30 mm to pass the validation.

VentCores of Various Duct Types

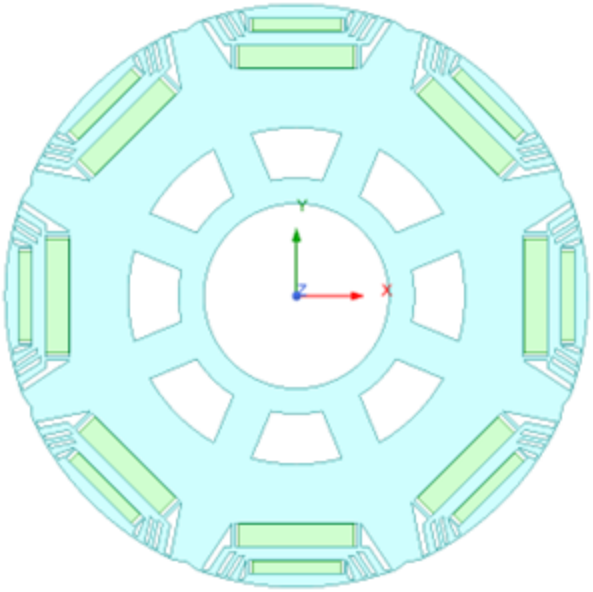
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following figure shows an example in which a vent core is used as a base object, and several U-type IPM ducts are used as tool objects to be subtracted from the vent core to form a poly-layer IPM core with vent ducts.

Poly-Layer IPM Core with Vent Ducts

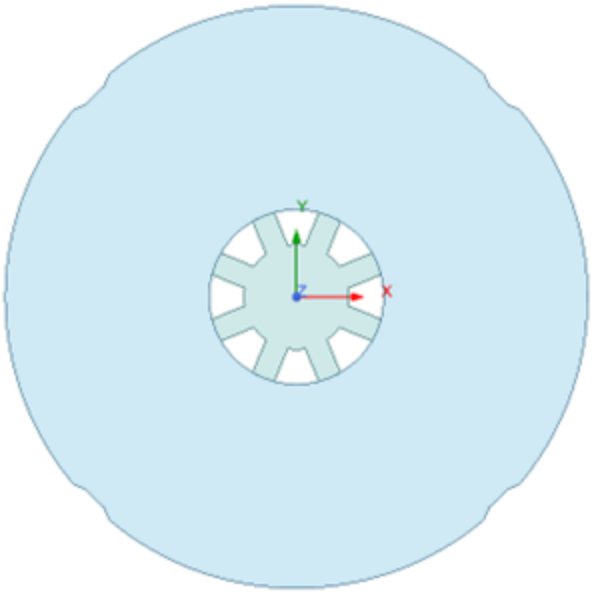
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Creating a Shaft (VentCore)

You can create a vent core shaft using the VentCore UDP by setting the value of **InfoCore** to 1.

A shaft, together with a V-type IPM core with **DuctType** = 2, is shown below.

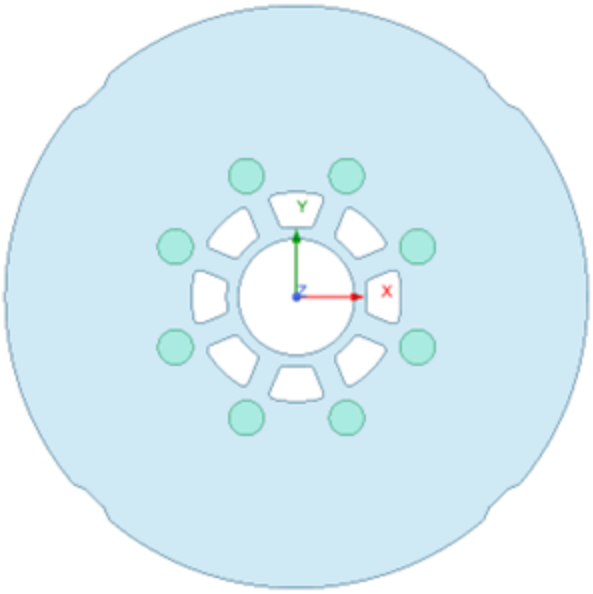


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating Vent Ducts (VentCore)

All vent ducts of a vent core are created by setting the value of **InfoCore** to 2. Subtracting a vent duct set, you can create a vent core with poly-layer vent ducts.

A vent duct set, together with a vent core, is shown in the figure below. The parameters of the additional ducts can be totally different from those of the original core.



Vent Duct Set of DuctType = 1 and a Vent Core of DuctType = 3

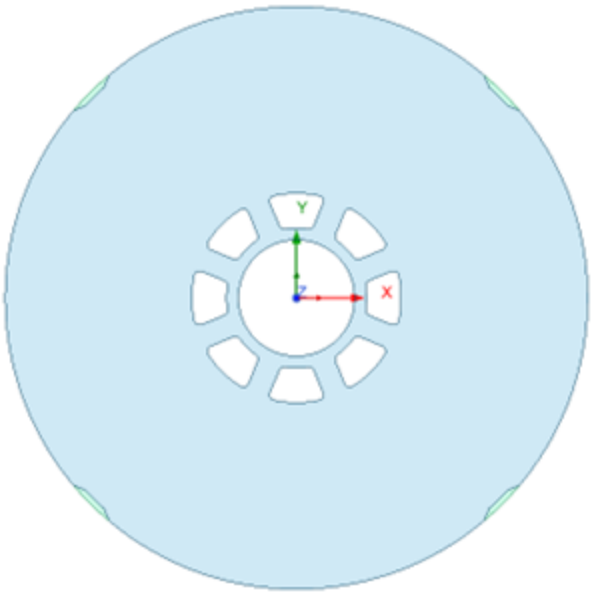
If you subtract the additional vent duct set from a vent core, you can get a core with two-layer vent ducts. Continuing to create and subtract additional vent duct sets, you can create a vent core with an arbitrary number of layers of vent ducts.

Creating Notches (VentCore)

All notches of a vent core are created by setting the value of **InfoCore** to 3, which can be subtracted from an IPM core to improve the air-gap flux density waveforms.

All created notches, together with a vent core of **DuctType** = 3, are shown in the figure below. The parameters of the additional Notches can be totally different from those of the original core.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Notches Together with a Vent Core of DuctType = 3

UDPs for Other Cores and Coils

UDPs defined for other cores and coils include

- DiskSlotCore UDP
- DiskCoil UDP
- DiskPMCore UDP
- LinearMCore UDP
- TransCore UDP
- TransCoil UDP
- TransTwist UDP

DiskSlotCore UDP

The DiskSlotCore UDP is used to create a stator/rotor core for axial-flux machines. It can also create a squirrel-cage winding of the axial-flux type.

Property	Description
DiaOuter	Core outer diameter
DiaInner	Core inner diameter
Thickness	Core axial thickness per side

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Property	Description
Gap	Gap between core & xy plane (>0: upper; <0: lower; =0: double-sided)
Skew	Skew angle
Slots	Number of slots
SlotType	Slot type: 1 to 6
Hs0	Slot opening height
Hs01	Slot closed bridge height
Hs1	Slot wedge height
Hs2	Slot body height
Bs0	Slot opening width
Bs1	Slot wedge maximum width
Bs2	Slot body bottom width, 0 for parallel teeth
Rs	Slot body bottom fillet
FilletType	0: a quarter circle; 1: tangent connection
HalfSlot	0: symmetric slots; 1: half slots
RingLength	One-side radial ring length
RingHeight	Axial ring height
SegAngle	Deviation angle for slot arches (10~30, <10 for true surface)
InfoCore	0: core only; 1: squirrel cage only

These parameters are used in the following figures.

Note:

- For parameters such as **Skew**, **FilletType**, **SegAngle**, **HalfSlot** and parameters related to slots, refer to SlotCore UDP parameters.
- Figures are shown in pairs to compare parameters when the **InfoCore** value varies

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

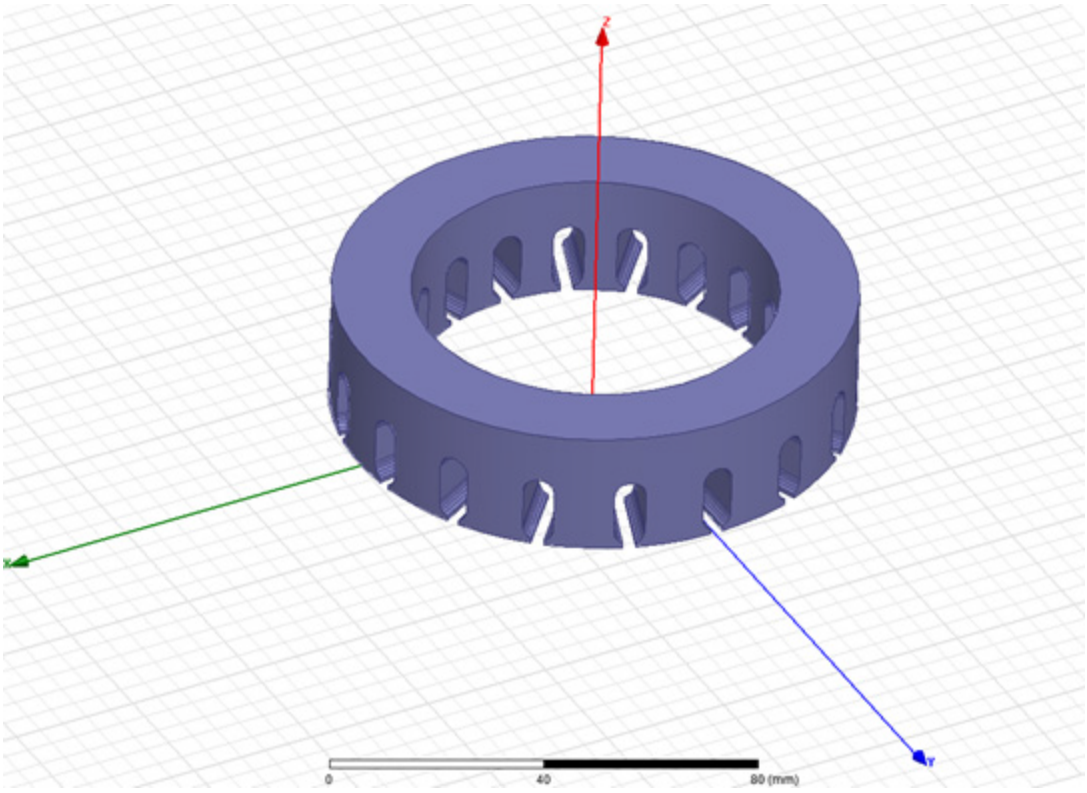


Figure 9-77 Overview of DiskSlotCore (InfoCore set to 0)

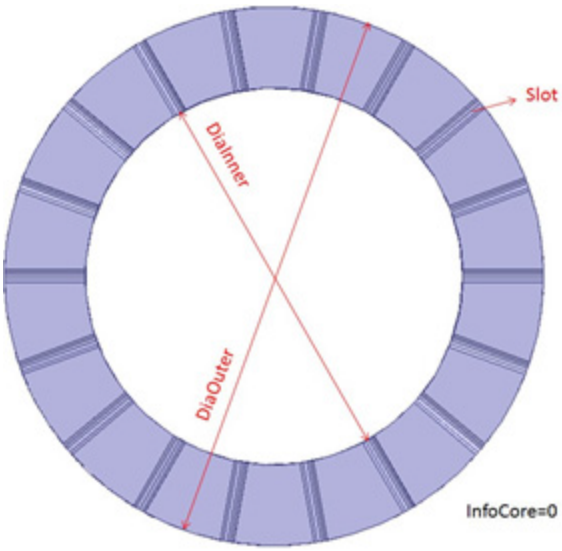


Figure 9-78 DiskSlot Core (InfoCore set to 0)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

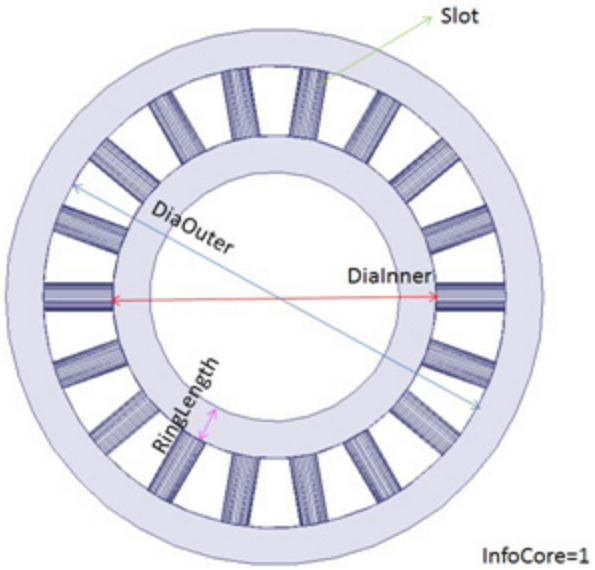


Figure 9-79 DiskSlot Core (InfoCore set to 1)

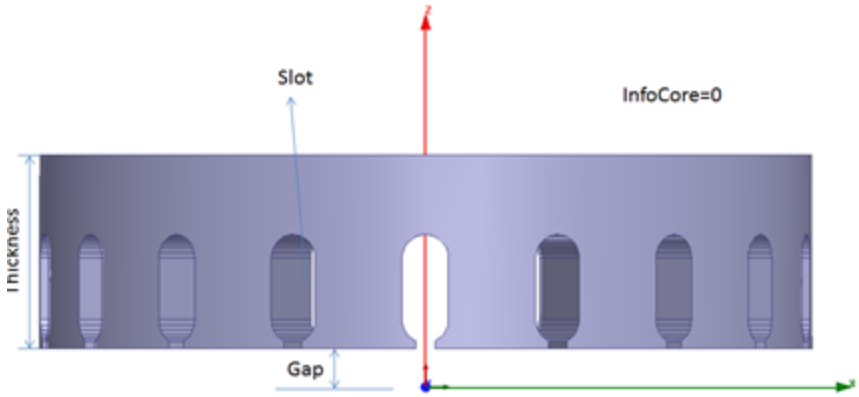


Figure 9-80 3D DiskSlot Core (InfoCore set to 0)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

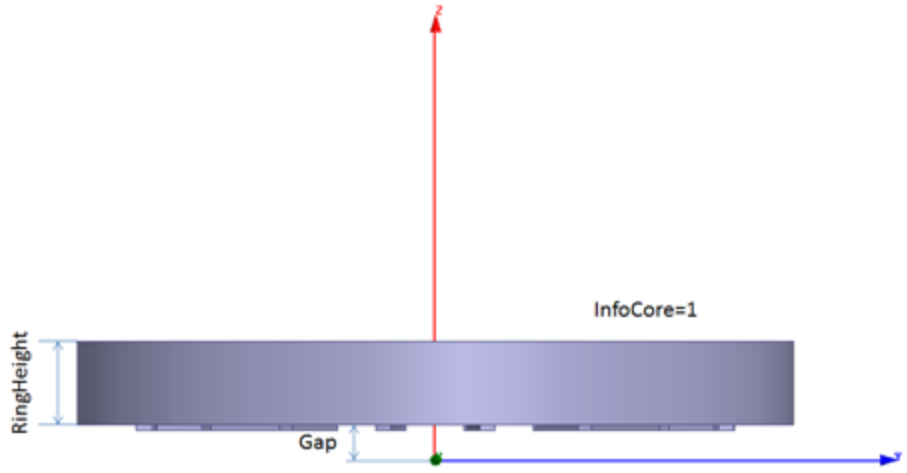


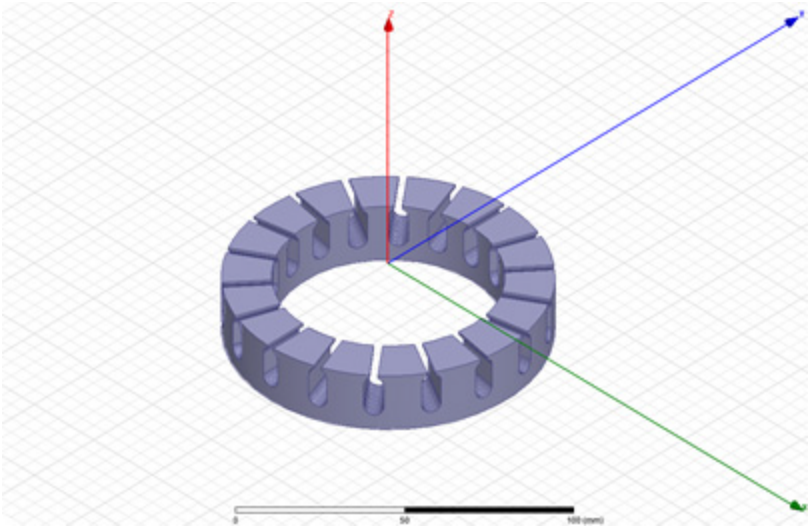
Figure 9-81 3D DiskSlot Core (InfoCore set to 1)

Creating an Upper or Lower Axial-FluxSlot Core (DiskSlot Core)

You can create an axial-flux slot core of a DiskSlot Core manually by setting the value of **InfoCore** to 0.

For an upper core, set the value of **Gap** to be larger than 0, as shown in " Overview of DiskSlotCore (InfoCore set to 0)" on page 9-201 and " 3D DiskSlot Core (InfoCore set to 0)" on the previous page; for a lower core, set this value to be smaller than 0, which is shown in " Lower axial-flux slot core" on the next page.

You can assign the value of this property either when creating a DiskSlot core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 9-82 Lower axial-flux slot core

Creating a Squirrel-Cage Winding

You can create squirrel-cage winding of a DiskSlot core manually by setting the property of **InfoCore** to 1. An example of squirrel-cage winding is shown in " DiskSlot Core (InfoCore set to 1)" on page 9-202 .

You can assign the value of this property either when creating an axial-flux slot core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

DiskCoil UDP

The DiskCoil UDP is used to create a lap-type coil in an axial-flux slotted core created by the DiskSlotCore UDP for axial-flux machines. It can also create a coil terminal for current assignment.

Property	Description
DiaOuter	Core outer diameter
DiaInner	Core inner diameter
Thickness	Core axial thickness per side
Gap	Gap between core & xy plane (>0: upper; <0: lower; =0: double-sided)
Skew	Skew angle
Slots	Number of slots
SlotType	Slot type: 1 to 7
Hs0	Slot opening height
Hs1	Slot wedge height
Hs2	Slot body height
Bs0	Slot opening width
Bs1	Slot wedge maximum width
Bs2	Slot body bottom width, 0 for parallel teeth
Rs	Slot body bottom fillet
FilletType	0: a quarter circle; 1: tangent connection
Layers	Number of winding layers
CoilPitch	Coil pitch measured in slots
EndExt	One-side radial end extended length
EndClr	Clearance between two end spans; 0 for no span
SegAngle	Deviation angle for end span (5~15, <5 for true surface)
InfoCoil	0: winding; 1: coil; 2: terminal

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- For parameters such as **Skew**, **FilletType**, **SegAngle**, **HalfSlot** and parameters related to slots, refer to SlotCore UDP parameters.
- " One disk coil (InfoCore set to 1)" on the next page can be used in comparison to " All disk coils (InfoCore set to 0)" below to show **CoilPitch** parameter.
- A DiskSlot core is used as reference in " EndExt and EndClr parameters (InfoCore set to 0)" on the next pageto show **EndExt** and **EndClr** parameters.

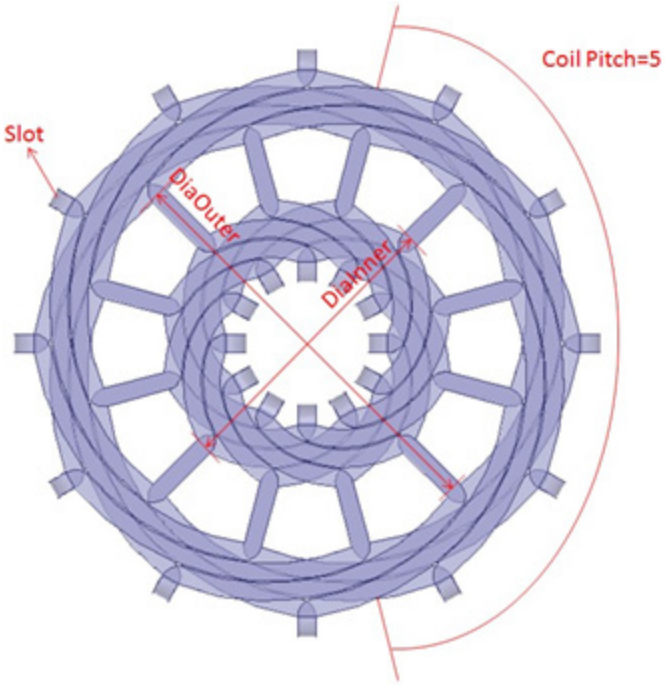


Figure 9-83 All disk coils (InfoCore set to 0)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

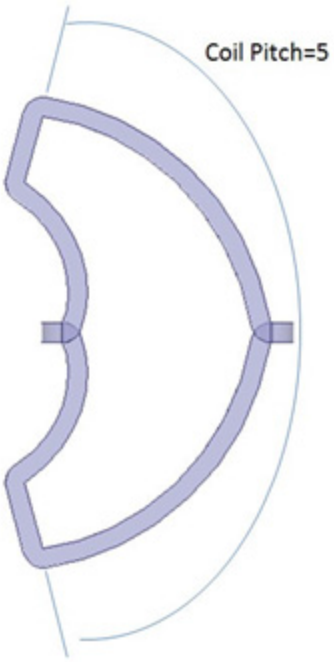


Figure 9-84 One disk coil (InfoCore set to 1)

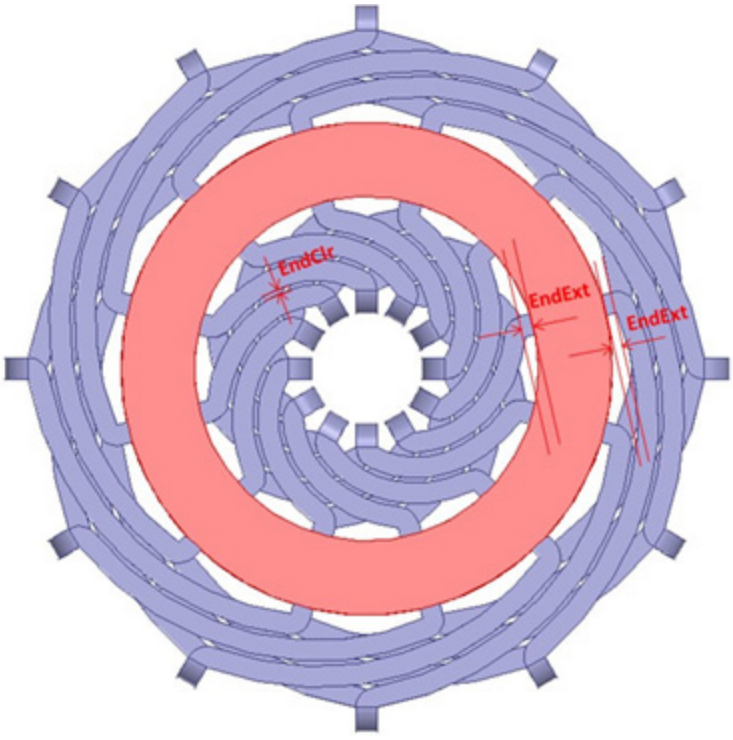


Figure 9-85 EndExt and EndClr parameters (InfoCore set to 0)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

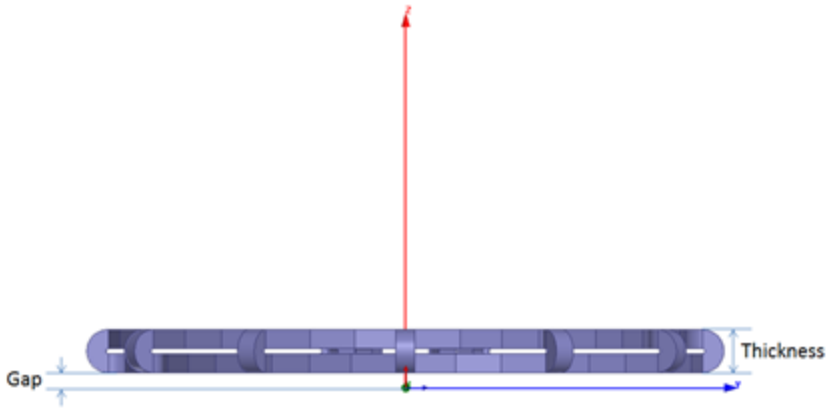


Figure 9-86 Gap and Thickness parameters (InfoCore set to 0)

Creating a Whole Winding (Upper or Lower)

You can create a whole winding of a DiskCoil Core manually by setting the property of **InfoCoil** to 0.

For an upper core, set the value of **Gap** to be larger than 0, which is shown in " Gap and Thickness parameters (InfoCore set to 0)" above; for a lower core, set this value to be smaller than 0, which is shown in " Lower Whole Winding" below.You can assign the value of this property either when creating the DiskCoil or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

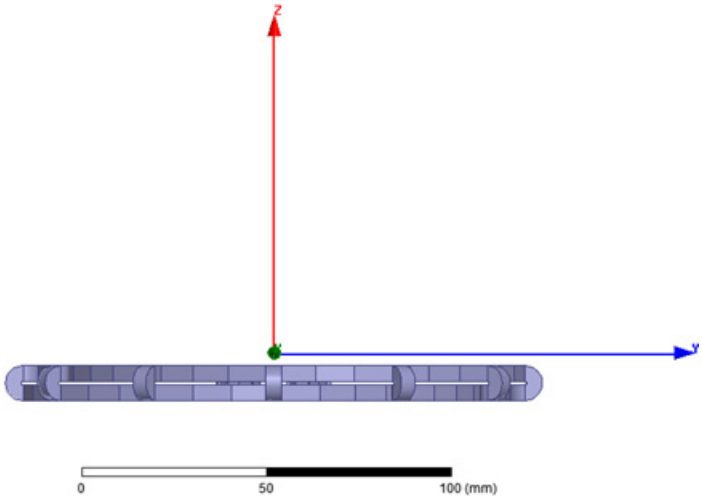


Figure 9-87 Lower Whole Winding

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating a Lap Coil and its Terminals

You can create a lap coil of a DiskCoil manually by setting the value of **InfoCoil** to 1. An example is shown in " One disk coil (InfoCore set to 1)" on page 9-206 .

You can create a terminal of a DiskCoil manually by setting the value of **InfoCoil** to 2. An example is shown in " Terminal of a DiskCoil UDP" below.

You can assign the value of this property either when creating a DiskCoil or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

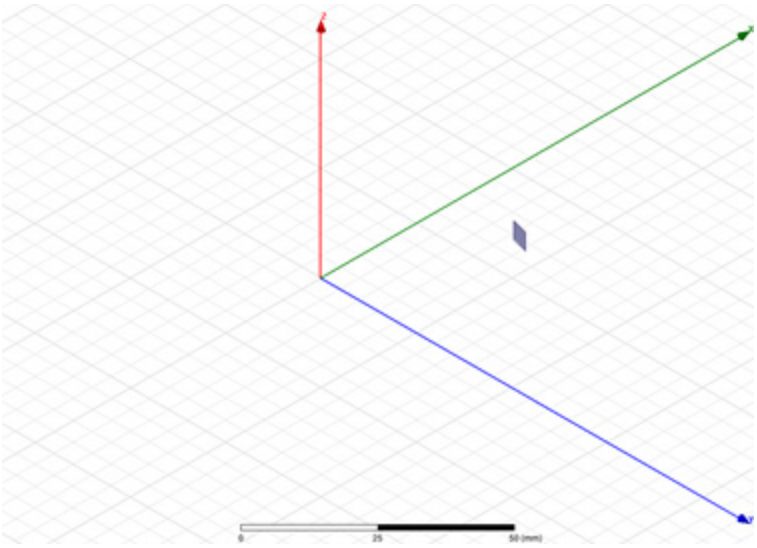


Figure 9-88 Terminal of a DiskCoil UDP

DiskPMCore UDP

The DiskPMCore UDP is used to create a PM core of the axial-flux type for axial-flux PM machines.

Property	Description
DiaOuter	Core outer diameter
DiaInner	Core inner diameter
Thickness	Core axial thickness per side
Gap	Gap between core & xy plane (>0: upper; <0: lower; =0: double-sided)
Skew	Skew angle
Poles	Number of poles

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Property	Description
Embrace	Pole embrace
ThickMag	Magnet axial thickness per side
InfoCore	0: core only; 1: all PMs; 2: one PM only

Note: There are two objects in each of the following figures. Both objects in each figure are DiskPMCore. The value of **InfoCore** of one DiskPMCore is 1 (red object), the other is 0 (gray object).

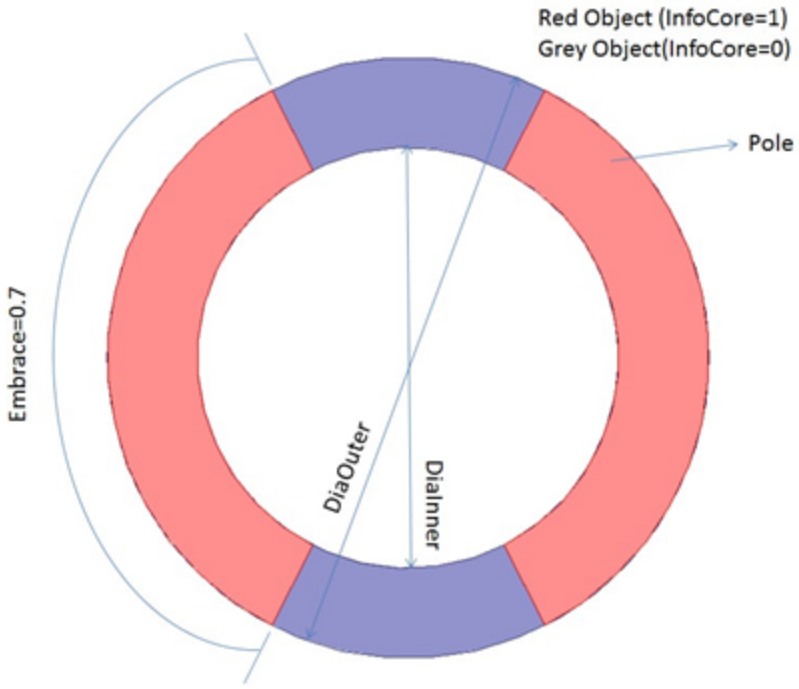


Figure 9-89 DiskPM Core Parameters on X-Y Plane

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

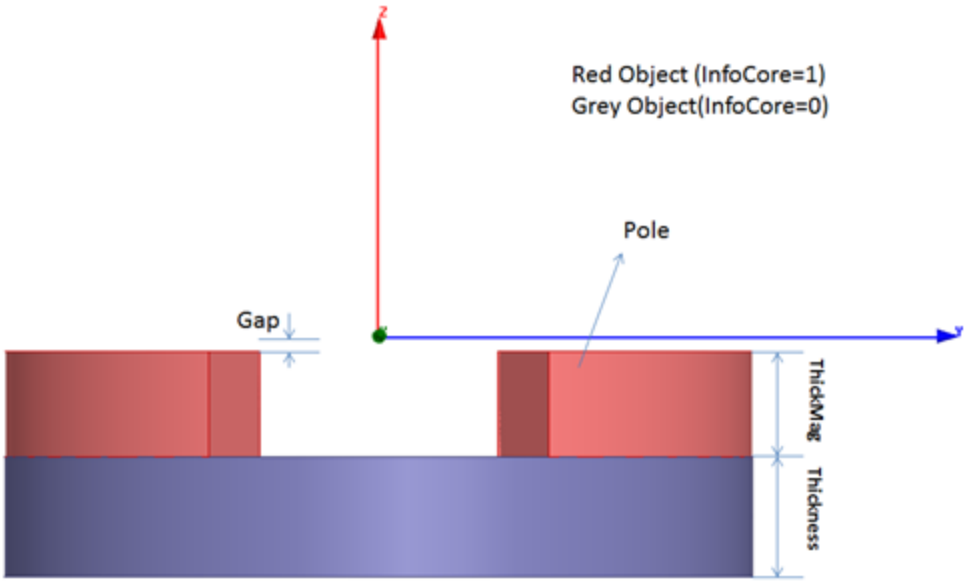


Figure 9-90 DiskPM Core Parameters in Axial Direction

Creating an Upper or Lower Axial-Flux PM Core

You can create an axial-flux PM core of a DiskPM core manually by setting the value of **InfoCore** to 0.

For an upper core, set the value of **Gap** to be larger than 0; for a lower core, set this value to be smaller than 0.

An example of an upper core is shown in "Upper Axial-Flux PM Core" on the facing page, whereas a lower core is in " Lower Axial-Flux PM Core" on the facing page.

You can assign the value of this property either when creating a Disk PM core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

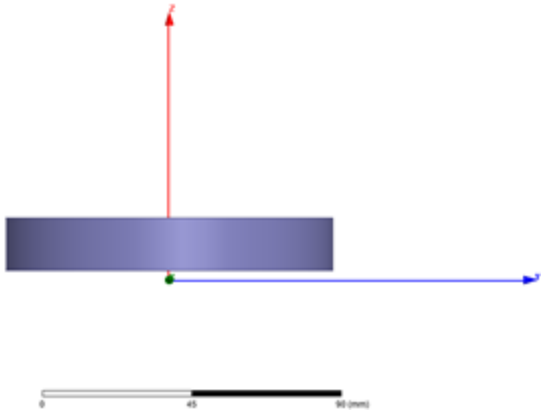


Figure 9-91 Upper Axial-Flux PM Core

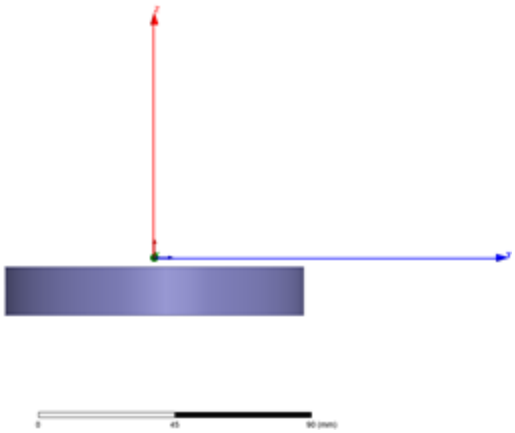


Figure 9-92 Lower Axial-Flux PM Core

Creating All Magnets (Disk PM Core)

You can create all magnets of a disk PM core manually by setting the value of **InfoCoil** to 1. An example of all magnets is shown in " DiskPM Core Parameters on X-Y Plane" on page 9-209 and " DiskPM Core Parameters in Axial Direction" on the previous page as a red object.

You can assign the value of this property either when creating a disk PM core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Creating One Magnet Only (Disk PM Core)

You can create a single magnet of a disk PM core manually by setting the property of **InfoCoil** to 2. An example of one magnet is shown in " One Magnet of Disk PM Core" on the next page.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can assign the value of this property either when creating a disk PM core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

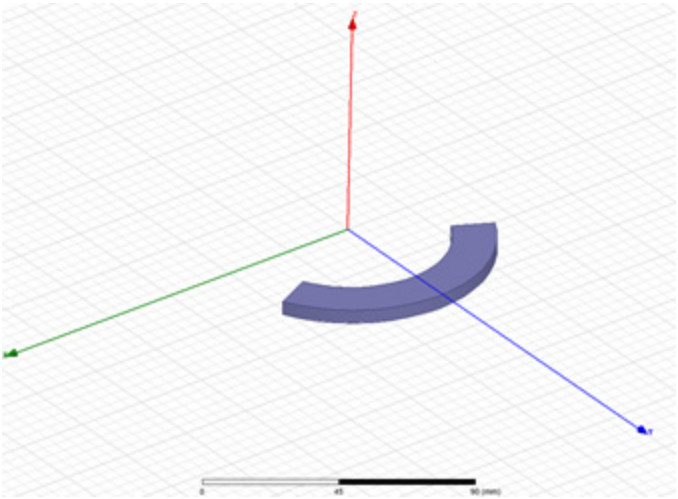


Figure 9-93 One Magnet of Disk PM Core

LinearMCore UDP

The LinearMCore UDP is used to create the mover core and the armature winding for linear motors. It can also create a coil terminal for current assignment.

Property	Description
WidthCore	Core width in motion direction
ThickCore	Core thickness
Length	Core length
SlotPitch	The distance between two slots
Slots	Number of slots
SlotType	Slot type: 1 to 7
Hs0	Slot opening height
Hs1	Slot wedge height
Hs2	Slot body height
Bs0	Slot opening width
Bs1	Slot wedge maximum width
Bs2	Slot body bottom width, 0 for parallel teeth
Rs	Slot body bottom fillet
Layers	Number of winding layers

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Property	Description
CoilPitch	Coil pitch measured in slots
EndExt	One-side end extended length
SpanExt	Axial length of end span; 0 for no span
SegAngle	Deviation angle for slot arches (10~30, <10 for true surface)
InfoCore	0: core only; 1: core & all coils; 2: one coil only

Note:

- For parameters such as **SegAngle** and parameters related to slots, refer to SlotCore UDP parameters.
- The InfoCore values of the following two figures are both 1 to better describe other parameters such as **EndExt** and **SpanExt**

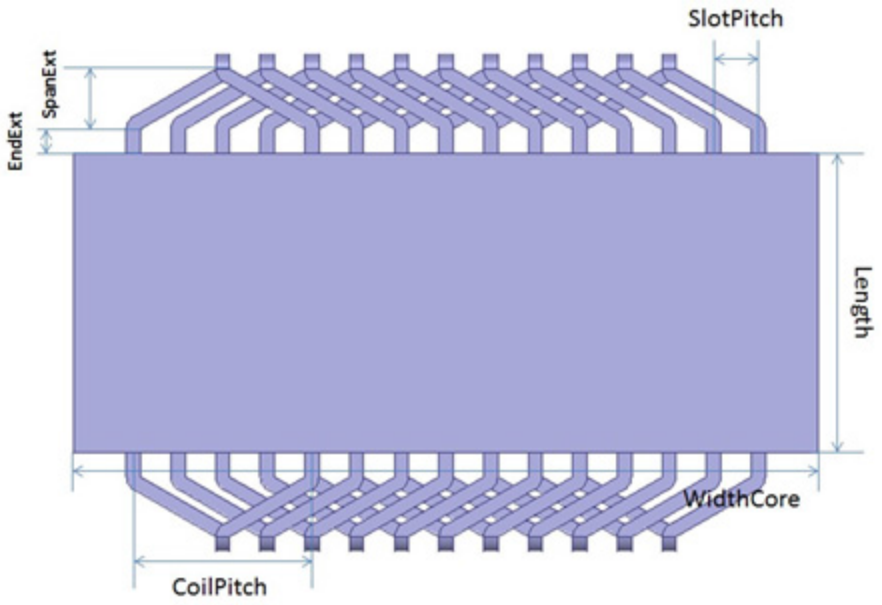


Figure 9-94 Parameters of LinearMCore in X-Y Plane (InfoCore set to 1)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

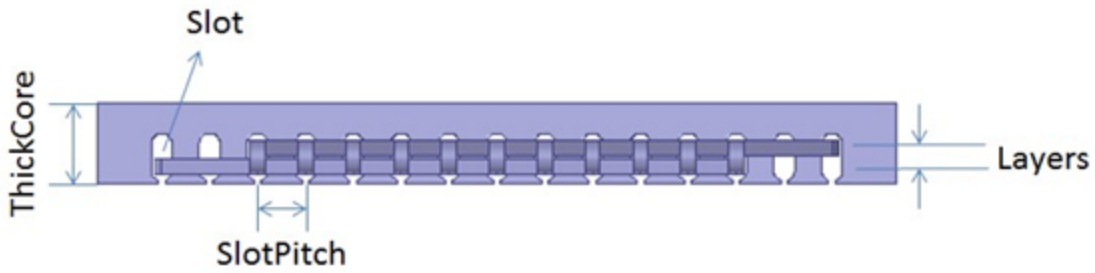


Figure 9-95 Parameters of LinearMCore in Axial Direction (InfoCore set to 1)

Creating a Linear Machine Core

You can create a linear machine core manually by setting the value of **InfoCore** to 0.

An example of a linear machine core is shown in " Linear Machine Core (InfoCore set to 0)" below.

You can assign the value of this property either when creating a linear-machine core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

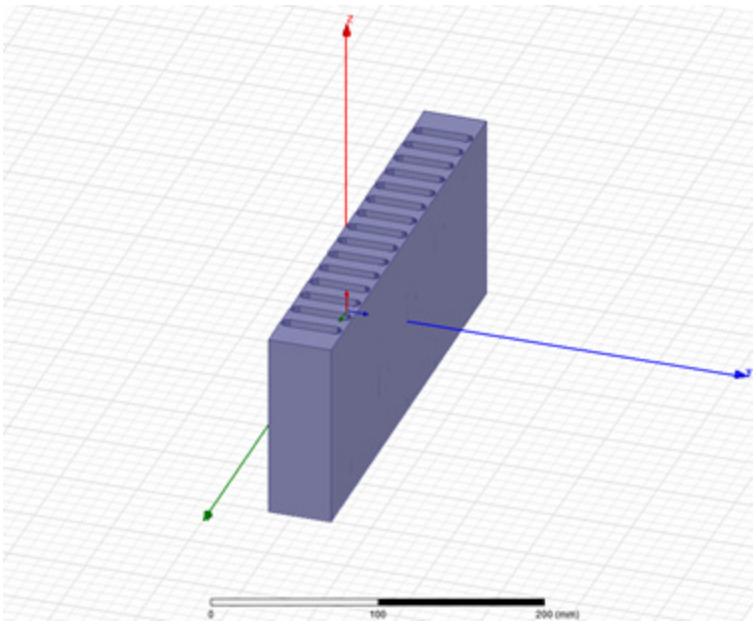


Figure 9-96 Linear Machine Core (InfoCore set to 0)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating a Core and All Coils (Linear Machine Core)

You can create a core and all coils manually by setting the value of **InfoCore** to 1. An example of a core and all coils of a linear-machine core is shown in "Core and all Coils of a Linear Machine Core (InfoCore set to 1)" below.

You can assign the value of this property either when creating a linear machine core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

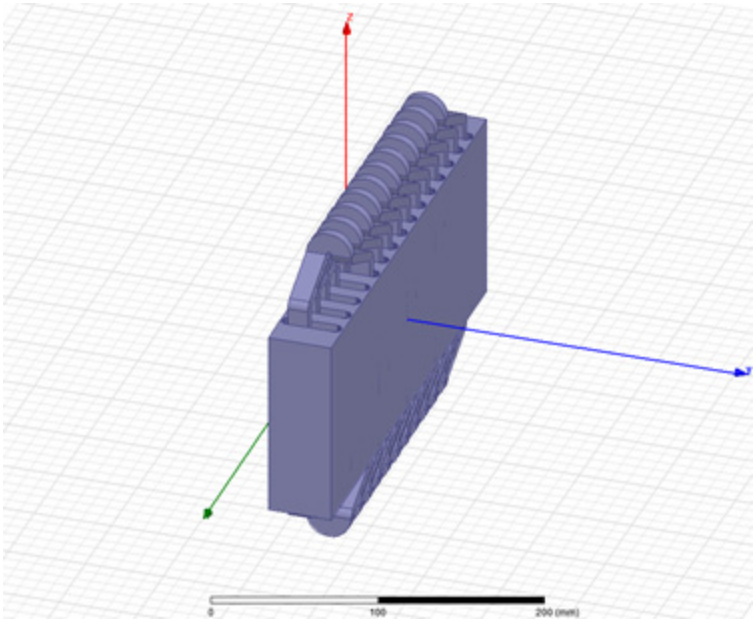


Figure 9-97 Core and all Coils of a Linear Machine Core (InfoCore set to 1)

Creating One Coil Only (Linear Machine Core)

You can create a one coil only manually by setting the value of **InfoCore** to 2.

An example of a coil of a linear machine core is shown in " One Coil of a Linear Machine Core (InfoCore set to 2)" on the next page.

You can assign the value of this property either when creating a linear machine core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

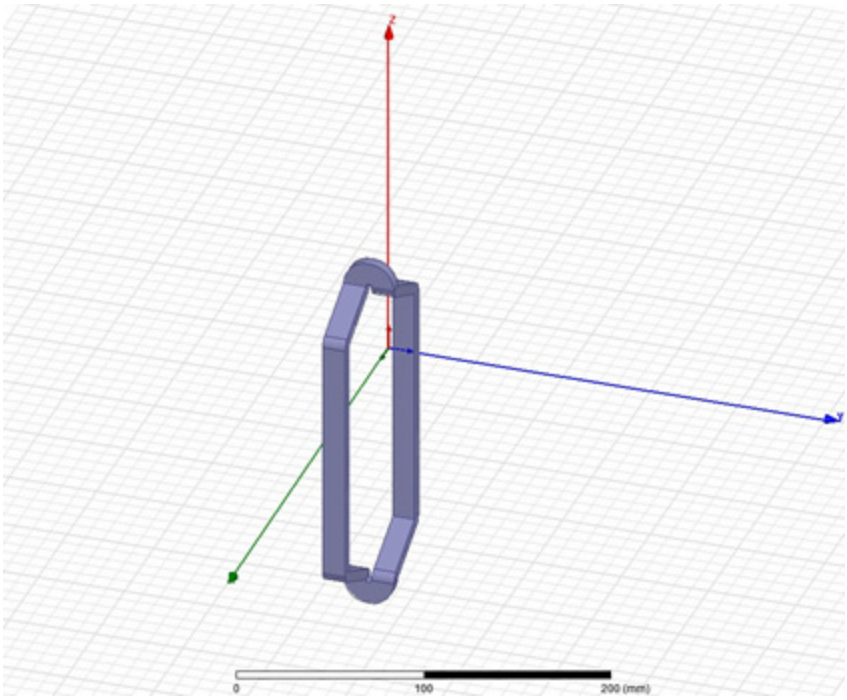


Figure 9-98 One Coil of a Linear Machine Core (InfoCore set to 2)

TransCore UDP

The TransCore UDP is used to create a three-leg transformer core for three-phase transformers.

Property	Description
DiaLeg	Outer diameter of leg cross-section
DistLeg	Leg center to center distance
DistYoke	Yoke center to center distance
Stages	Number of stages of leg cross-section
ThickCore	Core thickness, only used for Stages = 1
WidthYoke	Yoke width, =0 for same cross-section as leg's
InfoCore	0: whole core; 1: legs only; 2: yokes only

Note: " TransCore when Stages is not 1 and WidthYoke is 0" on the facing page, " TransCore when Stages is not 1 and WidthYoke is not 0" on the facing page, and " TransCore when Stages is 1 and WidthYoke is not 0" on page 9-218 show a TransCore UDP with different values of **Stages** and **WidthYoke**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

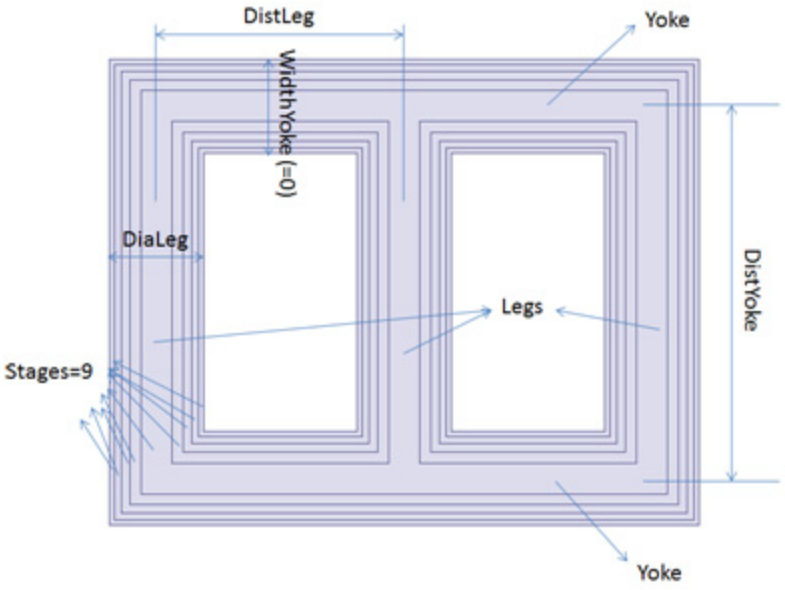


Figure 9-99 TransCore when Stages is not 1 and WidthYoke is 0

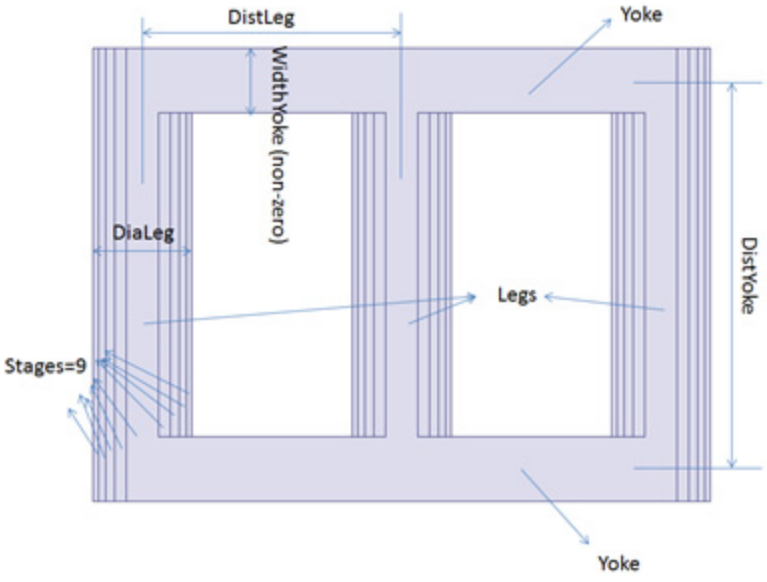


Figure 9-100 TransCore when Stages is not 1 and WidthYoke is not 0

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

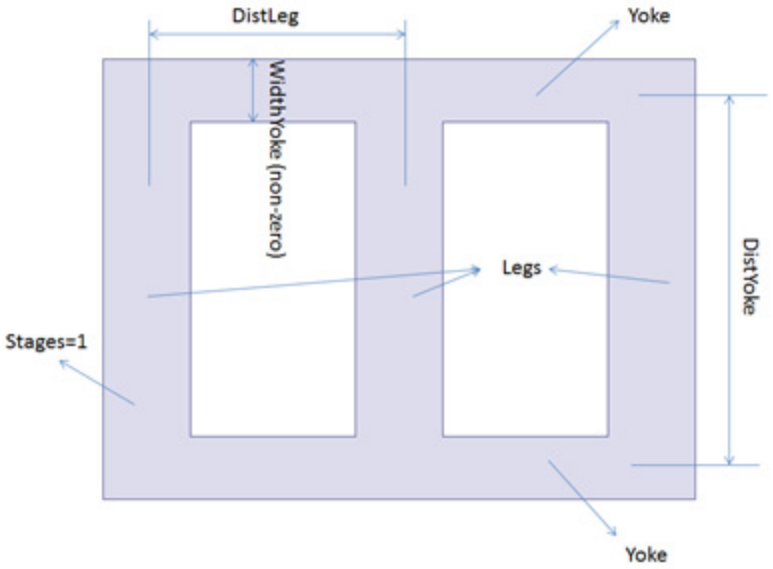


Figure 9-101 TransCore when Stages is 1 and WidthYoke is not 0

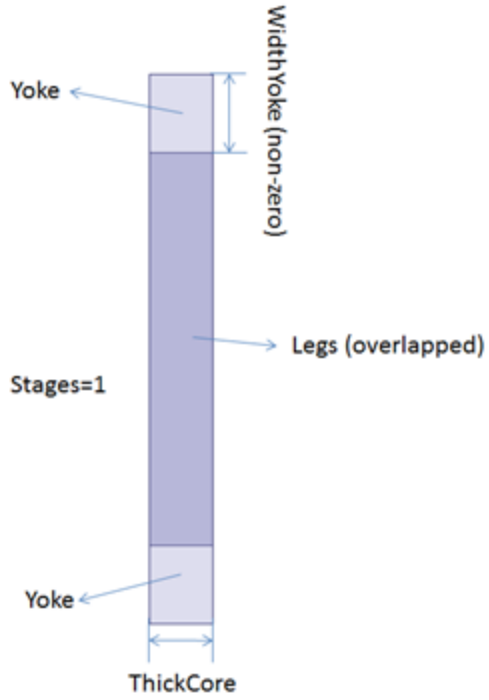


Figure 9-102 Parameters of TransCore in Axial Direction (when Stages is 1 and WidthYoke is not 0)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating a Three-Leg Transformer Core

You can create a three-leg transformer core manually by setting the value of **InfoCore** to 0.

Examples of a three-leg transformer core are shown in " TransCore when Stages is not 1 and WidthYoke is 0" on page 9-217 , " TransCore when Stages is not 1 and WidthYoke is not 0" on page 9-217 , " TransCore when Stages is 1 and WidthYoke is not 0" on the previous page, and " Parameters of TransCore in Axial Direction (when Stages is 1 and WidthYoke is not 0)" on the previous page.

You can assign the value of this property either when creating a three-leg transformer or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Creating Three Legs Only

You can create three legs of the core manually by setting the value of **InfoCore** to 1.

An example of three legs is shown in " Three Legs of TransCore (InfoCore set to 1)" below.

You can assign the value of this property either when creating a three-leg transformer core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

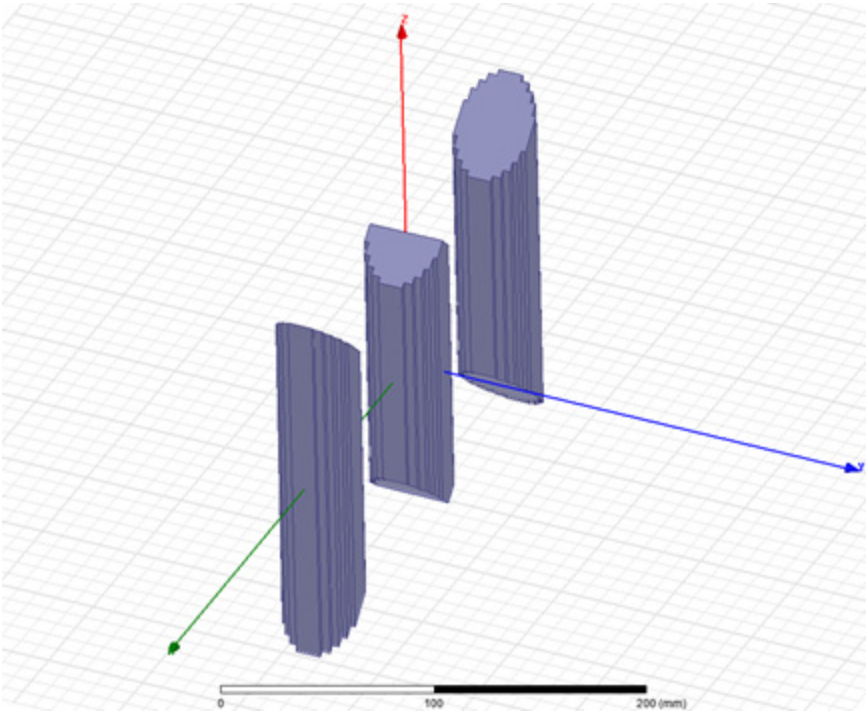


Figure 9-103 Three Legs of TransCore (InfoCore set to 1)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating Two Yokes Only

You can create two yokes of the core manually by setting the value of **InfoCore** to 2.

An example of two yokes is shown in " Two Yokes of TransCore (InfoCore set to 2)" below.

You can assign the value of this property either when creating a three-leg transformer core or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

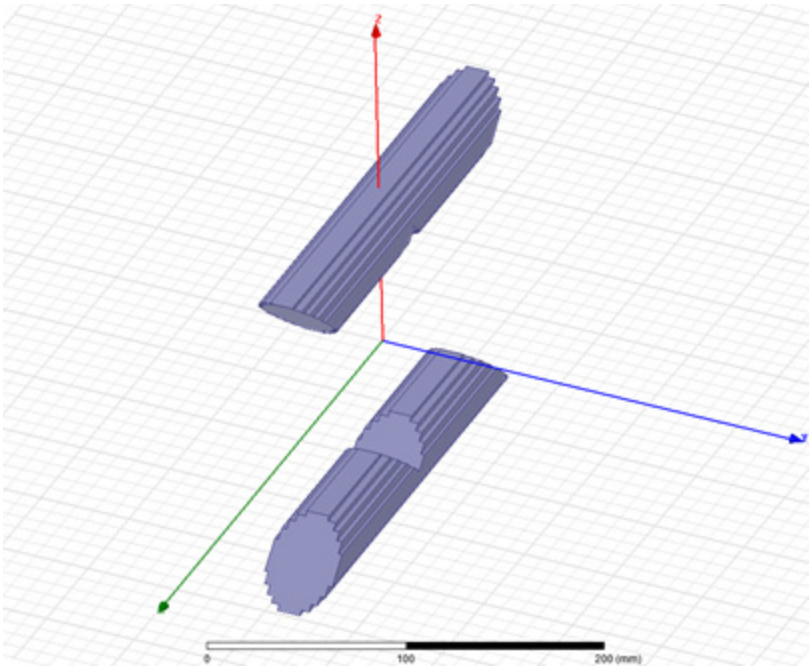


Figure 9-104 Two Yokes of TransCore (InfoCore set to 2)

TransCoil UDP

The TransCoil UDP is used to create a coil in a three-leg transformer core created by the TransCore UDP for three-phase transformers. It can also create a coil terminal for current assignment.

Property	Description
DistLeg	Leg center to center distance
CoilType	Coil type: 1 for solenoid coil; 2 for pancake coil
WidthIn	Coil width between two inner sides
DepthIn	Coil depth between two inner ends

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Property	Description
RadiusIn	Coil inner fillet radius
ThickCoil	Coil thickness of one side
HighCoil	Coil height
Layers	Number of layers
GapLayer	Gap between two layers
InfoCore	0: all coils; 1: one coil only

Note: The key to making the cross-section of each layer of a coil to be a circle is to set the value of **WidthIn** equal to that of **DepthIn** and at the same time set the value of **RadiusIn** to be half of that of **WidIn**. That is, **WidthIn**=**DepthIn**=2***RadiusIn**.

The cross-section of a layer could be a non-circle if the value of **WidthIn** is not equal to that of **DepthIn** or **RadiusIn**'s value is not half of **WidthIn**'s. " RadiusIn of TransCoil" on the next page shows a situation where a layer is a non-circle.

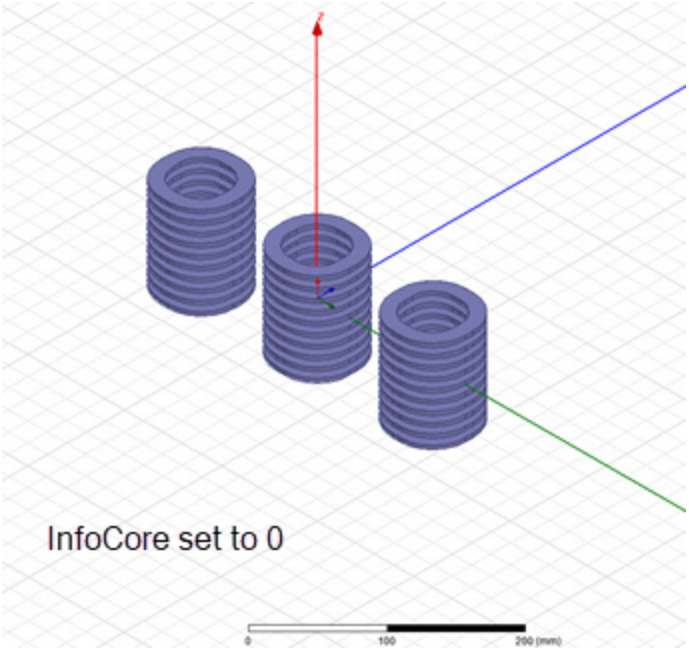


Figure 9-105 Overview of TransCoil

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

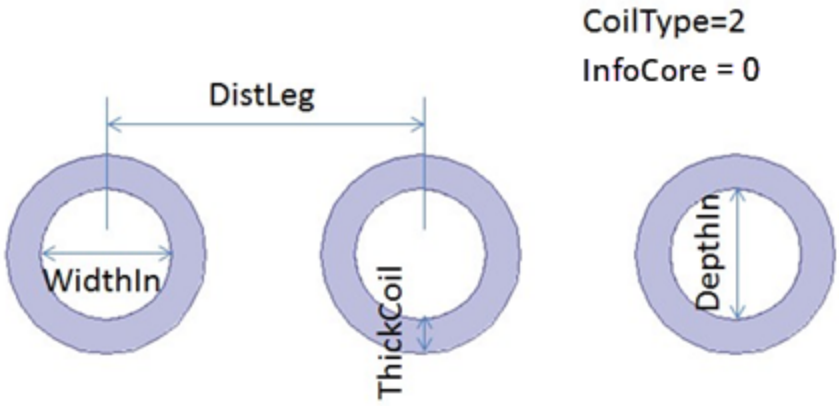


Figure 9-106 Parameters of TransCoil in X-Y Plane

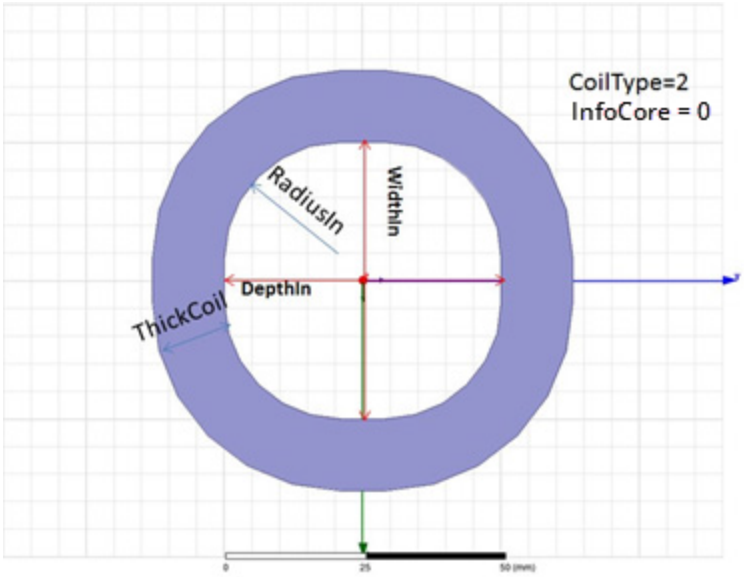


Figure 9-107 RadiusIn of TransCoil

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

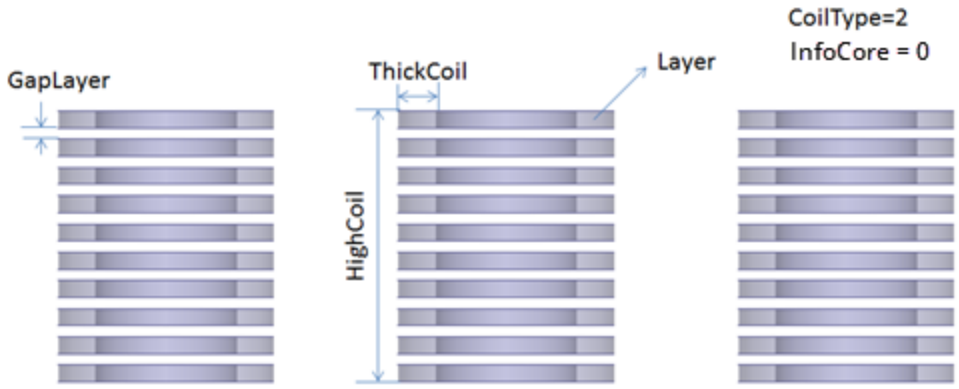


Figure 9-108 Parameters of TransCoil in Axial Direction

Creating Coils for All Three Legs

You can create coils for three legs manually by setting the value of **InfoCore** to 0.

Examples of coils for TransCoil UDP are shown in " Overview of TransCoil" on page 9-221 , " Parameters of TransCoil in X-Y Plane" on the previous page, and " Parameters of TransCoil in Axial Direction" above.

You can assign the value of this property either when creating a TransCoil UDP or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Creating One-Leg Coil Only

You can create a one-leg coil manually by setting the value of **InfoCore** to 1.

An example of a one-leg coil for a TransCoil UDP is shown in " One-Leg Coil for TransCoil UDP" on the next page.

You can assign the value of this property either when creating a TransCoil UDP or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

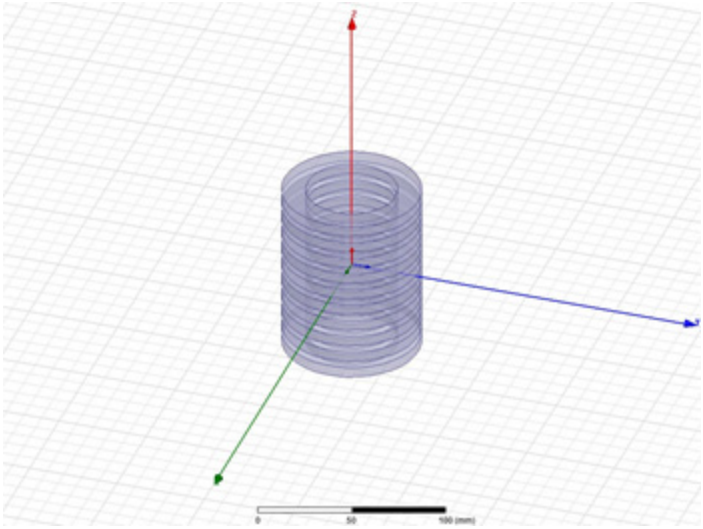


Figure 9-109 One-Leg Coil for TransCoil UDP

TransTwist UDP

Property	Description
CoreType	1: shell-type; 2: core-type
WidthOut	Core outer width
DepthOut	Core outer depth in coil turn direction
RadiusOut	Core outer fillet radius
WidthIn	Core window width
DepthIn	Core window depth in coil turn direction
RadiusIn	Core window fillet radius
DistCenters	Window center-to-center distance, for shell-type only
HeightCore	Core height
Coils	Number of coils: 1 or 2
WiresCoil	Wires per coil: 1, 2, 3, or 4 (no twist for 1 wire)
TurnsCoil	Number of turns per coil
TurnsTwist	Total twist turns in one coil turn; 360 degrees in one twist turn
DiaWire	Wire diameter
Segments	Polygon segments for wire cross section: <=2 for true circle
InsType	Insulator type: 1: round insulator; 2: filleted square insulator
ThickIns	Outer minimum thickness of insulator
GapWires	The gap between two round wire profiles

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Property	Description
GapEndTurns	The gap between two end turns of round insulator profiles
GapCoreIns	The gap between core and round insulator profiles inside two windows and at two ends
LeadExt	Lead extension from lead terminal to core end terminal
InfoCore	0: All; 1: Core; 2: Insulator; 3-6: Wires of coil#1; 7-10: Wires of coil#2
InfoTerm	0: 3D wire; 1: 2D go-term; 2: 2D re-term (for InfoCore = 3-10 only)

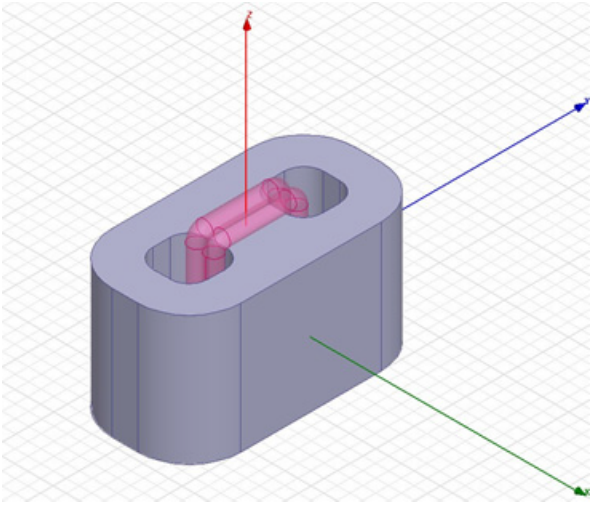


Figure 9-110 Overview of TransTwist UDP without Wires

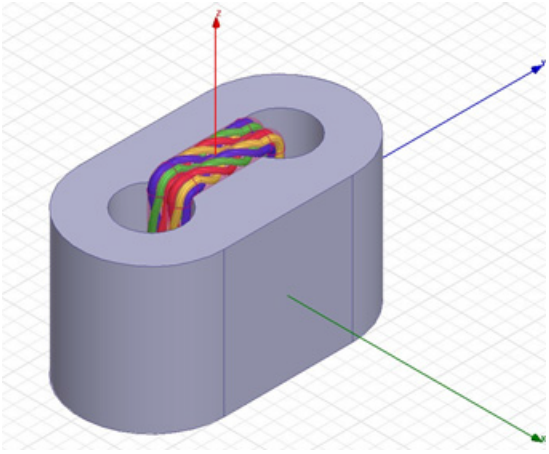


Figure 9-111 Overview of TransTwist UDP with Wires

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- " Overview of TransTwist UDP without Wires" on the previous page and " Overview of TransTwist UDP with Wires" on the previous page give an overview of the TransTwist UDP. The difference between them is " Overview of TransTwist UDP with Wires" on the previous page has four colorful wires inside the insulator whereas " Overview of TransTwist UDP without Wires" on the previous page has nothing in its insulator.
- In" Overview of TransTwist UDP without Wires" on the previous page and " Overview of TransTwist UDP with Wires" on the previous page and also other figures like " Gap-Related Parameters of TransTwist UDP" on page 9-228 , objects in different colors are all TransTwist UDP but with different **InfoCore** values. For example, in Figure 1-6, the gray object is a TransTwist UDP whose **InfoCore** value is 1(core), while red, blue, green and yellow objects (wires) are also TransTwist UDPs whose **InfoCore** values are 3, 4, 5, and 6 respectively. Also, the value of the **InfoCore** of pink tube that covers the wires is 2 (insulator). To draw this kind of figure, copy and paste the TransTwist UDP (or manually create multiples of this UDP) and then change their **InfoCore** values and colors.
- The reason for making colorful figures here is to show the parameters of the TransTwist UDP more clearly.
- The difference between " 3D Parameters of TransTwist UDP" on the facing page and " 3D Parameters of TransTwist UDP with One Turn" on page 9-228 is the parameter **TurnsCoil**, which means the number of turns for each coil (only one coil in each figure). There are 2 turns for the coil in Figure 1-4 whereas there is only 1 turn for the coil in " 3D Parameters of TransTwist UDP with One Turn" on page 9-228 .
- The purpose of" Coil-Related Parameters of TransTwist UDP" on page 9-229 is to show the concept of the parameter **Coils**. Note that when the value of **Coils** is 2 in this figure, the two coils are disconnected. For a comparison, see the one coil shown in " Gap-Related Parameters of TransTwist UDP" on page 9-228 .
- The purpose of " GapEndTurns of TransTwist UDP" on page 9-229 is to show the parameter of **GapEndTurns**, which is useful only when the value of **TurnsCoil** is larger than 2.
- The results of parameter **TurnsTwist** are difficult to show in a figure. It is used to define the number of twist turns in one coil turn for each coil (or wire). As shown in " LeakExt of TransTwist UDP" on page 9-230 , the green wire is twisted 0.5 turn at the bottom of the core. Then, for the entire coil turn, it may be twisted 3 to 3.5 twist turns (0.5 to 1 turn on the top of the core, and 1 turn on each side of the core).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

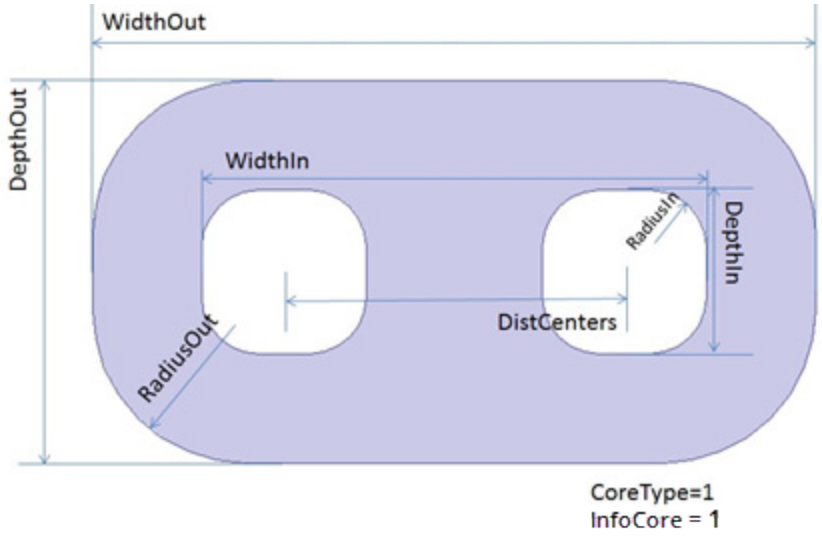


Figure 9-112 2D Parameters of TransTwist UDP

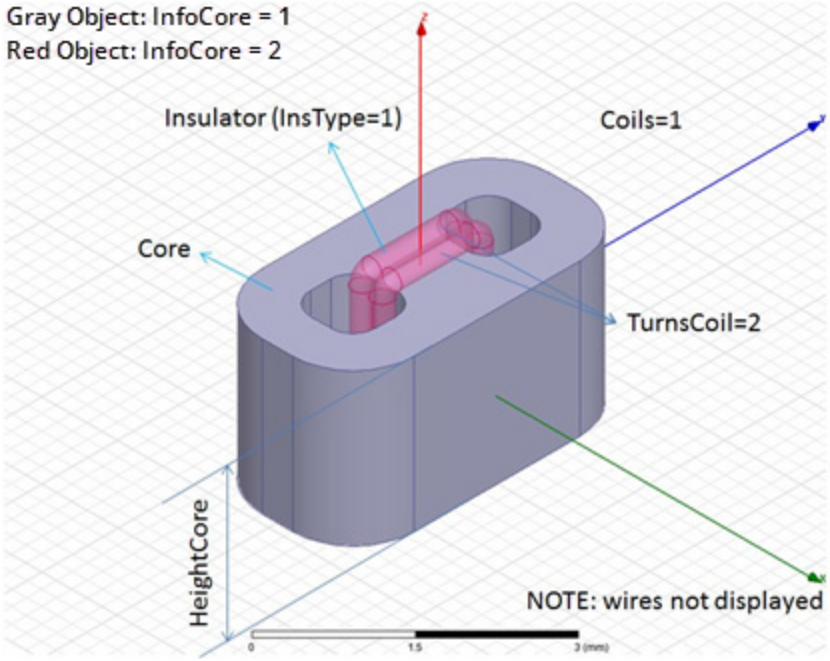


Figure 9-113 3D Parameters of TransTwist UDP

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

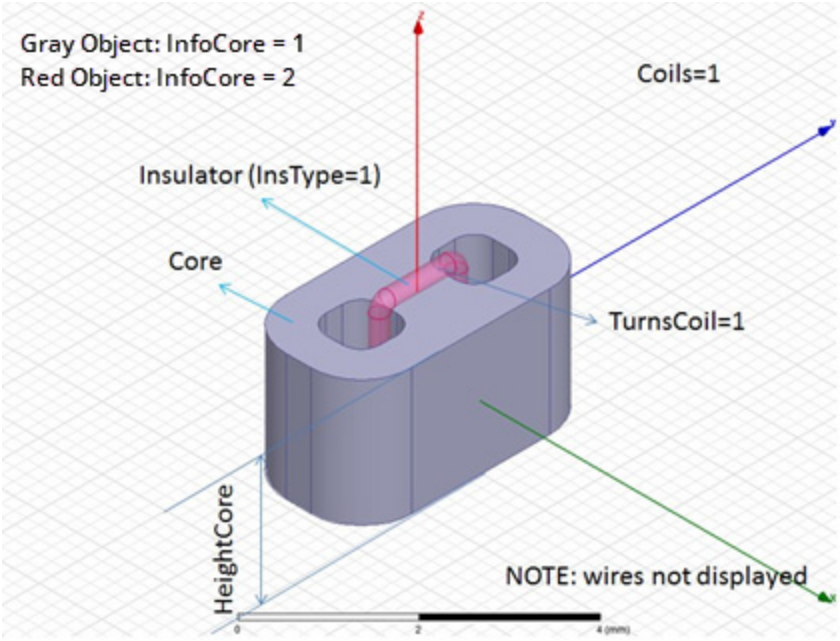


Figure 9-114 3D Parameters of TransTwist UDP with One Turn

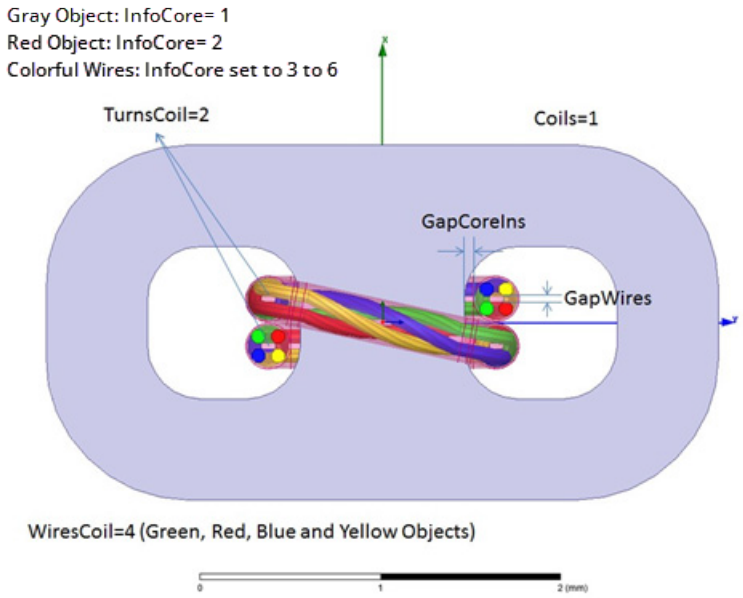


Figure 9-115 Gap-Related Parameters of TransTwist UDP

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

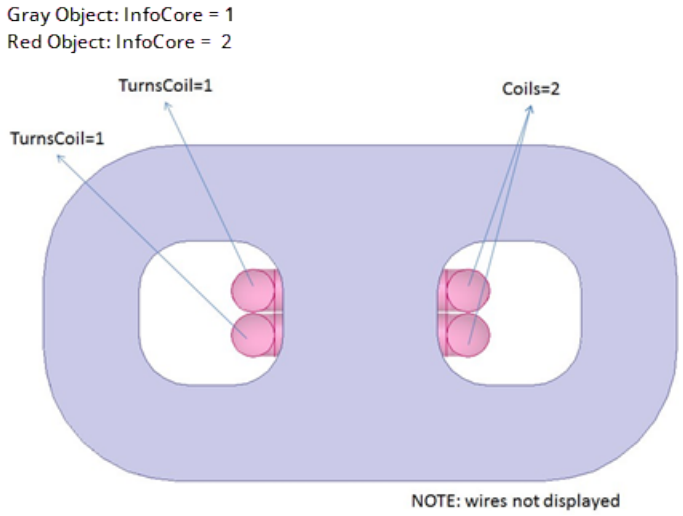


Figure 9-116 Coil-Related Parameters of TransTwist UDP

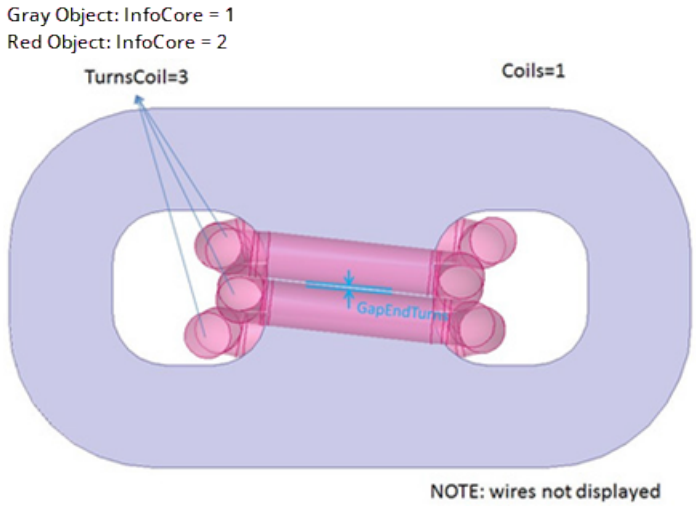


Figure 9-117 GapEndTurns of TransTwist UDP

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

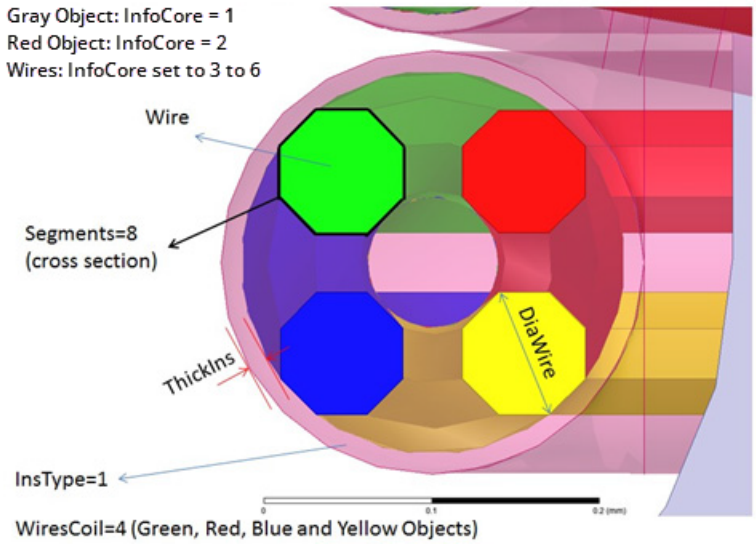


Figure 9-118 Wire-Related Parameters of TransTwist UDP

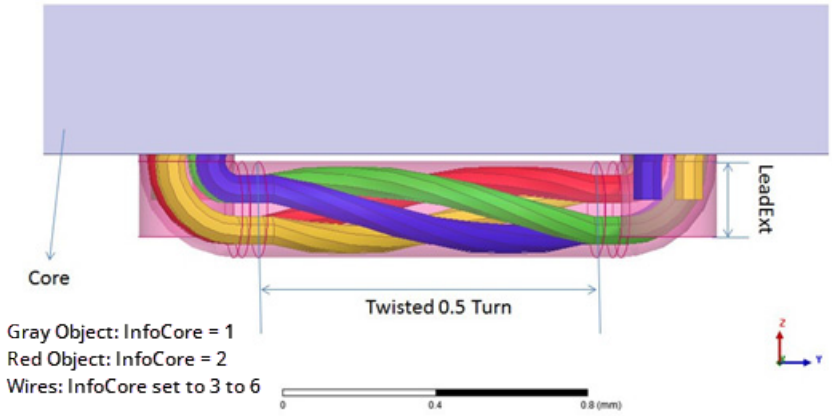


Figure 9-119 LeakExt of TransTwist UDP

Creating a Core and All Twist Coils

You can create a core and all twist coils manually by setting the value of InfoCore to 0, which is set by default in the Maxwell application.

An example of a core and all twist coils is shown in " TransTwist UDP Core" on the facing page.

You can assign the value of this property either when creating the UDP or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

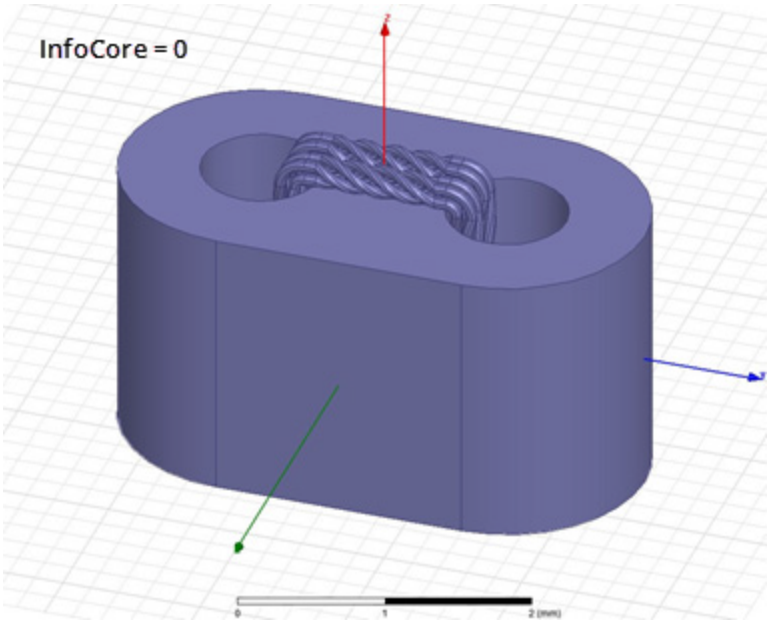


Figure 9-120 TransTwist UDP Core

Creating a Twist Wire and its Terminals

You can create a twist wire of a TransTwist manually by setting the value of **InfoCore** to 3 to 6 (or 7-10 for coil #2) and **Info Term** to 0. Examples are shown in " Overview of TransTwist UDP with Wires" on page 9-225 , " 3D Parameters of TransTwist UDP with One Turn" on page 9-228 , " Wire-Related Parameters of TransTwist UDP" on the previous page, and " LeakExt of TransTwist UDP" on the previous page as red, green, blue and yellow objects.

You can create a terminal of a wire manually by setting the value of **InfoCore** to 3 to 6 (or 7-10 for coil #2) and **Info Term** to 1 or 2 (for 2D go-term and 2D re-term respectively). An example is shown in " 2D Go-Terminal of a Wire" on the next page below.

You can assign the value of this property either when creating a TransTwist or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

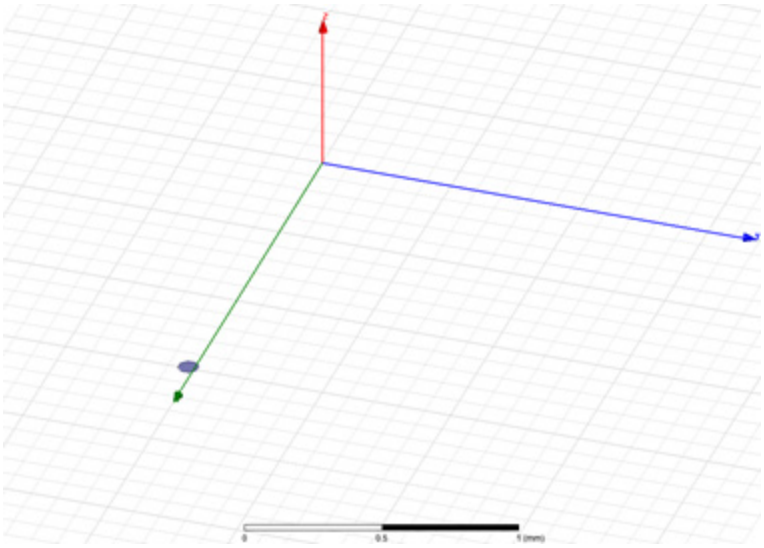
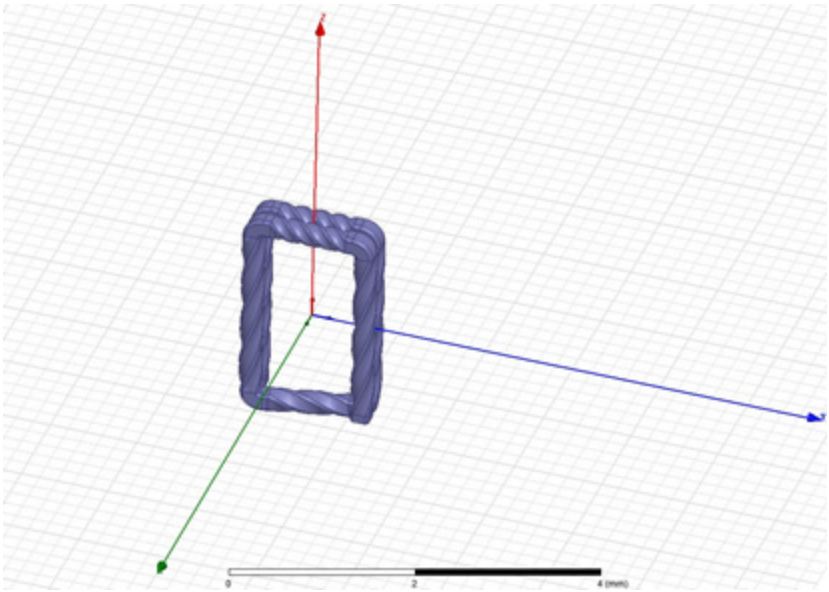


Figure 9-121 2D Go-Terminal of a Wire

Creating a Twist Insulator of All Coils

You can create an insulator of all coils manually by setting the value of **InfoCore** to 2 and **Info Term** to 0. An example is shown in " Twist Insulator of All Coils" on the facing page.

You can assign the value of this property either when creating a TransTwist or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.



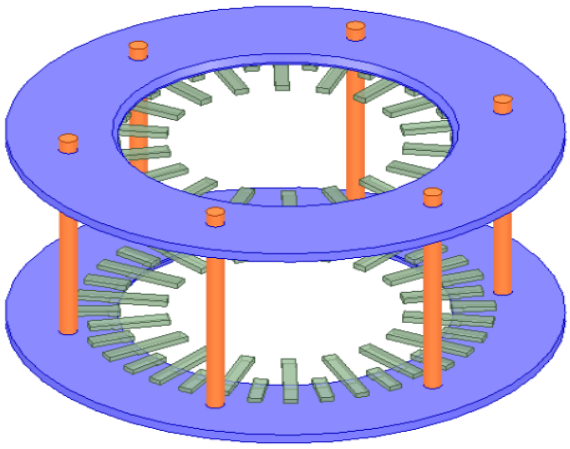
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 9-122 Twist Insulator of All Coils

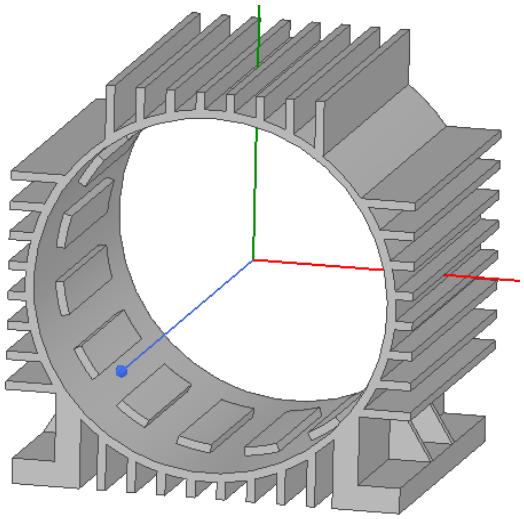
UDPs for Other Machine Parts

RMxprt includes UDPs to add various other types of machine parts. These include

- PressPlate UDP - used to add press plates, fingers, and stems that are used to hold together the stack of core laminations in large AC and DC machines.

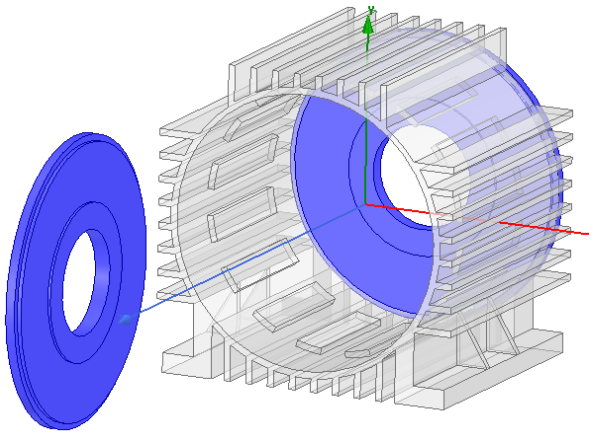


- Housing UDP - used to add machine housings and their associated fins, cooling ducts, and support feet.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- EndCap UDP - used to add the end caps and associated bearing surfaces for a housing.



PressPlate UDP

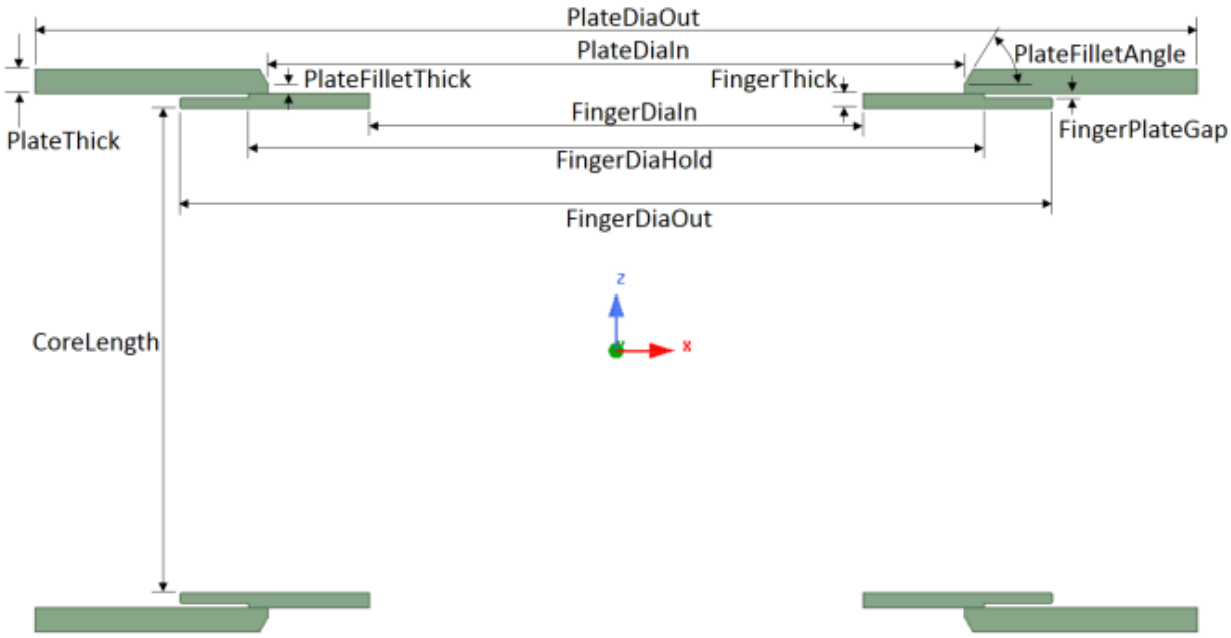
The PressPlate UDP is used to create press plates, fingers, and stems for large AC or DC machines. Press plates and fingers are put at two ends of a large laminated slot core and are locked in place by the stems so that the core laminations can be pressed tightly together.

Property	Description
CoreLength	Axial length of the core inside press plates
PlateDiaIn	Press plate inner diameter
PlateDiaOut	Press plate outer diameter
PlateThick	Press plate axial thickness
PlateFilletThick	Plate fillet thickness at inner diameter
PlateFilletAngle	Plate fillet angle at inner diameter
Fingers	Number of end fingers, or the number of core slots
FingerDiaIn	Inner diameter for distributed fingers
FingerDiaOut	Outer diameter for distributed fingers
FingerDiaHold	Finger holder outer diameter
FingerDiaBot	Inner diameter for slot-bottom fingers, 0 without bottom fingers
FingerWidth	Finger width to cover tooth width
FingerThick	Finger holder axial thickness
FingerPlateGap	Length of the axial gap between fingers and plates
Stems	Number of stems to lock two end press plates.
StemLength	Axial stem length
StemDia	Stem diameter

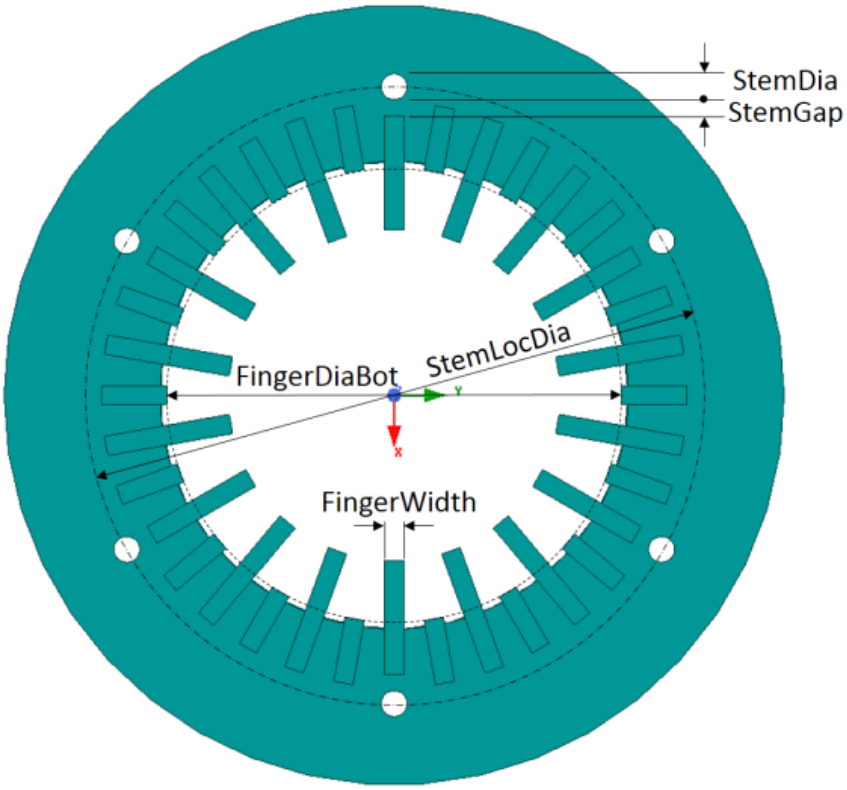
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

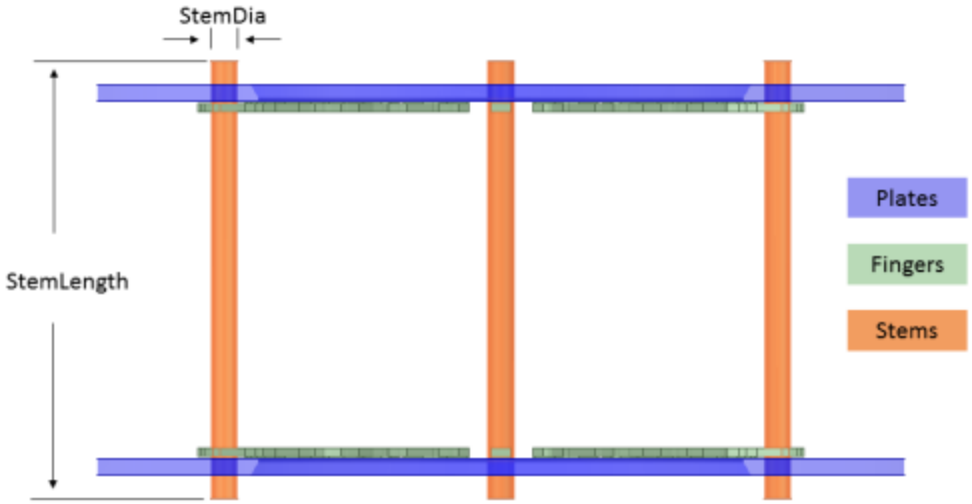
Property	Description
StemLocDia	Diameter of stem center layout circle
StemGap	Clearance gap around a stem
InfoPart	Enter 0 to generate all parts (plates, fingers, and stems); 1 for press plates only; 2 for fingers; 3 for stems

These parameters are shown in the following 2D examples:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



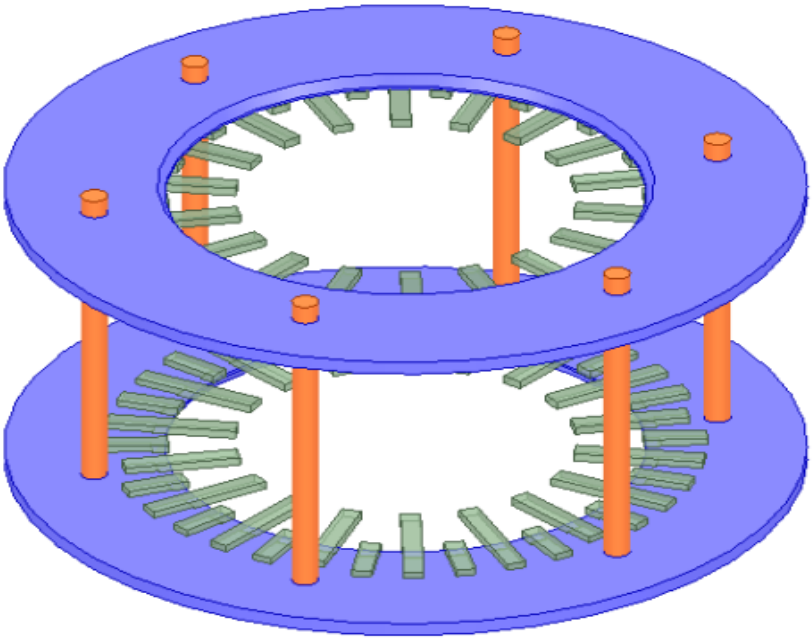


The following figure shows a typical arrangement of press plates, fingers, and stems in which there are eighteen fingers under the press plates. The six stems serve to "lock" these parts at the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

two ends of the machine's laminated core (not shown), holding the entire assembly firmly together.

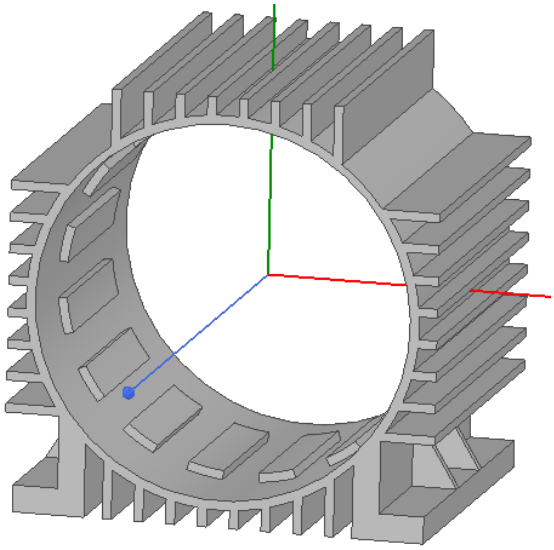
Press plates with fingers, and stems



Housing UDP

The Housing UDP is used to create the housing parts for machines. These include the fins, cooling ducts, and feet that support machines.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



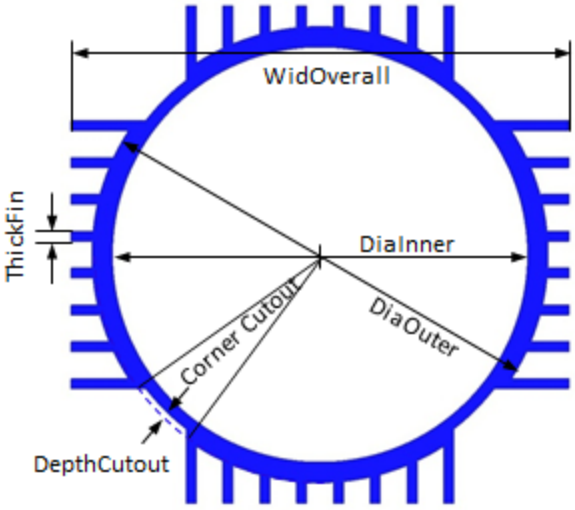
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

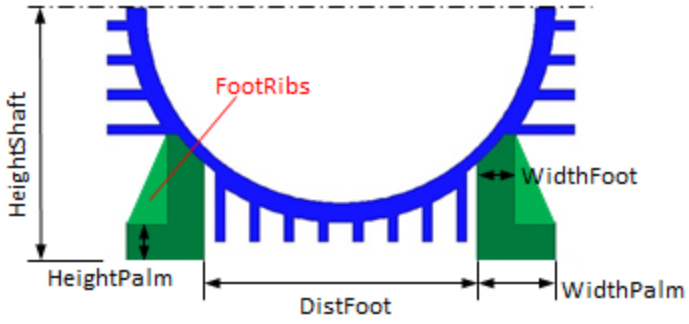
Property	Description
DiaOuter	Housing outer diameter
DiaInner	Housing inner diameter
WidOverall	Housing overall width
Length	Housing length
Fins	Number of fins; Fins = 0 for uniform distribution
FinType	1: radial round; 2: radial square; 3: parallel round; 4: parallel square; 5: square ring; 6: servo ring
RatioCutout	Corner cutout ratio for FinType≠5
DepthCutout	Corner cutout depth for FinType≠5; min fin depth for FinType=5
ThickFin	Fin thickness
Ducts	Number of ducts
DuctType	1: axial wind duct; 2: axial water jacket; 3: spiral water jacket (see below)
HeightDuct	Duct height in radial direction
EmbraceDuct	Ratio of the duct span, or duct width, to duct pitch 0 < EmbraceDuct < 1
LengthDuct	Axial duct length for axial wind ducts only
SegAngle	Deviation angle for spiral water jacket; 0 for true-surface circular jacket
WidthFoot	Foot width at the junction with housing
LengthFoot	Foot end-to-end length in axial direction
DistFoot	Distance between two feet
DistAxial	Foot distance in axial direction
WidthPalm	Palm width at the junction with base plate
HeightPalm	Palm Height
LengthPalm	Palm length in axial direction
HeightShaft	Shaft-center Height from palm bottom
FootRibs	Number of reinforcing ribs for foot and palm
ThickRib	Reinforcing-rib thickness
PitchRib	Pitch between two reinforcing ribs
OffsetFoot	Axial offset of foot center to housing center (not currently used)
OffsetPalm	Axial offset of palm center to housing center

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

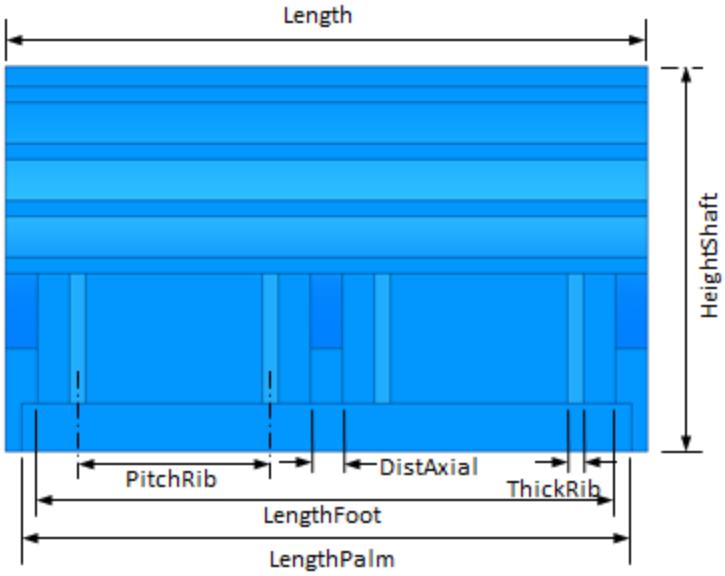
Property	Description
	(not currently used)
InfoPart	0: housing; 100: ducts

These parameters are shown in the following 2D examples:

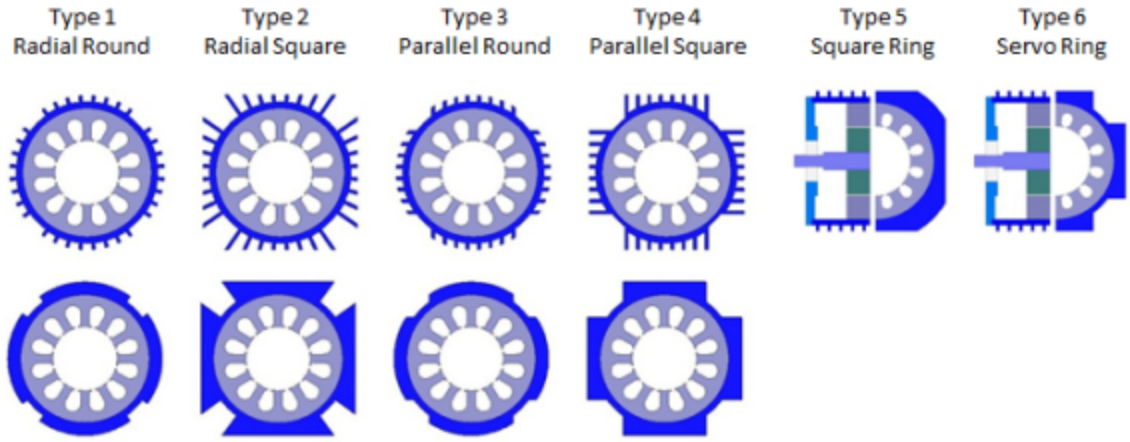




Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

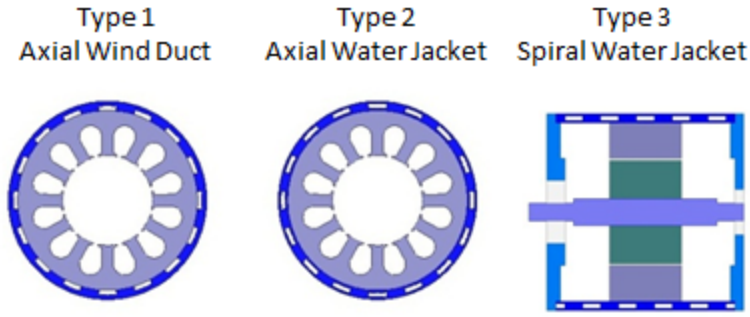


Housing FinType



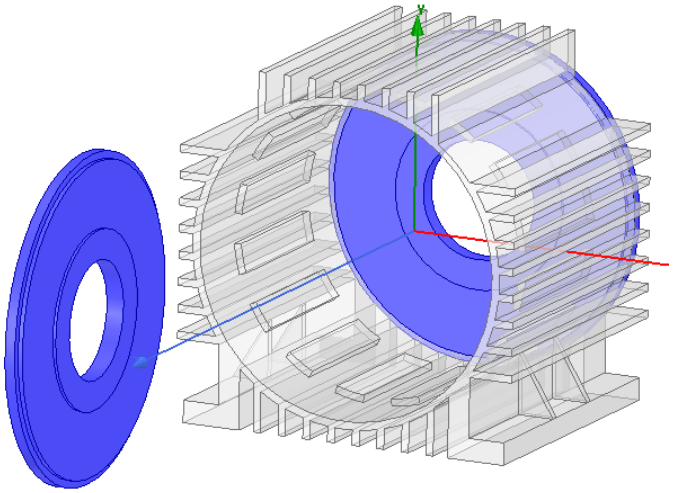
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Housing DuctType



EndCap UDP

The EndCap UDP is to used to add the end caps, bearing surfaces, and rectangular cooling plate for a housing.

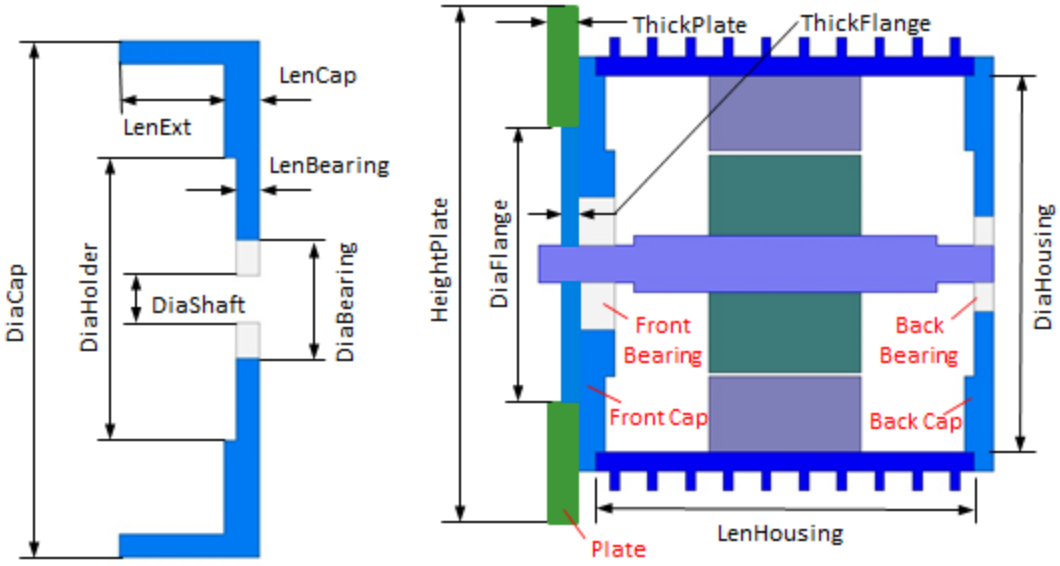


Property	Description
DiaHousing	Housing rabbet inner diameter
LenHousing	Housing axial length
DiaCap	End cap outer diameter
LenCap	Axial length, or thickness, of end cap
LenExt	Length from housing end to cap inner surface, <0 for rabbet length
DiaHolder	Outer diameter of bearing holder
DiaBearing	Bearing outer diameter

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Property	Description
LenBearing	Bearing axial length
DiaShaft	Shaft diameter, or bearing inner diameter
DiaFlange	Diameter of the flange section that covers the front bearing
ThickFlange	Thickness of the flange section that covers the front bearing
HeightPlate	Height of the rectangular cooling plate
WidthPlate	Width of the rectangular cooling plate
ThickPlate	Thickness of the rectangular cooling plate
InfoPart	0: front cap; 1: back cap; 2: front bearing; 3: back bearing; 4: flange.

These parameters are shown in the following 2D example:



Helix UDPs

Helix UDPs are shapes of inductance design and are widely used in electronic devices and signal integrity area.

Helix UDPs consist of the following types:

- PolygonHelix UDP
- RectHelix UDP

PolygonHelix UDP

The shape of the cross-section of a PolygonHelix is a polygon.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Property	Description
PolygonSegments	Number of cross-section polygon segments, 0 for circle
PolygonRadius	Outer radius of cross-section polygon
StartHelixRadius	Start radius from polygon center to helix center
RadiusChange	Radius change per turn
Pitch	Helix pitch
Turns	Number of turns
SegmentsPerTurn	Number of segments per turn, 0 for true surface
RightHanded	Helix direction, non-zero for right handed

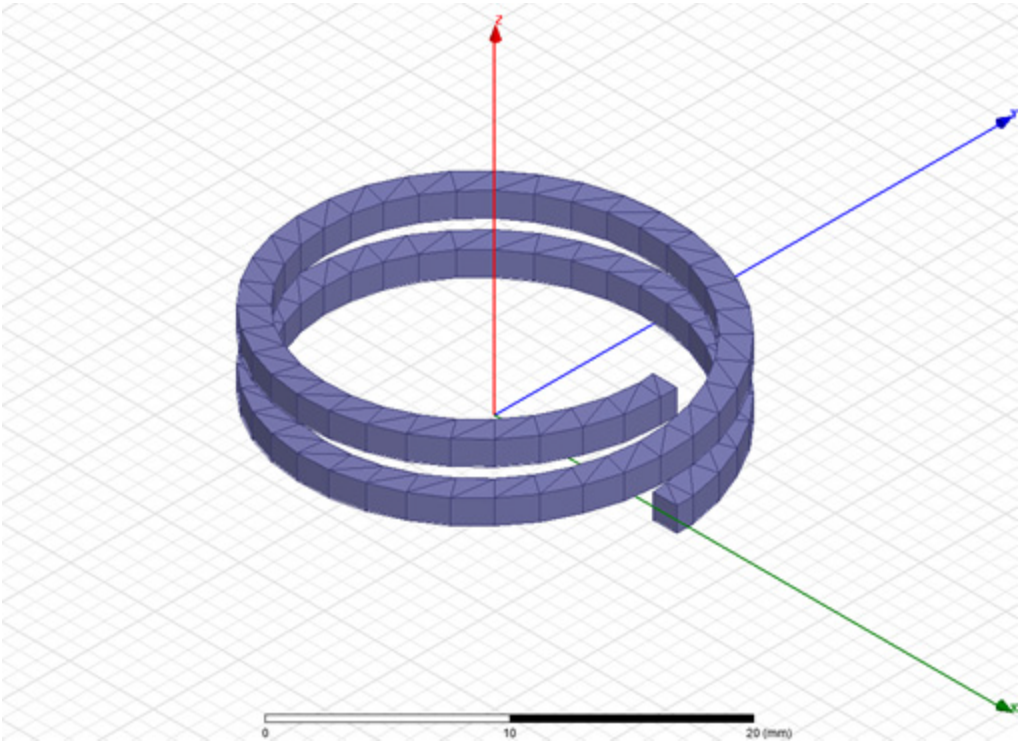


Figure 9-123 Overview of PolygonHelix UDP

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

" Overview of PolygonHelix UDP" on the previous page is an overview of the PolygonHelix UDP. “Polygon” means the cross-section of the helix is a polygon, which by default is a square (**PolygonSegments** set to 4).

Polygon helices in " Parameters of PolygonHelix UDP on X-Y Plane" below and " Parameters of PolygonHelix UDP on Axial Direction" on the next page have the default settings of this UDP, whereas each of the polygon helices in " Non-Zero RadiusChange Value" on the next page to " Turns Value = 3" on page 9-248 has only one parameter value that is different from the defaults shown in " Parameters of PolygonHelix UDP on X-Y Plane" below and " Parameters of PolygonHelix UDP on Axial Direction" on the next page.

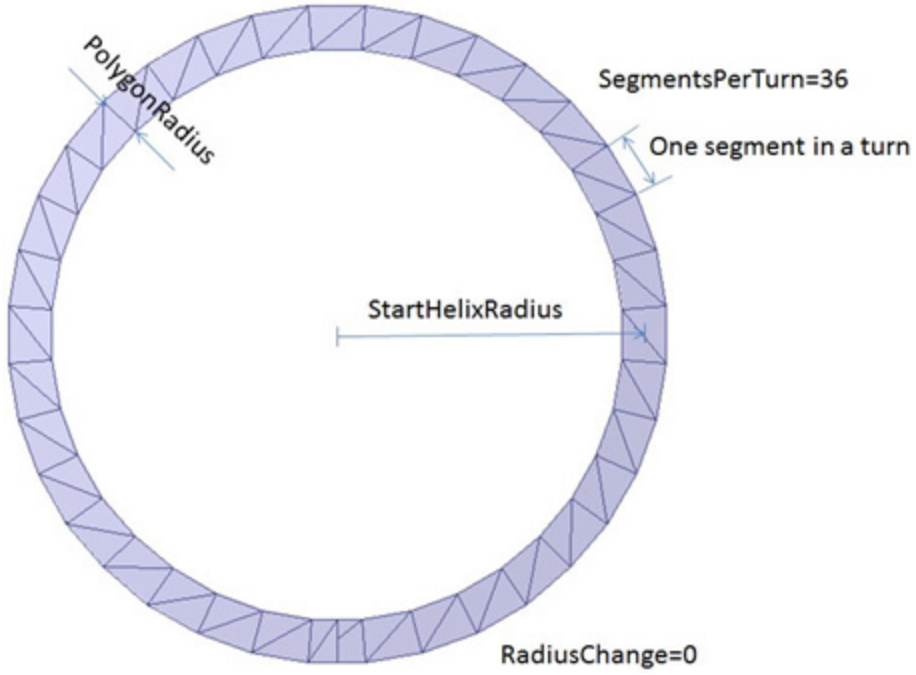


Figure 9-124 Parameters of PolygonHelix UDP on X-Y Plane

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

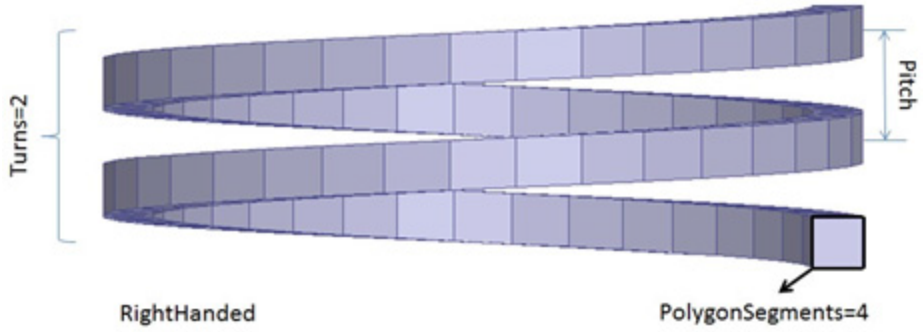


Figure 9-125 Parameters of PolygonHelix UDP on Axial Direction

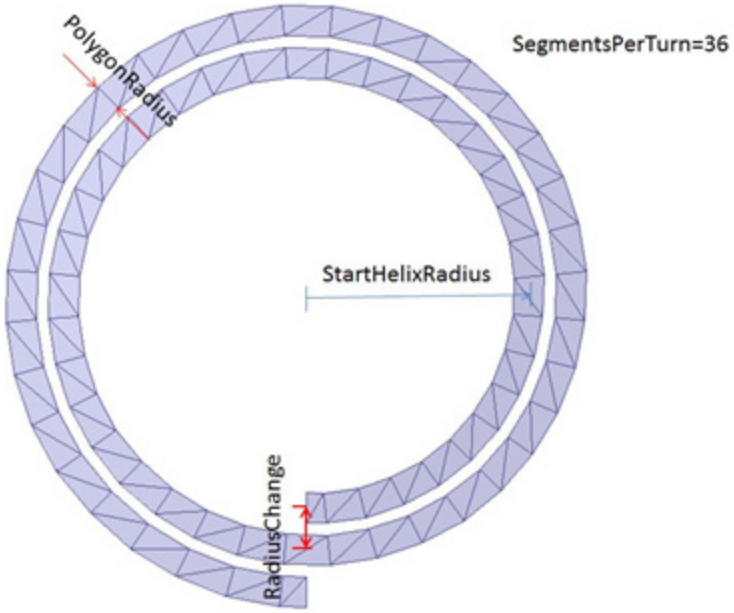


Figure 9-126 Non-Zero RadiusChange Value

Figure 9-127 Compare to " Parameters of PolygonHelix UDP on X-Y Plane" on the previous page, where the RadiusChange = 0.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

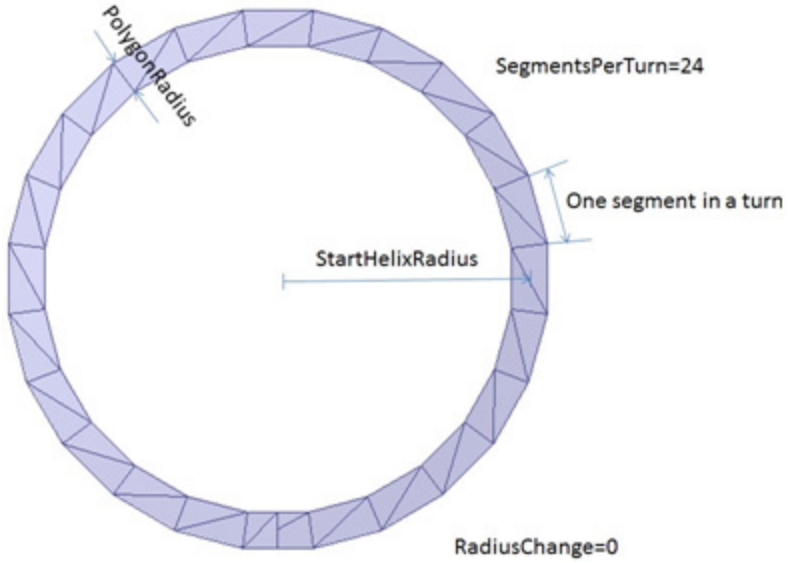


Figure 9-128 SegmentsPerTurn = 24

Figure 9-129 Compare to " Parameters of PolygonHelix UDP on X-Y Plane" on page 9-245 , where the value is 36.

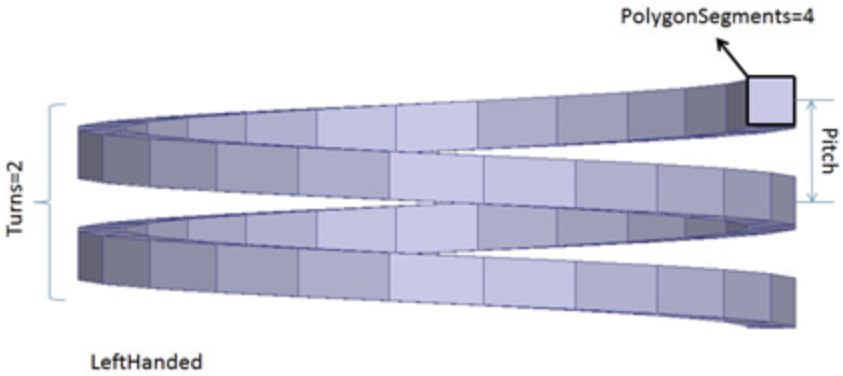


Figure 9-130 Left-Handed (RightHanded set to 0) Helix

Figure 9-131 " Parameters of PolygonHelix UDP on Axial Direction" on the previous page shows a right-handed helix.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

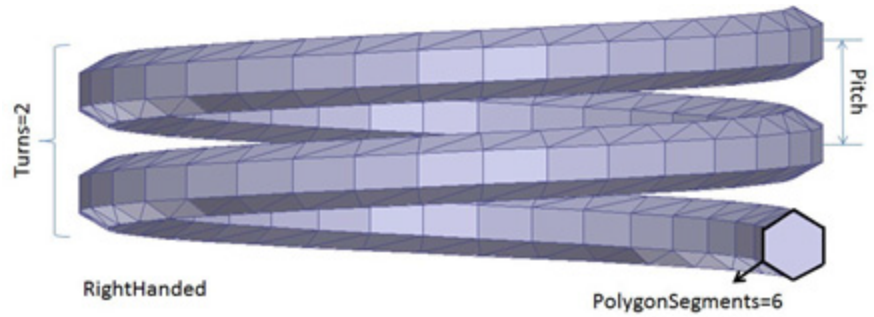


Figure 9-132 PolygonSegments Value = 6

Figure 9-133 Compare to " Parameters of PolygonHelix UDP on Axial Direction" on page 9-246 , where the value is 4.

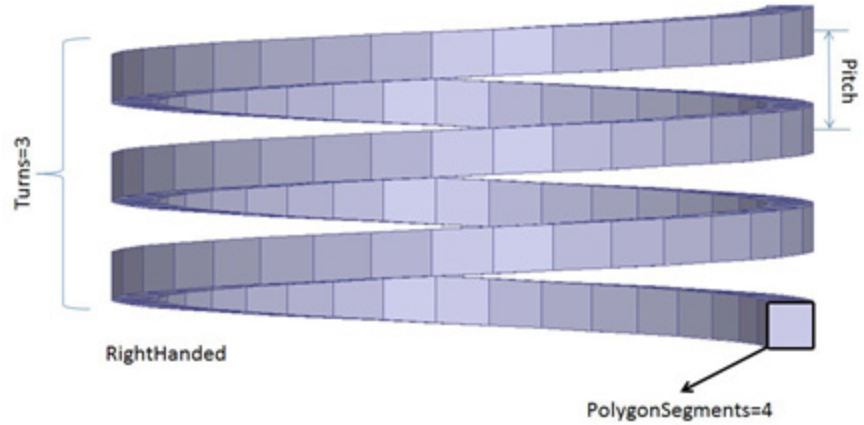


Figure 9-134 Turns Value = 3

Figure 9-135 Compare to " Parameters of PolygonHelix UDP on Axial Direction" on page 9-246 , where the value is 2.

Creating a Right-Handed or Left-Handed Polygon Helix

You can create a right-handed polygon helix manually by setting the value of **RightHanded** to 1 (or any non-zero value), which is set by default in Maxwell.

You can create a left-handed polygon helix by setting **RightHanded** to 0.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Examples of right- and left-handed polygon helices are shown in " Parameters of PolygonHelix UDP on Axial Direction" on page 9-246 and " Left-Handed (RightHanded set to 0) Helix" on page 9-247 , respectively.

You can assign the value of this property either when creating the UDP or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

RectHelix UDP

The shape of the cross-section in a RectHelix is a rectangle.

Property	Description
RectHeight	Cross-section rectangle height in z direction
RectWidth	Cross-section rectangle width in r direction
StartHelixRadius	Start radius from rectangle center to helix center
RadiusChange	Radius change per turn
Pitch	Helix pitch
Turns	Number of turns
SegmentsPerTurn	Number of segments per turn, 0 for true surface
RightHanded	Helix direction, non-zero for right handed

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

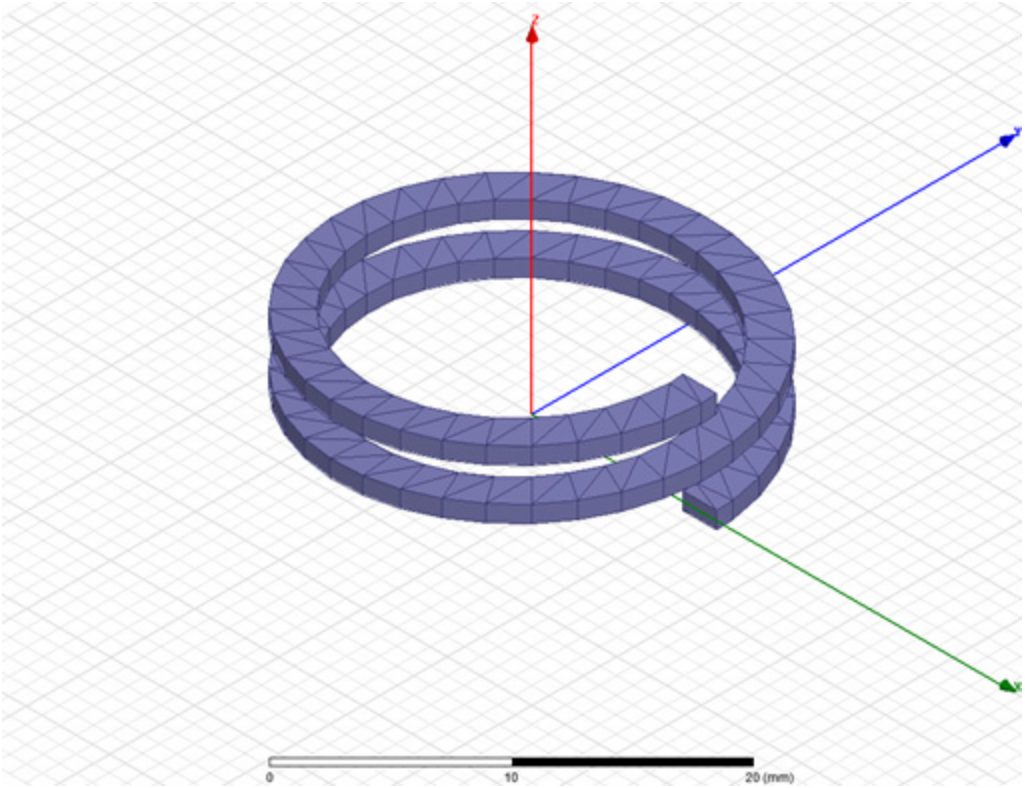


Figure 9-136 Overview of RectHelix UDP

Note:

- The major difference between the RectHelix UDP and the PolygonHelix UDP is the shape of the cross-sections of the helices. In RectHelix, the shape of its cross-section is a rectangle whereas that of PolygonHelix is a polygon.
- Except for **RectHeight** and **RectWidth**, parameters are the same as those in the Polygon Helix UDP. The one figure in this section describes **RectHeight** and **RectWidth**.
- For *all other* parameters, refer to Polygon Helix UDP.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

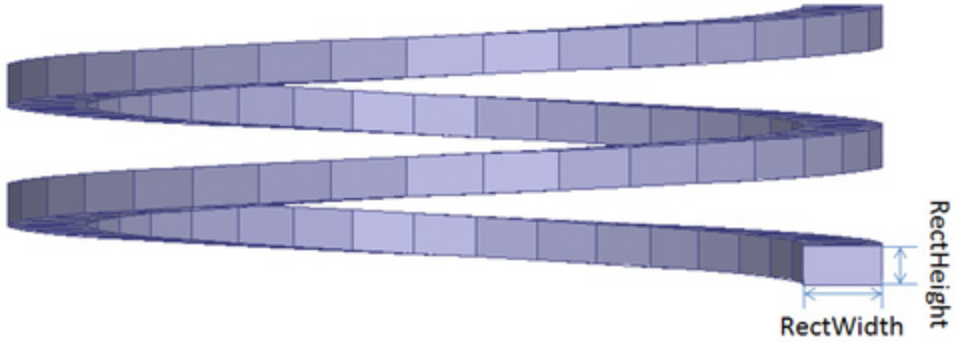


Figure 9-137 RectHeight and RectWidth in RectHelix UDP

Creating a Right-Handed or Left-Handed Rectangle Helix

You can create a right-handed rectangle helix manually by setting the value of **RightHanded** to 1 (or any non-zero value), which is set by default in Maxwell.

You can create a left-handed rectangle helix by setting **RightHanded** to 0.

Examples of right-handed and left-handed rectangle helices are shown in " Turns Value = 3" on page 9-248 and " Left-Handed Rectangle Helix" below, respectively.

You can assign the value of this property either when creating the UDP or after creating it, as described in Creating a Band and Creating an Outer Region, respectively.

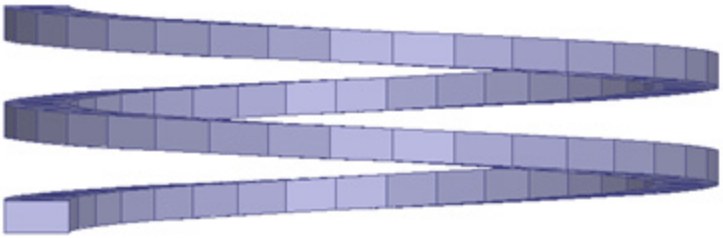


Figure 9-138 Left-Handed Rectangle Helix

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Information about Coding UDPs

This chapter uses an example of a Band UDP to provide information about how to write source code for a new UDP class when needed.

- Intrinsic Head Files presents some of the intrinsic structures and functions for all UDPs.
- Code Implementation uses a Band UDP to illustrate what functions or procedures are needed to create a new UDP.

Intrinsic Head Files

The following sections provide information on Intrinsic Structure Definitions and Intrinsic Function Declarations.

Head File for Intrinsic Structure Definition

Many intrinsic structures are written in C++ for UDPs. These intrinsic structures are provided in the UserDefinedPrimitiveStructures.h file (under directory: "...\ANSYS Inc\[%Primary.VersionNumber%]\AnsysEM\UserDefinedPrimitives\Examples\Headers").

Here are some common intrinsic structures that you need to know before starting coding for UDPs:

UDPPosition

Defines the most common and basic structure in a coordinate system, which is used to describe a point in the system.

```
struct UDPPosition
{
double x;
double y;
double z;
};
```

UDPPolylineDefinition

Defines the structure of polyline in a UDP, which is used to create sheets or lines in a UDP.

```
struct UDPPolylineDefinition
{
int noOfPoints;
struct UDPPosition* arrayOfPosition;
int noOfSegments;
struct UDPPolylineSegmentDefinition* arrayOfSegmentDefinition;
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
int isClosed;
int isCovered; /* Only if it is closed, then it can be covered */
};
```

UDPPrimitiveParameterDefinition

Defines the parameter/property definition of each UDP, such as *DiaGap*, *DiaYoke* and *Length* in the Properties window of each UDP. A definition for a parameter includes the name of the parameter, description of the parameter, unit of parameter, and default value of it.

```
struct UDPPrimitiveParameterDefinition
{
char* name; /* Name of the parameter */
char* description; /* some description of the parameter */
UDPUnitType unitType; /* No unit/length/angle */
double defaultValue;
};
```

UDPPolylineSegmentDefinition

Defines line segments used to form a polyline.

```
struct UDPPolylineSegmentDefinition
{
UDPPolylineSegmentType segmentType;
int segmentStartIndex;

/* Below information is required to be set only if you are using
Spline */
int numberOfPoints;

/* Below information is required to be set only if you are using
Angular Arc */
double angle; /* in degrees */
UDPPosition centerPoint;
UDPCoordinateSystemPlane arcPlane;
};
```

UDPSweepOptions

Sweep is a kind of operation to move an object from one position to another and meanwhile creating another object based on the track of the movement. For instance, sweep operations can be used to create a sheet from a line or a 3D object from a sheet. Different kinds of sweep

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

types (and sweep angles) are used in SweepAlongAxis and SweepAroundAxis functions to make a sheet or a 3D object.

```
struct UDPSweepOptions
{
UDPSweepDraftType draftType;
double draftAngle;
double twistAngle;
};
```

UDPVector

Vector structures are used in UDP coding, mainly for rotation, moving or other kinds of transformations of geometry.

```
struct UDPVector
{
double x;
double y;
double z;
};
```

Head File for Intrinsic Function Declaration

Intrinsic function declarations are inside the same file (UserDefinedPrimitiveStructures.h) as intrinsic structure definitions. Functions are declared by `typedef` keyword and function pointer. Some function declarations are shown below as examples:

Add a Message to the Message Window, with Desired Severity Level

This declaration is used in the DisplayInfo function in UDPs:

```
typedef void (*UDPAddMessageFP)
(
 UDPMessageSeverity, /* Message severity */
const char*, /* message string */
void* /* callbackData */
);
```

Function Pointers Used to Create Primitives

```
/* The return type for all primitives (except for spiral and helix)
is the ID of
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
 * the body created -1 is an invalid ID, which means the operation
failed.
 *
 * Spiral and helix are like sweep operations and operate on a
profile.
 * The return value indicates if the operation succeeded,
 * return value 0 means operation failed, any other value
 * means operation succeeded.
 */
/* 1D/2D ENTITIES */
typedef long (*UDPCreatePolylineFP)
(
const struct UDPPolylineDefinition* , /* polylineDefinition */
void* /* callbackData */
);

/* 3D ENTITIES */
typedef long (*UDPCreateCylinderFP)
(
UDPCoordinateSystemAxis, /* axis */
const struct UDPPosition*, /* center */
double, /* radius */
double, /* height */
void* /* callbackData */
);
```

Sweep Functions

Sweep functions are used to create a sheet from a line or create a solid (3D object) from a sheet, as shown below:

```
  /* Sweep operations (Sweep along vector, around axis and
 * along path) modify the profile body. The resultant body
 * id is same as the profile id.
 *
 * In case of path sweep, the path body is consumed during the
 * sweep operation.
 *
 * The return value indicates
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
 * if the operation succeeded, return value 0 means operation
 * failed, any other value means operation succeeded.
 */
/* SweepAlongPath is used to sweep along a specific path to create a
sheet (from a line) or a solid (from a sheet)*/
typedef long (*UDPSweepAlongPathFP)
(
long, /* profileID */
long, /* pathID */
const struct UDPSweepOptions*, /* sweepOptions */
void* /* callbackData */
);

/* SweepAroundAxis is used to sweep around a specific axis (x, y or
z) to create a sheet (from a line) or a solid (from a sheet)*/
typedef long (*UDPSweepAroundAxisFP)
(
long, /* profileID */
UDPCoordinateSystemAxis, /* sweepAxis */
double sweepAngle, /* sweepAngle */
const struct UDPSweepOptions*, /* sweepOptions */
void* /* callbackData */
);
```

Transform Operations

The following two functions are used for transformations:

```
/* Transform operations (Translate: move, rotate, mirror and
 * scale) modify the part body. The resultant body
 * id is same as the part id. The return value indicates
 * if the operation succeeded, return value 0 means operation
 * failed, any other value means operation succeeded.
 */
/* Translate function is used to do translational motion for a given
object. */
typedef long (*UDPTranslateFP)
(
long, /* partID */
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
const struct UDPVector*, /* translation vector */
void* /* callbackData */
);

/* Rotate function is used to rotate a given object. */
typedef long (*UDPRotateFP)
(
long, /* partID */
UDPCoordinateSystemAxis, /* axis */
double, /* angle */
void* /* callbackData */
);
```

Duplicate Operation

```
/* Duplicate operations (Duplicate along line, around axis
 * and duplicate and mirror) create new bodies.
 *
 * In case of duplicate and mirror, only 1 duplicate can be
 * created. The ID of the duplicate body is returned by the
 * function. ID of -1 means the operation failed.
 *
 * For the other two cases multiple duplicates can be created.
 * The IDs of all such duplicates are returned in the function
 * parameter. The function return value indicates if the operation
 * succeeded, return value 0 means operation failed, any other
 * value means operation succeeded.
 */
/* Duplicate an object around a given axis (of course can be used to
create either single or multiple copies around the axis) */
typedef long (*UDPDuplicateAroundAxisFP)
(
long, /* partID */
UDPCoordinateSystemAxis, /* axis */
double, /* angle */
int, /* num of clones (including original) */
long*, /* returned clone ids */
void* /* callbackData */
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
);
```

Boolean Operations

Boolean operations are used to unite, subtract, or intersect two objects.

```
/* Boolean operations (unite, subtract and intersect) modify the
 * blank body. The resultant body id is same as the blank id.
 * Tool bodies are consumed during the operation.
 * The return value indicates if the operation succeeded,
 * return value 0 means operation failed, any other value
 * means operation succeeded.
 */
/* Unite two objects */
typedef long (*UDPUniteFP)
(
long*, /* objectIDs */
int, /* numObjects */
void* /* callbackData */
);

/* Subtract two objects */
typedef long (*UDPSubtractFP)
(
long*, /* blankIDs */
int, /* numBlanks */
long*, /* toolIDs */
int, /* numTools */
void* /* callbackData */
);
```

Function Library with a Collection of UDP Functions

When coding for a UDP, you should declare an instance of UDPFunctionLib in the UDP and then use functions directly when needed.

For example:

```
UDPFunctionLib* functionLib;
functionLib->addMessage(kUDPInfoMessage, msg0, callbackData);
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
struct UDPFunctionLib
{
/* Utility functions */
UDPAddMessageFP addMessage;
UDPNameAFaceFP nameAFace;
UDPNameAEdgeFP nameAEdge;
UDPNameAVertexFP nameAVertex;
UDPGetFaceIDFromPositionFP getFaceIDFromPosition;
UDPGetEdgeIDFromPositionFP getEdgeIDFromPosition;
UDPGetTempDirPathFP getTempDirPath;
UDPGetInstallDirPathFP getInstallDirPath;
UDPGetSysLibDirPathFP getSysLibDirPath;
UDPGetUserLibDirPathFP getUserLibDirPath;
UDPGetPersonalLibDirPathFP getPersonalLibDirPath;
UDPSetProgressFP setProgress;


/* Primitives */
/* 1D/2D ENTITIES */
UDPCreatePolylineFP createPolyline;
UDPCreateRectangleFP createRectangle;
UDPCreateArcFP createArc;
UDPCreateCircleFP createCircle;
UDPCreateEllipseFP createEllipse;
UDPCreateRegularPolygonFP createRegularPolygon;
UDPCreateEquationBasedCurveFP createEquationBasedCurve;
UDPCreateEquationBasedSurfaceFP createEquationBasedSurface;
UDPCreateSpiralFP createSpiral;

UDPGetProjectPathFP getProjectPath;


/* BEGIN - RESERVED FOR FUTURE 1D/2D PRIMITIVES */
/* DO NOT INVOKE THESE RESERVED FUNCTIONS IN YOUR UDP DLL CREATION
CODE */
UDPReservedForFutureFP reserved1D2DPrimitive5;
UDPReservedForFutureFP reserved1D2DPrimitive6;
UDPReservedForFutureFP reserved1D2DPrimitive7;
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
UDPReservedForFutureFP reserved1D2DPrimitive8;
UDPReservedForFutureFP reserved1D2DPrimitive9;
UDPReservedForFutureFP reserved1D2DPrimitived10;
/* END - RESERVED FOR FUTURE 1D/2D PRIMITIVES */


/* 3D ENTITIES */
UDPCreateBoxFP createBox;
UDPCreateSphereFP createSphere;
UDPCreateCylinderFP createCylinder;
UDPCreateConeFP createCone;
UDPCreateTorusFP createTorus;
UDPCreatePolyhedronFP createPolyhedron;
UDPCreateHelixFP createHelix;


/* BEGIN - RESERVED FOR FUTURE 3D PRIMITIVES */
/* DO NOT INVOKE THESE RESERVED FUNCTIONS IN YOUR UDP DLL CREATION
CODE */
UDPReservedForFutureFP reserved3DPrimitive2;
UDPReservedForFutureFP reserved3DPrimitive3;
UDPReservedForFutureFP reserved3DPrimitive4;
UDPReservedForFutureFP reserved3DPrimitive5;
UDPReservedForFutureFP reserved3DPrimitive6;
UDPReservedForFutureFP reserved3DPrimitive7;
UDPReservedForFutureFP reserved3DPrimitive8;
UDPReservedForFutureFP reserved3DPrimitive9;
UDPReservedForFutureFP reserved3DPrimitive10;
/* END - RESERVED FOR FUTURE 3D PRIMITIVES */


/* Operations */
/* Booleans */
UDPUniteFP unite;
UDPSubtractFP subtract;
UDPIntersectFP intersect;

/* Sweeps */
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
UDPSweepAlongVectorFP sweepAlongVector;
UDPSweepAroundAxisFP sweepAroundAxis;
UDPSweepAlongPathFP sweepAlongPath;

/* Transformation */
UDPTranslateFP translate;
UDPRotateFP rotate;
UDPMirrorFP mirror;
UDPScaleFP scale;

/* Duplicate */
UDPDuplicateAlongLineFP duplicateAlongLine;
UDPDuplicateAroundAxisFP duplicateAroundAxis;
UDPDuplicateAndMirrorFP duplicateAndMirror;


/* Connect */
UDPConnectFP connect;

/* Offset */
UDPOffsetFP offset;

/* Section */
UDPSectionFP section;

/* Split */
UDPSplitFP split;

/* Import native body */
UDPImportNativeBodyFP importNativeBody;

/* Clone */
UDPCloneFP clone;

/* Delete part */
UDPDeletePartFP deletePart;
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
/* Create Object From Face */
UDPCreateObjectFromFaceFP createObjectFromFace;

/* Fillet/Chamfer */
UDPFilletFP fillet;
UDPChamferFP chamfer;

/* Detach Faces/Edges */
UDPDetachFacesFP detachFaces;
UDPDetachEdgesFP detachEdges;

UDPCreateObjectFromEdgeFP createObjectFromEdge;

UDPSheetThickenFP sheetThicken;
UDPSweepFaceAlongNormalFP sweepFaceAlongNormal;
/* cover/uncover */
UDPCoverLineFP coverLine;
UDPCoverSurfaceFP coverSurface;
UDPUncoverFacesFP uncoverFaces;

UDPSeparateBodiesFP separateBodies;

UDPMoveFacesFP moveFaces;
UDPWrapSheetFP wrapSheet;
UDPImprintFP imprint;
UDPImprintProjectionFP imprintProjection;

/* Import ansoft geometry */
UDPImportAnsoftGeometryFP importAnsoftGeometry;

/* correctly returns part ids */
UDPImportNativeBody2FP importNativeBody2;

UDPTransformFP transform;


/* BEGIN - RESERVED FOR FUTURE OPERATIONS */
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
/* DO NOT INVOKE THESE RESERVED FUNCTIONS IN YOUR UDP DLL CREATION
CODE */
UDPReservedForFutureFP reservedOperation20;
/* END - RESERVED FOR FUTURE OPERATIONS */
};
```

Code Implementation

This section provides several sample functions and structures to show how to write real code for a UDP. All codes are from a sample source file written for Band UDP.

There are two files, UDPBase.h and UDPBase.cpp, that include basic structures and functions for all UDPs. All UDPs should be built based on these two files.

Define UDP Parameters and Default Values

A structure is needed for parameters and default values at the beginning of each UDP. In the following code, UDPPrimitiveParameterDefinition, an intrinsic structure of UDP shown in Head File for Intrinsic Structure Definition, is used to create an array of parameters for a UDP.

```
struct UDPPrimitiveParameterDefinition primParams[] =
{
{"DiaGap", "Band diameter in gap center, DiaGap<DiaYoke for outer
band",
kUDPLengthUnit, 101},
{"DiaYoke", "Band diameter on yoke side, DiaYoke<DiaGap for inner
band",
kUDPLengthUnit, 40},
{"Length", "Band length", kUDPLengthUnit, 200},
{"SegAngle", "Deviation angle for band (0.1~5 degrees).",
kUDPAngleUnit, 3},
{"Fractions", "Number of circumferential fractions, 1 for circular
region.",
kUDPNoUnit, 1},
{"HalfAxial", "0: full model; 1: half model in axial direction.",
kUDPNoUnit, 0},
{"InfoCore", "0: band; 1: tool; 2: independent; 3: dependent; 100:
region.", kUDPNoUnit, 0}
};
```

Function Inside Band UDP

```
int AreParameterValuesValid(char ** error, double* paramValues)
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
{
obj.IniParameters(paramValues, numOfParameters);
return obj.AreParametersValid(error);
}
```

`IniParameters()` and `AreParametersValid()` are two member functions, which are defined in the UDP base class (in UDPBase.cpp file) and shown in the next topic.

Initialize Parameters and Check Validity

First initialize parameters using `IniParameters()`, and then check the validity of values for parameters. These two functions are inside the UDPBase class and do not need to be implemented.

```
// set oriParams pointer, and make a copy of data to paramValues
void CUDPBase::IniParameters(double* paramValues0, int num)
{
oriParams = paramValues0; // set oriParams pointer
numOfParams = num;
delete[] paramValues;
paramValues = new double [num];
for (int i=0; i<num; i++) paramValues[i] = oriParams[i]; // make a
copy
}
```

The following function is not required to be implemented, but it calls three other functions, `GetParameters()`, `ValidateParameters`, and `PutParameters()`, which should be implemented in the new UDP class.

```
int CUDPBase::AreParametersValid(char ** error)
{
GetParameters(); // the implementation is shown below
checkOK = 0;
char *err = ValidateParameters(); // the implementation is shown
below
if (err) {
*error = err;
return 0;
}
PutParameters(); // the implementation is shown below
checkOK = 1;
return 1;
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
}
```

Parameter Validation Functions

This section covers the three functions that you need to implement in the new UDP classes. The functions could be used as templates for new UDP classes.

These functions are very straightforward. You should follow the examples to add or remove parameters for a new UDP.

GetParameters()

```
/* To get the values from array of parameters*/
void CBand::GetParameters()
{
diaGap = diaYoke = 0;
length = 0;
infoCore = 0;
infoOut = -1;

segAng = 3;
numFras = 1;
infoHalf = 0;

for (int i=0; i<numOfParams; i++) {
if (GetValue(i, "DiaGap", diaGap)) continue;
if (GetValue(i, "DiaYoke", diaYoke)) continue;
if (GetValue(i, "Length", length)) continue;

if (GetValue(i, "SegAngle", segAng)) continue;
if (GetValue(i, "Fractions", numFras)) continue;
if (GetValue(i, "HalfAxial", infoHalf)) continue;
if (GetValue(i, "InfoCore", infoCore)) continue;
}
rRegion = diaYoke / 2;
}
```

PutParameters()

```
/* To store the values into array of parameters*/
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
void CBand::PutParameters()
{
for (int i=0; i<numOfParams; i++) {
if (PutValue(i, "DiaGap", diaGap)) continue;
if (PutValue(i, "DiaYoke", diaYoke)) continue;
if (PutValue(i, "Length", length)) continue;

if (PutValue(i, "SegAngle", segAng)) continue;
if (PutValue(i, "Fractions", numFras)) continue;
if (PutValue(i, "InfoCore", infoCore)) continue;
}
}
```

ValidateParameters()

```
/* To check parameters one by one to see if their values are valid */
char* CBand::ValidateParameters()
{
if (!paramValues) return 0;

if (diaYoke < 0) diaYoke = 0;
if (diaGap <= 0) return "DiaGap must be larger than 0.";
if (length < 0) length = 0; // length==0 for 2D geometry
infoOut = diaYoke>diaGap ? 1 : -1;

if (numFras < 1) numFras = 1;
if (segAng > 5) segAng = 5;
if (segAng <= 0) segAng = 0;
else if (segAng < 0.1) segAng = 0.1;
if (segAng > 0) {
numSegs = (int)(360/segAng + 0.5);
int remain = numSegs % numFras;
if (remain > 0) numSegs += numFras - remain;
segAng = 360.0 / numSegs;
}
else numSegs = 0;
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
if (infoCore < 100) {
if (infoCore < 0) infoCore = 0;
if (infoCore > 3) infoCore = 3;
}

if (infoCore >= 100) {
if (diaYoke <= 0) return "DiaYoke must be larger than 0.";
}
else if (infoCore >= 1) { // for tool; independent & dependent
if (diaYoke <= diaGap) return "DiaYoke must be larger than DiaGap.";
}

return 0;
}
```

Implement Main Function to Create Primitive

In this part, functions are used to create the UDP. You need to write two functions for UDP creation.

- The first function is the main function to create this UDP. It calls two other functions, one of which is shown in part 2.

  ```
  extern "C" DLLEXPORT
  long CreatePrimitive(struct UDPFunctionLib* functionLib, void*
  callbackData,
  double* paramValues)
  {
  obj.SetParameterPointer(paramValues);
  return obj.CreatePrimitive(functionLib, callbackData);
  }
  ```

- The following function is used to create a UDP based on different **InfoCore** values. For instance, a region is created when **InfoCore** is equal to 100 while a tool is created when **InfoCore** is equal to 2.

For functions to create specific object like region, tool and independent, refer to Source Code Files for more detail.

```
long CBand::CreatePrimitive(UDPFunctionLib* functionLib, void*
callbackData)
{
if (!checkOK) return 0;
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
// display information of modified parameters
SetFunctionLib(functionLib, callbackData);
DisplayInfo();

// create region
if (infoCore == 100) {
long region = CreateRegion();
return region==-1 ? 0 : 1;
}

// create tool
if (infoCore == 1) {
long tool = CreateTool();
return tool==-1 ? 0 : 1;
}

// create independent
if (infoCore == 2) {
long independent = CreateIndependent();
return independent==-1 ? 0 : 1;
}

// create dependent
if (infoCore == 3) {
long dependent = CreateIndependent(false);
return dependent==-1 ? 0 : 1;
}

// create band
long band = CreateBand();
return band==-1 ? 0 : 1;
}
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Map Parameters between Different Versions

Parameters mapping function is used when the UDP version changes or some new parameters are added into a UDP.

You should implement the `MapParametersDefinitionVersions()` function. It calls two other functions `IniParameters()` and `MapParameters()`, which is in the UDPBase class.

```
extern "C" DLLEXPORT
int MapParametersDefinitionVersions(double** newParamValues, char*
oldVersion,
int oldNumberOfParams, struct UDPPrimitiveParameterDefinition*
oldParamDefinition)
{
obj.IniParameters(*newParamValues, numOfParameters);
obj.MapParameters(oldParamDefinition, oldNumberOfParams);// map para
by keywords

int verN = getVerNum(oldVersion);
if (verN < 200) {
obj.SetDefaultParameter(*newParamValues, "HalfAxial", 0);
}

return 1;
}
```

The following function is the `MapParameters()` function, which has already been implemented in the UDPBase class.

```
// map parameter values of old version to new version based on
keywords,
// which is written in UDPBase class and does not need to be
implemented by user
void CUDPBase::MapParameters(UDPPrimitiveParameterDefinition* params,
int oldNum)
{
if (!primParams) return;
int i, j, j0=0;
for (i=0; i<numOfParams; i++) {
oriParams[i] = primParams[i].defaultValue;
if (j0<oldNum && strcmp(primParams[i].name, params[j0].name)==0) {
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
oriParams[i] = params[j0++].defaultValue;
continue;
}
for (j=0; j<oldNum; j++) {
if (strcmp(primParams[i].name, params[j].name) == 0) {
oriParams[i] = params[j].defaultValue;
j0 = j+1;
break;;
}
}
}
}
```

Source Code Files

This section provides examples for the following:

- UDPBase.h
- UDPBase.cpp
- Band.h
- Band.cpp

UDPBase.h

```
// File: UDPBase.h

#if !defined (UDP_BASE_H)
#define UDP_BASE_H

#include <math.h>
#include <string.h>
#include <stdio.h>

#include "../Headers/UserDefinedPrimitiveDLLInclude.h"

//**************************************
// Basic functions
//**************************************
// get version number * 100
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
int getVerNum(char *ver);

// get the distance of two points
double getDistance(UDPPosition p0, UDPPosition p1);

// filter segments with length < minD from a polyline
void purifyPolyline(UDPPolylineDefinition& polylineDefinition, double
minD);

//*************************************
// Base class for any UDP
//*************************************
class CUDPBase
{
protected:
double minD;

int numOfParams;
double *oriParams;
double *paramValues;
bool checkOK; // 1: check OK; 0: check failed

UDPFunctionLib* functionLib;
void* callbackData;

protected:
bool GetValue(int i, char *s, double& v);
bool PutValue(int i, char *s, double& v);
bool GetValue(int i, char *s, int& v);
bool PutValue(int i, char *s, int& v);
bool GetValue(int i, char *s, bool& v);
bool PutValue(int i, char *s, bool& v);

virtual void GetParameters() = 0;
virtual void PutParameters() = 0;
virtual char *ValidateParameters() = 0; // return NULL if initialize
OK
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
void SetFunctionLib(UDPFunctionLib* functionLib0, void*
callbackData0);

// display messages
void DisplayInfo(char *msg=0);

// create lines
long CreateLine(UDPPosition startPos, UDPPosition endPos);
long CreatePolyLine(UDPPosition *pos, int numPos);
long CreateZAxisPath(double z0, double z1);

// create sheets
long CreateSheet(UDPPosition *pos, int numPos, int isSheet=1);

// create solids
long SweepAlongAxis(long obj, UDPCoordinateSystemAxis axis, double
v0, double v1);
long SweepAroundAxis(long obj, UDPCoordinateSystemAxis axis, double
angle);

// edit objects
long RotateAroundZVector(long obj, double angle, double x0=0, double
y0=0);
long DuplicateAroundAxis(long obj, UDPCoordinateSystemAxis axis, int
num,
double angle=0);
long AddObject(long objBase, long objAdd);
long SubObject(long objBase, long objSub);

public:
CUDPBase();
~CUDPBase() {delete[] paramValues;}

void IniParameters(double* paramValues0, int num);
void SetParameterPointer(double* paramValues0) {oriParams =
paramValues0;}
void MapParameters(UDPPrimitiveParameterDefinition* params, int
oldNum);
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
void SetDefaultParameter(double *returnParam, char *key, double
defVal);
int AreParametersValid(char ** error);

virtual long CreatePrimitive(UDPFunctionLib* functionLib, void*
callbackData) = 0;
};

#endif //if !defined (UDP_BASE_H)
```

UDPBase.cpp

```
#include <math.h>
#include <stdlib.h>

#include "../Headers/UDPBase.h"

// used to get parameter names in DisplayInfo() function
extern UDPPrimitiveParameterDefinition primParams[];

const double PI = 3.1415926535897932;

//**************************************
// Basic functions
//**************************************
int getVerNum(char *ver)
{
if (!ver) return 0;
double verD = 0;
sscanf(ver, "%lf", &verD);
return int(verD*100 + 0.5);
}

double getDistance(UDPPosition p0, UDPPosition p1)
{
double dx = p0.x - p1.x;
double dy = p0.y - p1.y;
double dz = p0.z - p1.z;
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
return sqrt(dx*dx + dy*dy + dz*dz);
}

// filter segments with length < minD from a polyline
void purifyPolyline(UDPPolylineDefinition& polylineDefinition, double
minD)
{
int noOfSegments = polylineDefinition.noOfSegments;
int noOfPoints = polylineDefinition.noOfPoints;
UDPPosition* pos = polylineDefinition.arrayOfPosition;
UDPPolylineSegmentDefinition* edge =
polylineDefinition.arrayOfSegmentDefinition;

int i0=0, j0=0, i, j;
for (j=0; j<noOfSegments; j++) {
int i1 = edge[j].segmentStartIndex;
int i2 = i1 + 1;
if (edge[j].segmentType == kUDPArcSegment) i2++;
double length = getDistance(pos[i1], pos[i2 % noOfPoints]);
if (length <= minD) continue;
if (j0 == j) {j0++; i0=i2; continue;}

edge[j0].segmentStartIndex = i0;
edge[j0++].segmentType = edge[j].segmentType;
for (i=i1+1; i<=i2; i++) pos[++i0] = pos[i % noOfPoints];
}
polylineDefinition.noOfPoints = i0;
polylineDefinition.noOfSegments = j0;
}

//***********************************************
// Member function definition for base class
//***********************************************
CUDPBase::CUDPBase() {
minD = 0;
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
paramValues = 0;
checkOK = 0;

functionLib = 0;
callbackData = 0;
}

bool CUDPBase::GetValue(int i, char *s, double& v)
{
if (strcmp(primParams[i].name, s) == 0) {
v = paramValues[i];
return 1;
}
return 0;
}

bool CUDPBase::PutValue(int i, char *s, double& v)
{
if (strcmp(primParams[i].name, s) == 0) {
paramValues[i] = v;
return 1;
}
return 0;
}

bool CUDPBase::GetValue(int i, char *s, int& v)
{
if (strcmp(primParams[i].name, s) == 0) {
v = (int)paramValues[i];
return 1;
}
return 0;
}

bool CUDPBase::PutValue(int i, char *s, int& v)
{
if (strcmp(primParams[i].name, s) == 0) {
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
paramValues[i] = v;
return 1;
}
return 0;
}

bool CUDPBase::GetValue(int i, char *s, bool& v)
{
if (strcmp(primParams[i].name, s) == 0) {
v = paramValues[i] > 0;
return 1;
}
return 0;
}

bool CUDPBase::PutValue(int i, char *s, bool& v)
{
if (strcmp(primParams[i].name, s) == 0) {
paramValues[i] = v ? 1.0 : 0;
return 1;
}
return 0;
}

// set oriParams pointer, and make a copy of data to paramValues
void CUDPBase::IniParameters(double* paramValues0, int num)
{
oriParams = paramValues0;
numOfParams = num;
delete[] paramValues;
paramValues = new double [num];
for (int i=0; i<num; i++) paramValues[i] = oriParams[i];
}

// map parameter values of old version to new version based on
keywords
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
void CUDPBase::MapParameters(UDPPrimitiveParameterDefinition* params,
int oldNum)
{
if (!primParams) return;
int i, j, j0=0;
for (i=0; i<numOfParams; i++) {
oriParams[i] = primParams[i].defaultValue;
if (j0<oldNum && strcmp(primParams[i].name, params[j0].name)==0) {
oriParams[i] = params[j0++].defaultValue;
continue;
}
for (j=0; j<oldNum; j++) {
if (strcmp(primParams[i].name, params[j].name) == 0) {
oriParams[i] = params[j].defaultValue;
j0 = j+1;
break;;
}
}
}
}

// set default values based the keyword of *key
void CUDPBase::SetDefaultParameter(double *returnParam, char *key,
double defVal)
{
for (int i=0; i<numOfParams; i++) {
if (strcmp(primParams[i].name, key) == 0) {
returnParam[i] = defVal;
return;
}
}
}

int CUDPBase::AreParametersValid(char ** error)
{
GetParameters();
checkOK = 0;
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
char *err = ValidateParameters();
if (err) {
*error = err;
return 0;
}
PutParameters();
checkOK = 1;

return 1;
}

void CUDPBase::SetFunctionLib(UDPFunctionLib* functionLib0, void*
callbackData0)
{
functionLib = functionLib0;
callbackData = callbackData0;
}

void CUDPBase::DisplayInfo(char *msg0)
{
if (msg0) {
functionLib->addMessage(kUDPInfoMessage, msg0, callbackData);
return;
}

char msg[BUFSIZ];
for (int i=0; i<numOfParams; i++) {
double err = primParams[i].unitType==kUDPLengthUnit ? minD : 1e-6;
if (fabs(paramValues[i] - oriParams[i]) <= err) continue;
sprintf(msg, "%s is modified from %g to %g",
primParams[i].name, oriParams[i], paramValues[i]);
functionLib->addMessage(kUDPInfoMessage, msg, callbackData);
}
}

// create a line from startPos to endPos
long CUDPBase::CreateLine(UDPPosition startPos, UDPPosition endPos)
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
{
UDPPosition posArray[2] = {startPos, endPos};
UDPPolylineSegmentDefinition SegArray = {kUDPLineSegment, 0};
UDPPolylineDefinition lineDef = {2, posArray, 1, &SegArray, 0, 0};
return functionLib->createPolyline(&lineDef, callbackData);
}

// create a polyline
long CUDPBase::CreatePolyLine(UDPPosition *pos, int numPos)
{
if (numPos <= 1) return -1;

long IDs[2] = {-1, -1};
for (int i=0; i<numPos-1; i++) {
IDs[1] = CreateLine(pos[i], pos[i+1]);
IDs[0] = AddObject(IDs[0], IDs[1]);
}

return IDs[0];
}

// Create a path on z-axis symmetrically with given length for
sweeping along a path
long CUDPBase::CreateZAxisPath(double z0, double z1)
{
UDPPosition startPos = {0, 0, z0};
UDPPosition endPos = {0, 0, z1};
return CreateLine(startPos, endPos);
}

// create a cover sheet of straight line edges
long CUDPBase::CreateSheet(UDPPosition *pos, int numPos, int isSheet)
{
if (isSheet == 0) return CreatePolyLine(pos, numPos);

int numSeg = isSheet==1 ? numPos : numPos-1;
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
UDPPolylineSegmentDefinition *seg = new UDPPolylineSegmentDefinition
[numSeg];
for (int i=0; i<numSeg; i++) {
seg[i].segmentType = kUDPLineSegment;
seg[i].segmentStartIndex = i;
}

UDPPolylineDefinition profile = {numPos, pos, numSeg, seg, isSheet,
isSheet};
purifyPolyline(profile, minD);
if (profile.noOfPoints < 3) return -1;
long sheet = functionLib->createPolyline(&profile, callbackData);
delete[] seg;
return sheet;
}

// sweep along the specified axis within (v0, v1)
long CUDPBase::SweepAlongAxis(long obj, UDPCoordinateSystemAxis axis,
double v0, double v1)
{
if (obj == -1) return obj;
if (functionLib==0 || callbackData==0) return obj;
if (v0 == v1) return obj;

UDPPosition startPos = {0, 0, 0};
UDPPosition endPos = {0, 0, 0};
if (axis == kUDPXAxis) {
startPos.x = v0;
endPos.x = v1;
}
if (axis == kUDPYAxis) {
startPos.y = v0;
endPos.y = v1;
}
if (axis == kUDPZAxis) {
startPos.z = v0;
endPos.z = v1;
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
}

UDPSweepOptions sweepOptions = {kUDPRoundDraft, 0.0, 0.0};
long sweepPath = CreateLine(startPos, endPos);
if (sweepPath == -1) return obj;
functionLib->sweepAlongPath(obj, sweepPath, &sweepOptions,
callbackData);

return obj;
}

// sweep around the specified axis for ang degrees without twist
long CUDPBase::SweepAroundAxis(long obj, UDPCoordinateSystemAxis
axis, double angle)
{
if (obj == -1) return obj;
if (functionLib==0 || callbackData==0) return obj;
if (angle == 0) return obj;

UDPSweepOptions sweepOptions = {kUDPRoundDraft, 0.0, 0.0};
functionLib->sweepAroundAxis(obj, axis, angle, &sweepOptions,
callbackData);

return obj;
}

// rotate around a vector through (x0, y0) in parallel with Z axis
long CUDPBase::RotateAroundZVector(long obj, double angle, double x0,
double y0)
{
if (obj == -1) return obj;
angle -= floor(angle/360) * 360;
if (angle == 0) return obj;

UDPVector vector = {-x0, -y0, 0};
functionLib->translate(obj, &vector, callbackData);
functionLib->rotate(obj, kUDPZAxis, angle, callbackData);
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
UDPVector vector2 = {x0, y0, 0};
functionLib->translate(obj, &vector2, callbackData);
return obj;
}

// duplicate an object around given axis and return the united part
long CUDPBase::DuplicateAroundAxis(long obj, UDPCoordinateSystemAxis
axis,
int num, double angle)
{
if (obj == -1) return obj;
if (num == 1) return obj;

long *IDs = new long [num];
if (angle==0) angle = 360.0 / num;
long bSuccess = functionLib->duplicateAroundAxis(obj, axis, angle,
num, IDs,
callbackData);

long id = IDs[0];
if (bSuccess == 1) {
IDs[num - 1] = obj;
bSuccess = functionLib->unite(IDs, num, callbackData);
}
delete[] IDs;

return bSuccess==0 ? obj : id;
}

// add objAdd to objBase and unite them together
long CUDPBase::AddObject(long objBase, long objAdd)
{
if (objAdd < 0) return objBase;
if (objBase < 0) return objAdd;

long IDs[] = {objBase, objAdd};
functionLib->unite(IDs, 2, callbackData);
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
return objBase;
}

// subject objSub from objBase
long CUDPBase::SubObject(long objBase, long objSub)
{
if (objSub < 0) return objBase;
if (objBase < 0) return -1;

long bSuccess = functionLib->subtract(&objBase, 1, &objSub, 1,
callbackData);
if (bSuccess == 0) return -1;

return objBase;
}
```

Band.h

```
// File: Cores.h

#if !defined (CORES_H)
#define CORES_H

#include "../Headers/UDPBase.h"

//**************************************
// Basic class for Band
//**************************************
class CBand : public CUDPBase
{
protected:
double diaGap; // diameter at the gap side
double diaYoke; // diameter at the yoke side
double length; // core length or model depth: length = 0 for 2D

int numSegs; // number of segments of whole circular band
int numFras; // number of circumferential fractions
int infoHalf; // 1 for half model in axial direction
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
int infoCore; // information for drawing different parts of the
object
int infoOut; // information for an inner or outer core

double rRegion; // region radius
double segAng; // angle per segment

protected:
virtual void GetParameters();
virtual void PutParameters();
virtual char *ValidateParameters();

long SweepAlongZAxis(long obj);

long CreateBand();
long CreateTool();
long CreateMaster(bool isMaster=true);
long CreateRegion(double r0=0, int fras=0);

public:
virtual long CreatePrimitive(UDPFunctionLib* functionLib, void*
callbackData);
};

#endif //if !defined (CORES_H)
```

Band.cpp

```
#include "../Headers/Band.h"

const double PI = 3.1415926535897932;

static struct UDPPrimitiveTypeInfo primitiveInfo =
{
"Band",
"Create a band, shaft or outer region",
"ANSYS, Incorporated",
"06-01-2012",
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
"2.0"
};

extern "C" DLLEXPORT
struct UDPPrimitiveTypeInfo* GetPrimitiveTypeInfo()
{
return &primitiveInfo;
}

struct UDPPrimitiveParameterDefinition primParams[] =
{
{"DiaGap", "Band diameter in gap center, DiaGap<DiaYoke for outer
band",
kUDPLengthUnit, 101},
{"DiaYoke", "Band diameter on yoke side, DiaYoke<DiaGap for inner
band",
kUDPLengthUnit, 40},
{"Length", "Band length", kUDPLengthUnit, 200},

{"SegAngle", "Deviation angle for band (0.1~5 degrees).",
kUDPAngleUnit, 3},
{"Fractions", "Number of circumferential fractions, 1 for circular
region.",
kUDPNoUnit, 1},
{"HalfAxial", "0: full model; 1: half model in axial direction.",
kUDPNoUnit, 0},
{"InfoCore", "0: band; 1: tool; 2: independent; 3: dependent; 100:
region.", kUDPNoUnit, 0}
};

static int numOfParameters = sizeof(primParams)/sizeof(primParams
[0]);

extern "C" DLLEXPORT
int GetPrimitiveParametersDefinition(struct
UDPPrimitiveParameterDefinition**
paramDefinition)
{
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
*paramDefinition = primParams;
return numOfParameters;
}

extern "C" DLLEXPORT
char* GetLengthParameterUnits()
{
return "mm";
}

static CBand obj;

// In case of error this function should return 0
extern "C" DLLEXPORT
int AreParameterValuesValid(char ** error, double* paramValues)
{
obj.IniParameters(paramValues, numOfParameters);
return obj.AreParametersValid(error);
}

extern "C" DLLEXPORT
int MapParametersDefinitionVersions(double** newParamValues, char*
oldVersion,
int oldNumberOfParams, struct UDPPrimitiveParameterDefinition*
oldParamDefinition)
{
obj.IniParameters(*newParamValues, numOfParameters);
obj.MapParameters(oldParamDefinition, oldNumberOfParams);// map para
by keywords

int verN = getVerNum(oldVersion);
if (verN < 200) {
obj.SetDefaultParameter(*newParamValues, "HalfAxial", 0);
}

return 1;
}
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
extern "C" DLLEXPORT
long CreatePrimitive(struct UDPFunctionLib* functionLib, void*
callbackData, double* paramValues)
{
obj.SetParameterPointer(paramValues);
return obj.CreatePrimitive(functionLib, callbackData);
}

//***************************************************
// Member function definition for class CBand
//***************************************************
void CBand::GetParameters()
{
diaGap = diaYoke = 0;
length = 0;
infoCore = 0;
infoOut = -1;

segAng = 3;
numFras = 1;
infoHalf = 0;

for (int i=0; i<numOfParams; i++) {
if (GetValue(i, "DiaGap", diaGap)) continue;
if (GetValue(i, "DiaYoke", diaYoke)) continue;
if (GetValue(i, "Length", length)) continue;

if (GetValue(i, "SegAngle", segAng)) continue;
if (GetValue(i, "Fractions", numFras)) continue;
if (GetValue(i, "HalfAxial", infoHalf)) continue;
if (GetValue(i, "InfoCore", infoCore)) continue;
}
rRegion = diaYoke / 2;
}

void CBand::PutParameters()
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
{
for (int i=0; i<numOfParams; i++) {
if (PutValue(i, "DiaGap", diaGap)) continue;
if (PutValue(i, "DiaYoke", diaYoke)) continue;
if (PutValue(i, "Length", length)) continue;

if (PutValue(i, "SegAngle", segAng)) continue;
if (PutValue(i, "Fractions", numFras)) continue;
if (PutValue(i, "InfoCore", infoCore)) continue;
}
}

char* CBand::ValidateParameters()
{
if (!paramValues) return 0;

if (diaYoke < 0) diaYoke = 0;
if (diaGap <= 0) return "DiaGap must be larger than 0.";
if (length < 0) length = 0; // length==0 for 2D geometry
infoOut = diaYoke>diaGap ? 1 : -1;

if (numFras < 1) numFras = 1;
if (segAng > 5) segAng = 5;
if (segAng <= 0) segAng = 0;
else if (segAng < 0.1) segAng = 0.1;
if (segAng > 0) {
numSegs = (int)(360/segAng + 0.5);
int remain = numSegs % numFras;
if (remain > 0) numSegs += numFras - remain;
segAng = 360.0 / numSegs;
}
else numSegs = 0;

if (infoCore < 100) {
if (infoCore < 0) infoCore = 0;
if (infoCore > 3) infoCore = 3;
}
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
if (infoCore >= 100) {
if (diaYoke <= 0) return "DiaYoke must be larger than 0.";
}
else if (infoCore >= 1) { // for tool; independent & dependent
if (diaYoke <= diaGap) return "DiaYoke must be larger than DiaGap.";
}

return 0;
}

long CBand::SweepAlongZAxis(long obj)
{
if (obj == -1) return obj;
if (functionLib==0 || callbackData==0) return obj;
if (length == 0) return obj;

double z0 = infoHalf==0 ? -length/2 : 0;
UDPSweepOptions sweepOptions = {kUDPRoundDraft, 0.0, 0.0};
long sweepPath = CreateZAxisPath(z0, length/2);
if (sweepPath == -1) return obj;
functionLib->sweepAlongPath(obj, sweepPath, &sweepOptions,
callbackData);

return obj;
}

long CBand::CreateBand()
{
long band = -1;
double r0 = diaGap / 2;
if (numSegs == 0) band = CreateRegion(r0);
else {
double angle = 2*PI / numSegs;
int n = numSegs / numFras;
double x0 = r0 * cos(angle);
double y0 = r0 * sin(angle);
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
UDPPosition pos[] = { 0, 0, 0,
r0, 0, 0,
x0, y0, 0};
band = CreateSheet(pos, 3);
band = SweepAlongZAxis(band);
band = DuplicateAroundAxis(band, kUDPZAxis, n, angle*180/PI);
}
if (diaYoke <= diaGap) return band;

long region = CreateRegion();
return SubObject(region, band);
}

long CBand::CreateTool()
{
UDPPosition p0 = {0, 0, -length/2};
double r0 = 0.55 * diaYoke;
long tool = functionLib->createCylinder(kUDPZAxis, &p0, r0, length,
callbackData);
long region = CreateRegion();
return SubObject(tool, region);
}

// Create independent or dependent
long CBand::CreateIndependent(bool isIndependent)
{
double r0 = 0.5 * diaYoke;
UDPPosition startPos = {0, 0, 0};
UDPPosition endPos = {r0, 0, 0};
long independent = CreateLine(startPos, endPos);
if (!isIndependent) RotateAroundZVector(independent, 360.0/numFras);
return SweepAlongZAxis(independent);
}

long CBand::CreateRegion(double r0, int fras)
{
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
if (r0 == 0) r0 = diaYoke/2;
if (fras == 0) fras = numFras;
UDPPosition startPos = {0, 0, 0};
UDPPosition endPos = {r0, 0, 0};
long region = CreateLine(startPos, endPos);
region = SweepAlongZAxis(region);
return SweepAroundAxis(region, kUDPZAxis, 360.0/fras);
}

long CBand::CreatePrimitive(UDPFunctionLib* functionLib, void*
callbackData)
{
if (!checkOK) return 0;

// display information of modified parameters
SetFunctionLib(functionLib, callbackData);
DisplayInfo();

// create region
if (infoCore == 100) {
long region = CreateRegion();
return region==-1 ? 0 : 1;
}

// create tool
if (infoCore == 1) {
long tool = CreateTool();
return tool==-1 ? 0 : 1;
}

// create independent
if (infoCore == 2) {
long independent = CreateIndependent();
return independent==-1 ? 0 : 1;
}

// create dependent
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
if (infoCore == 3) {
long dependent = CreateIndependent(false);
return dependent==-1 ? 0 : 1;
}

// create band
long band = CreateBand();
return band==-1 ? 0 : 1;
}
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10 - Assigning Boundaries and Excitations for 3D Designs

For every project, you need to assign boundaries and excitations. For 3D designs, you can use the **Maxwell 3D** menu or the project tree to do the following:

- Define boundary conditions that control how the electric or magnetic field behaves at object faces, planes of symmetry and periodicity, and edges of the problem region.
- Define excitations of voltage, charge, coil, and current.
- Identify conductors in which eddy currents are induced.
- For AC magnetic and magnetic transient projects, set up a winding and an external circuit connection.

Each field solver requires you to specify excitations of electric or magnetic fields and references for computing these fields.

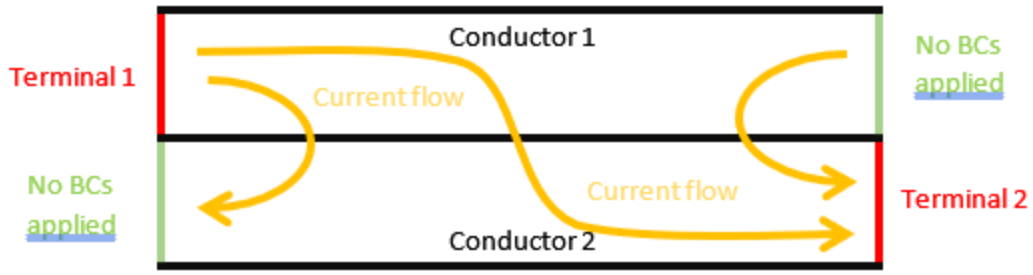
- Magnetostatic Boundaries and Excitations
- AC Magnetic (Eddy Current) T-Omega Boundaries and Excitations
- AC Magnetic (Eddy Current) A-Phi Boundaries and Excitations
- Transient Boundaries and Excitations
- Transient A-Phi Formulation Boundaries and Excitations
- Electrostatic Boundaries and Excitations
- DC Conduction Boundaries and Excitations
- Electric Transient Boundaries and Excitations
- AC Conduction Boundaries and Excitations

You must specify at least one of the boundary conditions or excitations listed in these sections, so that the simulator can compute accurate values for fields and parameters.

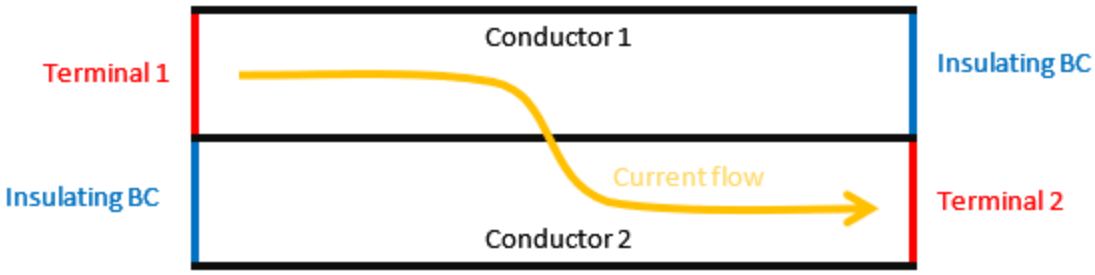
Depending on the problem to be solved, you must also define the current flowing through the conducting domain faces touching the domain boundaries (ports) by assigning them excitations or setting their current to zero by defining insulating boundary conditions. If no constraint is applied to that boundary, the current will flow freely through these ports, which may lead to undesired but still physical results.

Case 1: Conducting faces without boundary conditions (BCs) applied (green lines). The current flows freely through these faces.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Case 2: Insulating boundary conditions applied (blue lines). No current is allowed to flow through these faces.



Selecting Objects and Faces in the Geometry

Before creating a boundary or excitation, you must specify its location by selecting the object or surface on which to assign the boundary condition or source excitation.

Defining Boundary Conditions

To assign a boundary condition:

1. Select the section of the geometry on which you want to apply the boundary condition.
2. Click **Maxwell 3D** > **Boundaries** > **Assign**, or right-click **Boundaries** in the Project Manager tree and click **Assign**.
3. Select the desired boundary type from the submenu. The specific boundary types you can define depend on the solver you have chosen for your project.

    - Magnetostatic Boundary Conditions
    - Electrostatic Boundary Conditions
    - DC Conduction (Electric) Boundary Conditions
    - AC Magnetic T-Omega Boundary Conditions

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- AC Magnetic A-Phi Boundary Conditions
- Transient Boundary Conditions
- Transient A-Phi Formulation Boundary Conditions

4. Enter the name for the boundary or accept the default.
5. Depending on the boundary type, you may need to define additional parameters.
6. Click **OK** to create the new boundary.

The new boundary is added to the **Boundaries** list in the Project Manager tree.

Defining Excitations

The specific excitations you can define depend on the solver you have chosen for your project.

- Magnetostatic Excitations
- Electrostatic (Electric) Excitations
- AC Conduction (Electric) Excitations
- DC Conduction (Electric) Excitations
- AC Magnetic T-Omega Excitations
- AC Magnetic A-Phi Excitations
- Transient Excitations
- Transient A-Phi Formulation Excitations
- Electric Transient Excitation

To assign an excitation:

1. Select the section of the geometry on which you want to apply the excitation.
2. Click **Maxwell 3D** > **Excitations** > **Assign**, or right-click **Excitations** in the Project Manager tree, and click **Assign**.

   A submenu appears.

3. Select an excitation type from the submenu. The choices vary depending on the solver type you have chosen. Choices include
   - Voltage
   - Voltage Drop
   - Charge
   - Volume Charge Density
   - Current Density
   - Current
   - Permanent Magnet Field
   - Sink
4. Enter the name for the excitation type or accept the default.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Enter the desired parameter values, units, and other settings for the excitation in the dialog box. You can also enter a mathematical function to determine values for voltage, current density, charge density, and other parameters. The Permanent Magnet Field excitation also requires you set up a link to a source project.

   > **Note:** When entering current, current density terminals, or coil terminals, the arrow associated with current in the selected object shows the direction as if a positive value is entered for the current. If a negative value is entered the actual current flow direction is opposite to what the arrow shows. Click **Swap Direction** if you want to reverse the direction of the arrow.

6. Click **OK** to create the new excitation.

The new excitation is added to the excitation list in the Project Manager tree.

Viewing Boundaries and Excitations

To view a list of all boundaries or excitations set for the project:

1. In the Project Manager tree, right-click **Boundaries** or **Excitations**.

   A shortcut menu appears.

2. Select **List** from the shortcut menu.

   The **Design List** window appears, with the tab page visible for the item you selected. This window contains tabs for **Model**, **Boundaries**, **Excitations**, **Parameters**, **Mesh Operations**, and **Analysis Setup**.

3. To view the properties for an item in the list, select the item, and click **Properties**.
4. To delete an item from the list, select the item, and click **Delete**.
5. When you are finished in the **Design List** window, click **Done** to close it.

Setting the Visibility for 3D Boundaries and Excitations

To show or hide a boundary:

1. Click **View** > **Active View Visibility**.

   The **Active View Visibility** dialog box appears.

2. Click the **Boundaries** tab.
3. Select or clear the **Visibility** check box for each boundary listed.
4. Click **Done**.

To show or hide a excitation:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **View** > **Active View Visibility**.

   The **Active View Visibility** dialog box appears.

2. Click the **Excitations** tab.
3. Select or clear the **Visibility** check box for each excitation listed.
4. Click **Done**.

Functional Boundaries and Excitations

Functional boundaries and excitations can be used to do the following:

- Define the value of a boundary or excitation quantity (such as the voltage, magnetic field, or current density) using a mathematical relationship – such as one relating its value to that of another quantity.
- Define the value of a boundary or excitation as a function of position.
  - Current Density Excitation (Magnetostatic or AC Magnetic solver type)
  - Voltage Excitation (Electric solver types)
  - Volume Charge Density Excitation (Electrostatic or Electrostatic + DC solver types)
- If the parametric analysis capability was purchased, identify which boundary or excitation quantities are to be varied during a parametric sweep. These variables are always set to constant values in the parametric analysis.

Assigning a Thin Layer Boundary

Maxwell 3D supports Thin Layer boundary types for the following solvers:

- Thin Layer Boundary for a 3D AC Conduction Solver

  This boundary condition can be used to model thin domains, for example: the contact resistance between two conductive domains, the capacitive effect of thin insulating materials or thin lossy domains in electronic boards without needing to explicitly model (and mesh) a 3D object having a finite thickness.

- Thin Layer Boundary for a 3D DC Conduction Solver

  This boundary condition can be used to model thin domains, for example, the contact resistance between two conductive domains without needing to explicitly model (and mesh) a 3D object having a finite thickness.

- Thin Layer Boundary for a 3D Magnetostatic Solver

  This boundary condition can be used to model narrow air gaps, such as those in power transformers, inductors, or any other electric device forming a magnetic circuit with an air gap without needing to explicitly model (and mesh) this gap.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Thin Layer Boundary for a 3D AC Magnetic Solver

  This boundary creates a field discontinuity between the two sides of the selected surface based on its thickness and the given electric and magnetic material proprieties. When the air gap option is selected, the conductivity is assumed to be zero and the relative magnetic permeability is assumed to be equal to one.

- Thin Layer Boundary for a 3D Transient Solver

  This boundary condition can be used to model narrow air gaps, such as those in power transformers, inductors, or any other electric device forming a magnetic circuit with an air gap without needing to explicitly model (and mesh) this gap.

Setting Eddy Effects and Displacement Current in 3D

(AC Magnetic and/or Transient Solvers)

For both AC magnetic and transient 3D solvers, you may need to specify the following additional setting:

- The behavior of eddy currents and the AC magnetic field in conductors. When you activate the **Eddy Effect** setting, the solver computes the induced eddy currents.

For 3D AC magnetic solvers only, you may need to specify the following additional setting:

- The **Displacement Current** on the objects in the model.

Typically, background objects are excluded from eddy and displacement current settings.

To set eddy effects:

1. Select the elements of the geometry on which you want to apply the excitation.
2. Click **Maxwell** > **Excitations** > **Set Eddy Effects**.

   The **Set Eddy Effect** dialog box appears.

3. For each object in the list, select or clear the check boxes for **Eddy Effect** and **Displacement Current** (3D AC Magnetic T-Omega solver). When selecting multiple entries, the changes are synchronized for all selected rows.

   > **Note:** Multi-select is supported by holding the **Ctrl** or **Shift** key while clicking.

4. Click the **Select Object By Name** button to enter the name of a specific object for selection.
5. Optionally, click the **Use suggested values** button to set the **Eddy Effects** for all

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

elements recommended by Maxwell 3D.

6. Click **OK**.

The **Deselect All** button will deselect all selected objects in the list.

Displacement Current for the 3D AC Magnetic A-Phi

For the 3D AC Magnetic A-Phi solver, the displacement current effect is set for the whole domain rather than for each object. As a consequence, the number of unknowns will increase and the simulation may require more memory and more simulation time.

To enable the displacement current effect:

1. From the Project Manager tree, right-click **Excitations** and select **Set Eddy Effects**.
2. Enable the **Displacement Current** check box.

Setting Core Loss for Transient and AC Magnetic Solvers

For a Transient or an AC Magnetic (Eddy Current) solver design, you may include core loss calculations in the simulation on any object that has a corresponding **core loss definition** (with core loss coefficient settings) in the material library.

To set core loss:

1. Click **Maxwell** > **Excitations** > **Set Core Loss**.

   The **Set Core Loss** dialog box appears.

   You can also open the **Set Core Loss** dialog box by right-clicking **Excitations** in the Project Manager, then selecting **Set Core Loss** on the context menu; or by right-clicking in the Modeler window and selecting **Assign Excitation** > **Set Core Loss**.

2. For each **Object** in the list, select or clear the check box for **Core Loss Setting** (if a corresponding core loss definition has been set). The **Defined in Material** column indicates whether the material assigned to each object contains a core loss definition. When selecting multiple entries, the changes are synchronized for all selected rows. Multi-select is supported by holding the **Ctrl** or **Shift** key while clicking.
3. Click the **Select By Name** button to enter the name of a specific object for selection. Regular expressions are also supported.
4. Optionally, click the **Deselect All** button to deselect all selected objects in the list.
5. For 3D and 2D transient solvers only, on the **Advanced** tab of the dialog box, you may select the option to **Consider core loss effect on field**. When selected, this option will estimate the core loss and subsequently modify the field solution, causing the resulting fields to be affected by the presence of the core loss. This option provides improved

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

solution accuracy for materials exhibiting high core loss at the expense of increased solution time.

> **Note:**
>
> - If the core loss model “hysteresis model” is chosen, then the “Consider core loss effect on field” checkbox is ignored, as the hysteresis model always models the effects of the hysteresis loss.
> - An iteration is required to consider loss effects on the field. For most really soft materials, the iteration is converged. However, if the loss coefficients kh and ke for magnetic steels, or Cm for power ferrites, are too large, the iteration could be diverged. In such cases, users need to double check the specified values, or not to consider core loss effects.

The default is to exclude this effect. In this case, the field calculation is performed without the core loss and the loss calculation is performed after the field solution is complete.

6. Click **OK**.

Specifying a Custom Core Loss

In addition to the standard core loss excitation detailed in Setting Core Loss for Transient and AC Magnetic Solvers, users can also specify a custom core loss formulation. Using a Python script and a data file, the user defines fields and mesh information that are passed on the solver through the Solve Setup window. The core loss formulation in the Python script is used by Maxwell to evaluate core loss, element by element.

This custom core loss capability extends the available core loss formulation for the Maxwell 2D Magnetic Transient solver and the Maxwell 3D Magnetic Transient solver (T-Omega formulation only). It can be used with Distribute solve technology (DSO) and Large-Scale Distribute solve technology (LS-DSO) on Windows and Linux. Time Domain Decomposition is not supported.

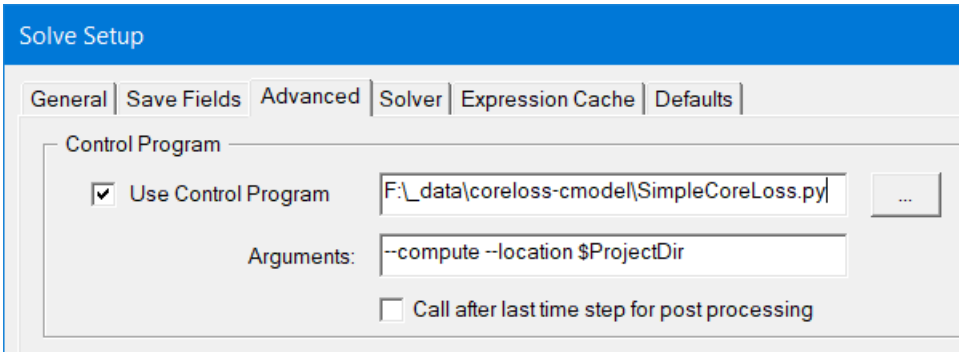
Several specialty formulations are included in the *//Program Files/ANSYS Inc/v252/AnsysEM/Examples/Maxwell/CustomCoreloss* directory.

Required Files

To use a custom core loss excitation, the user must provide two files: the control program Python file and the data file that contains the core loss settings.

The control program (a Python file) must be referenced in the **Advanced** tab of the Solve Setup window:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



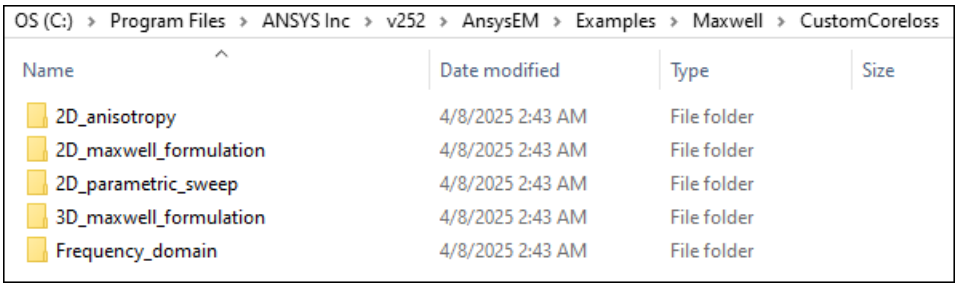
- In the Arguments field, the command `--compute --location $ProjectDir` must be specified (most common use case; see below for all the options for arguments).
- The **Call after last timestep for post processing** check box can be used to delete all temporary files.

The file that contains the core loss settings must be named *coreloss_user.data* and be placed in same directory as the Maxwell project (if `-location $ProjectDir` argument is used). Its format is described below.

When running the Maxwell magnetic transient simulation, the core loss program will be called at each timestep. The core loss values are exported in a text file (*Result_CoreLoss.txt*). In the case of a parametric sweep, the text file name is indexed with the sweep variables (*i.e.*, *Im_100A_Gamma_90deg_Results_CoreLoss.txt*)

Examples and Templates

Ansys Electronic Desktop provides users with examples and templates, which are located in the *\\Ansys Inc\VersionNumber\AnsysEM\Examples\Maxwell\CustomCoreloss* directory.



The following example formulations are available for the users, without having the necessity to know any Python programming skills:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Replication of Maxwell's core loss model for electric steel
- Maxwell's core loss model for electric steel with anisotropy (different values of core loss coefficients for X, Y and Z directions)
- Frequency-domain core loss (core loss coefficients are frequency dependent)

The material properties to enable those formulations are specified in *coreloss_used.data* file, which is included in each example directory.

The following templates are available and can be readily modified:

- 2D_maxwell_formulation replicates the Maxwell model.
- 2D_FrequencyDomain, 2D_FrequencyDomain_varKhKc and 3D_FrequencyDomain_varKhKc illustrate the Frequency domain formulation.
- 2D_parametric_sweep contains a 2D transient model with anisotropy, along with an example of parametric sweep. For the model to function, the user has to provide Khx, Khy, Kcx, Kcy, Kex, Key coefficients for each object.
- 3D_maxwell_formulation contains *ExampleCoreLoss3D.py*, which allows anisotropy. The user has to provide Khx, Khy, Khz, Kcx,Kcy,Kcz,Kex,Key,Kez coefficients for each object.

Defining the coreloss_user.data File

The custom core loss program does not use material properties from Maxwell. The *coreloss_user.data* file contains all the material properties and settings for the core loss simulation. Here is a 2D example for calculating the core loss on the object Stator using Maxwell's core loss model for electric steel:

```
#This provides user input data for the User Defined Core Loss Model
begin_data
    SimType 0
    Ncoord 2
    UseFreqDomain 0
    VarKhKc 0
    SymmMultiplier 8
    ObjectNames Stator
    ObjectIDs 928
    Kh 200.0
    Kc 0.5
    Ke 0.5
    Beta 2.0
    Alpha 0.75
    Stacking 1
    ModelDepth 0.083566
end_data
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The file must start with `begin_data` and ends with `end_data`. Comments can be added, as shown in line 1.

The following keywords can be entered and provide information on the settings:

Keyword	Description	Default
SimType	1 for electric machine, 0 otherwise	0
Ncoord	2 for Maxwell 2D, 3 for Maxwell 3D	2
UseFreqDomain	Enter 1 to enable frequency-domain core loss	1
VarKhKc	Enter 1 to enable variable Kh(Freq), Kc(Freq) coefficients	
SymmetryMultiplier	Enter the Symmetry multiplier present in Maxwell	1
Tau,Tmax	For the frequency-domain core loss model only. Tau is the electric period for the Fourier analysis. Tmax is the max time .	0.02,0.04
Npoles,SimPeriod	If SimType=1, Npoles is the number of poles of the machine, and SimPeriod is the number of electric periods to simulate (). Tau and Tmax are derived from these values.	1
ObjectNames	Names of the objects where the custom core loss is applied	None
ObjectIDs	IDs of the objects listed with ObjectNames; see below for instructions on how to get IDs	None
Kh,Kc,Ke (or Khx,Khy,Khz, Kcx,Kcy,Kcz, Kex,Key,Kez), Alpha, Beta, Stacking, ModelDepth	Core loss coefficients for each object as well as the stacking factor and the ModelDepth (in meters, for 2D only)	None
VarNames	Names of the swept variables when a parametric sweep is defined	None

Note the following:

- To obtain the ID of the object, open the Python console (**Tools** > **Open Command Window**) and type the following commands, in order (this example assumes there is an object named Stator in the project):

```
project=oDesktop.GetActiveProject()
design=project.GetActiveDesign()
editor=design.SetActiveEditor("3D Modeler")
editor.GetObjectIDByName("Stator")
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- In the case of an electric machine, SymType is set to 1. In this case, Tau and Tmax will be automatically calculated for frequency-domain core loss:
  - Tau = 120 /(speed*Npoles) (speed is read by the control program)
  - Tmax = SimPeriod*Tau
- When VarKhKc is activated, additional files must be provided to define frequency-dependent variables. See "Using VarKhKc Keyword: Kh and Kc Coefficient Vary with Frequency" that follows.

Examples of coreloss_user.dat

Ansys Electronic Desktop provides *coreloss_user.data* file samples for the most common use cases in the *\\Examples\Maxwell\CustomCoreloss* directory:

- 2D simulation, similar core loss model to the Maxwell model
- 2D simulation, core loss model with anisotropy
- 2D simulation, core loss model used with a parametric sweep
- 2D simulation, core loss model using the frequency-domain model with *constant* Kh and Kc coefficients
- 2D simulation, coreloss model using frequency domain model with *variable* Kh and Kc coefficients

Using VarKhKc Keyword: Kh and Kc Coefficient Vary with Frequency

When using frequency-domain core loss formulation, it is possible to define frequency-dependent core loss coefficients. To activate this feature, UseFreqDomain and VarKhKc must be set to 1 in the coreloss_user.data file. The frequency dependents are provided in a separate file:

- Each object has a corresponding coefficient file.
- The name of the file must be *KhKc_Coefficients_XYZ.txt*. XYZ is the object ID that is defined in coreloss_user.data file. For instance if the objectID is defined as 487 in the coreloss_user.data file, as shown below, the coefficient file will be named *KhKc_Coefficients_487.txt*.

```
ObjectNames core
ObjectIDs 487
Kh 0.0
Kc 0.5
Ke 0.0
```

- In this case, the Kh and Kc values in coreloss_user.data file are ignored.
- The *KhKc_Coefficients_XYZ.txt* file is generated from measurement data using a script. Contact Ansys support for further explanations.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Writing Your Own Formulations

When writing your own Python formulations, you can use existing examples as starting point or you can start from scratch.

To use an existing example as starting point:

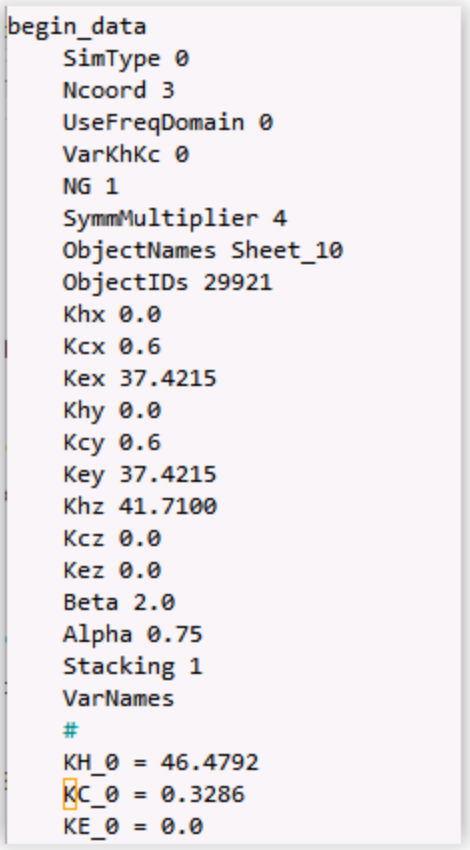
1. From the list of examples available in \\Examples\Maxwell\CustomCoreloss, copy the file and rename it.
2. Provide a Python file to Maxwell that does the following:
   - Imports libraries (import `CoreLoss`, `import` utilities)
   - Writes a class that derives from the base method
   - Defines your own version of the method `compute_coreloss_loss_in_a_2D_element`, or `compute_coreloss_loss_in_a_3D_element`.
   - Calls the `compute_coreloss` method from the main function.

To write your own Python formulation from scratch:

1. Set up the Python environment: You do not need to install Python itself. Instead, you need to ensure that Ansys Electronic Desktop is installed on your computer and that you have the licenses needed to run a Maxwell transient solve.
2. Create your own core loss routine: Use existing examples of `compute_core_loss_in_a_2D_element` or `compute_core_loss_in_a_3D_element` to define imports/inputs and outputs.
3. Add your own parameters/material properties to *coreloss_user.data*. You can add any parameter (integer/floating/list …) in *coreloss_user.data* that is needed for your formulation. In the example below, we have added three new parameters: KH_0, KC_0

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and KE_0. Those parameters will be available in your local core loss definition.



4. Read your parameters in the core loss definition. To use KH_0, KC_0 and KE_0 in your own core loss definition, use the syntax below. Note that the parameters are read as strings:

```
KH_0 = float(self.all_parameters.name_values.get("KH_0",0.0"))
KC_0 = float(self.all_parameters.name_values.get("KC_0",0.0"))
KE_0 = float(self.all_parameters.name_values.get("KE_0",0.0"))
```

5. Send your core loss calculation back to the framework. Once you have defined your own formulation, return the values into the following predefined variables:

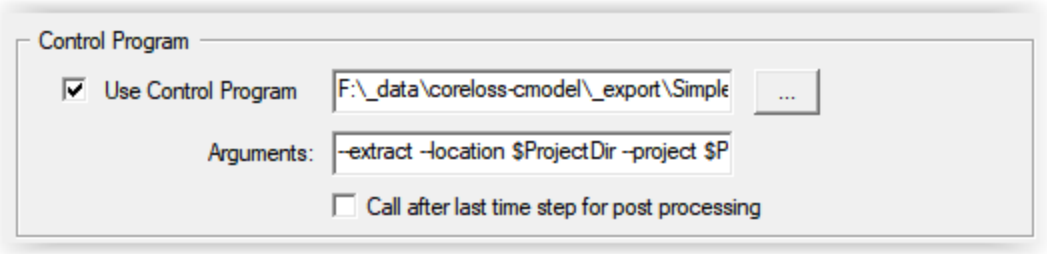
```
corelossCoeffComponents.CHYS = CHYS
corelossCoeffComponents.CEXC = CEXC
corelossCoeffComponents.CEDDY = CEDDY
return True
```

Using Your Control Program with a Maxwell Design

To use your control program with a Maxwell design:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Open the project in Ansys Electronics Desktop.
2. Open the properties of the desired analysis setup.
3. In the **Advanced** tab, for the Control Program Path:
    - Update the **Control Program Path** property in your Ansys Electronics Desktop project file to point to the location of your final Python calculation program.
    - Ensure that the path is correctly specified, including the file name and extension, and that it matches the actual location of your Python program on your system.
4. For the Control Program Arguments:
    - Update the **Control Program Arguments** property in your Ansys Electronics Desktop project file to include the appropriate arguments for your computation process.
    - The argument string can begin with the keyword `extract` instead of `compute`. With `Extract` mode, you can run the custom core loss simulation without having to run the Maxwell simulation. Contact Ansys support for more details.



Frequently Asked Questions

1. What are the key features and functionalities provided by the library?
    - Custom computation in Python: Enables Maxwell's core loss formulation to be replaced or extended using code written in Python.
    - Output file format: Core loss data to be saved in any format, which enables easier processing.
    - Input file format: The *coreloss_user.data* file can be in user specified format (such as json).
2. Do I need to install Python? What version of Python is supported?
    - The custom coreloss module uses the CPython version from the Ansys Electronic Desktop installation. It uses version 3.10.
3. Can you explain how the library integrates with Ansys Electronics Desktop and supports core loss computations?
    - The library enables you to write a control program and provide data to control the core loss computation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Ansys Electronic Desktop runs the control program for each timestep of transient analysis, including at the beginning and at the end.
- During initialization, the library uses your input data file to create a *user.ctl* file as needed by Ansys Electronic Desktop .
- During each timestep, the library reads the field information provided by Ansys Electronic Desktop and calls your code to compute core loss for each element in the specified interesting objects.

4. How does the library handle the specific engineering parameters or calculations relevant to core loss computation?
   - In your input data file, you can specify material properties for each object of interest.
   - In the time domain, it computes the value of magnetic field vectors and their derivatives needed for core loss computation.
5. What is the level of customization and extensibility offered by the library to accommodate my specific requirements?

   You can override many of the methods of the base class and write your own methods.

Permanent Magnet Field Excitations for 2D and 3D Transient and Magnetostatic Solvers

(Maxwell 2D/3D Transient and Magnetostatic Solvers)

Links to demagnetization and magnetization computation *source* designs can be added as permanent magnet field excitations in *target* designs. A permanent magnet field excitation can be either *object-based* (assigned to geometry) or *design-based* (not assigned to geometry). This topic details how to add or edit a permanent magnet field excitation.

Note:

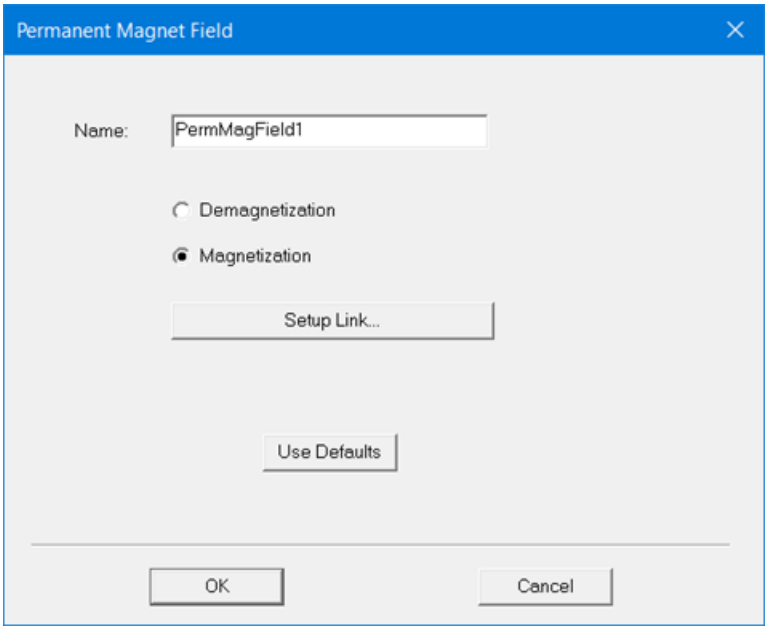
The following procedure assumes that a *source* design, in which either the demagnetization or magnetization operating points computations of the magnet have been modeled, is available for linking.

The *target* design object on which the Permanent Magnet Field is assigned should have a local coordinate system assigned.

- This coordinate system must be right-(or left-)handed, the same as in the *source* design.
- The *target* design object should be in the same quadrant of the new local coordinate system as the matching object in the *source* design.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

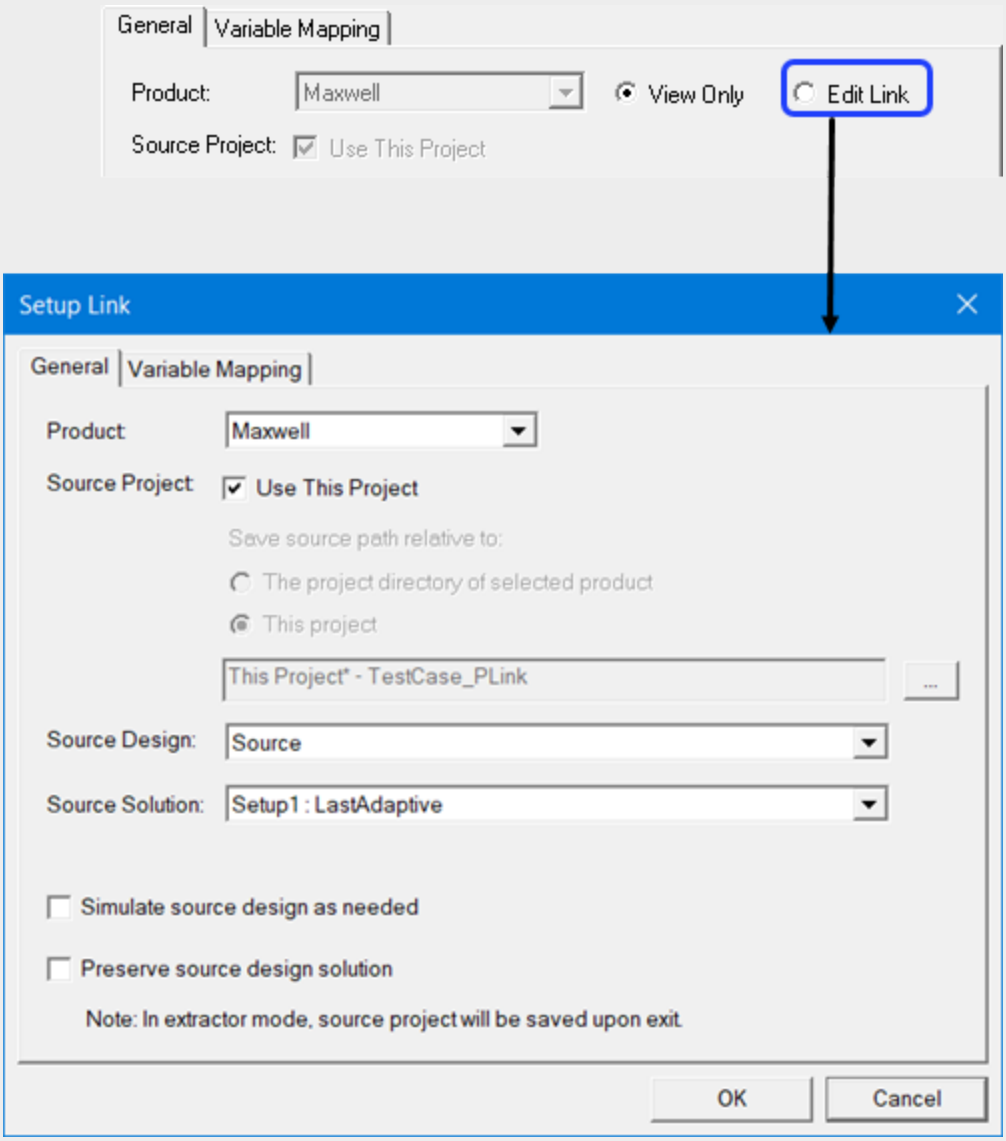
1. Do one of the following:
   - If you are adding a new *design-based* permanent magnet field excitation, select **Maxwell 2D** or **3D** > **Excitations** > **Assign** > **Permanent Magnet Field**. Alternatively, in the Project Manager you can right-click **Excitations** to open its context menu, then select **Assign** > **Permanent Magnet Field**.
   - If you are adding a new *object-based* permanent magnet field excitation, select one or more objects (magnets) that you wish to include in the permanent magnet field excitation, then either select **Maxwell 2D** or **3D** > **Excitations** > **Assign** > **Permanent Magnet Field**, right-click **Excitations** in the Project tree to open its context menu and select **Assign** > **Permanent Magnet Field**.
   - If you are editing an existing permanent magnet field excitation, right-click on the excitation in the project tree and select **Properties** from the context menu.
2. In the **Permanent Magnet Field** dialog box, enter (or edit) the **Name** for the excitation - or accept the default.



3. Depending on how the *source* design which is to be linked has been configured using **Set Demagnetization/Magnetization Computations**, choose either **Demagnetization** or **Magnetization** for the excitation type, then click **Setup Link**.
4. By default, the **Setup Link** dialog box opens on the **General** tab with only the **This project** radio button selected.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: If a link had previously been set up, the **Setup Link** dialog box opens in **View Only** mode with all settings disabled.



To enable settings, you can select **Edit Link**.

5. To select a source project do one of the following:

   - To use the current project as the source, check **Use This Project**. This disables the **Save source path relative to:** radio buttons and the ellipsis **[...]** button and its associated text field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To specify a source project file other than the current project click the ellipsis **[...]** button to open a file browser window. When you have selected the project, click the **Open** button to accept the project file for the setup. You can use the check box to **Open as read only**.

  Use the radio buttons to specify whether to save the source path relative to **The project directory of selected product** or **This project**.

6. When you select a source project file, the **Source Design** and the **Source Solution** fields are filled in with default values, and their drop-down menus contain any available designs and solutions. You can use the drop-down menus to choose from the available designs and solutions.

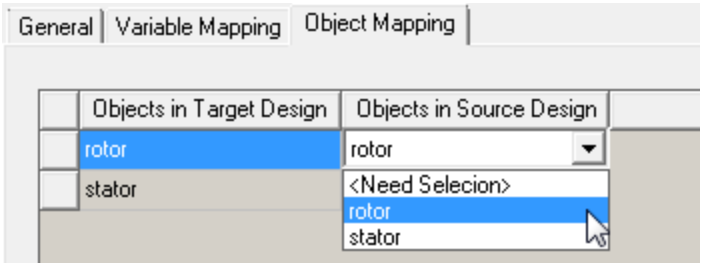
   The “Default” solution is the product-dependent solution of the first Setup. That is the setup listed first in the source design's project tree (alphanumerical order). A product-specific solution of this setup becomes the default solution. In most products, it is “LastAdaptive.”

7. Use the check box to specify whether to **Simulate source design as needed**.
8. Use the check box to specify whether to **Preserve source design solution**. Note that in the extractor mode, the source project will be saved upon exit. Extractor mode means that the software is opened during the link solely for the purpose of solving.
9. The **Variable Mapping** tab lets you view any variables contained in the selected project.

   When there are variables in the source design, you can choose to “map” these variables to constant values, expressions or variables in the target designs. Variable mapping becomes more important when the datalink type requires source and target design to be geometrically identical and source design is geometrically parameterized.

   For linked designs with variables of the same name, you can click **Map Variable by Name** to automatically map same named variables.

10. For object-based permanent magnet field excitations only, the **Object Mapping** tab lets you map objects in the target design to objects in the source design.



11. Click **OK** to close the **Setup Link** dialog box and return to the **Permanent Magnet Field** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12. Optionally, the **Defaults** tab allows you either to save the user-defined data for the excitation as the default for initializing new excitations of this type, or to clear any existing user-defined defaults and revert to standard defaults.
13. When finished, click **OK** to accept the settings and close the dialog box.

Setting Demagnetization/Magnetization Computations for Source Designs

(2D and 3D Magnetostatic and Transient Solvers)

When a magnetic device with permanent magnets is overloaded, or after a short circuit, irreversible demagnetization may occur due to magnetic field and temperature changes. As a result, the subsequent operating point will no longer lie along the original BH curve, but along the demagnetized recoil line at the worst operating condition.

This process can be modeled in Maxwell using **Set Demagnetization/Magnetization Computations**. The demagnetization/magnetization process of the magnet is modeled in the *source* design, and the practical normal application is modeled in the *target* design.

The linked demagnetization or magnetization designs act as sources (excitations) in the target design. These links are managed as Permanent Magnet Field excitations. Users can setup either object-based or design-based links. The Hc outputs from the source design are mapped to the target design mesh, thus eliminating the restriction of target/source designs employing the same geometries.

> **Note:** This mapping is straightforward for non-hysteresis based models because the Hc output from the source design is a single value plus direction per mesh element. For hysteresis-based models, the Hc output from the source design is a demagnetizing curve per mesh element, and a mapping approach is non-trivial. Thus this process is not applicable when the design has any object modeled with hysteresis materials. Demagnetization/magnetization links for this type of design continues to require the same mesh between target and source designs.
>
> **Limitations**:
>
> - A design can only be the target for either magnetization or demagnetization, but not both.
> - A design can compute either magnetized or demagnetized operating points, but not both.
> - Object-based and design-based links cannot coexist.
> - Object-based links are not supported when there is any object assigned with a hysteresis-based material model.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the magnetization computation, Maxwell provides two possibilities: **anisotropic magnetization**, and **isotropic magnetization**.

For **anisotropic magnetization**, the magnetization direction is determined by the direction specified by the user in the field for the Unit Vector input of Magnetic Coercivity on the panel of View/Edit Material during material setup.

> **Warning:** The combination of anisotropic and inhomogeneous magnetization (varying magnetization direction) is not supported in Maxwell because the magnetization state in the material volume must be connected to a single coordinate system.

For **isotropic magnetization**, the magnetization direction is determined by the orientation of the computed magnetizing field from the field solution. In such a case, the user has to set *all three* components of the Unit Vector input of Magnetic Coercivity to *zero (0)* on the panel of View/Edit Material window during material setup so that Maxwell knows the direction is computed, rather than specified by user.

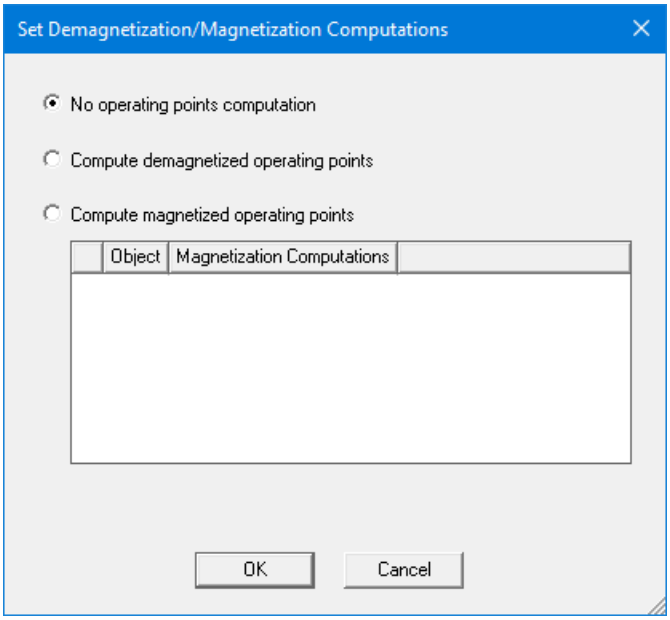
General Procedure for Setting Demagnetization/Magnetization Computations for Source Designs

To set demagnetization/magnetization process computations for a *source* design:

> **Note:** Demagnetization/Magnetization points computation settings specific to the Hysteresis Model-Based Magnetization Approach are described in the Using the Hysteresis Model-Based Magnetization Approach section.

1. Click **Maxwell** > **Excitations** > **Set Magnetization Computation** to open the **Set Demagnetization/Magnetization Computations** dialog box. You can also open the dialog box by right-clicking **Excitations** in the Project Manager, then selecting **Set Magnetization Computation** on the context menu; or by right-clicking in the Modeler window and selecting **Assign Excitation** > **Set Magnetization Computation**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



By default, **No operating points computation** is selected.

2. Select either the **Compute demagnetized operating points** or **Compute magnetized operating points** radio button. At least one object must be selected to compute data if **Compute magnetized operating points** is selected.

   > **Note:** Multi-select is supported by holding the **Ctrl** or **Shift** key while clicking.

   > **Note:** An object coordinate system (CS) must be created and assigned to each magnet object in the history tree.

3. Click **OK** to accept the settings and close the dialog.

The configured source design can now be linked to if you want to create Permanent Magnet Field Excitations for 2D and 3D Transient and Magnetostatic Solvers.

Using the Hysteresis Model-Based Demagnetization/Magnetization Approach

(Transient Solver Only)

> **Note:** The Transient A-Phi Formulation does not support the Hysteresis Model.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Currently, demagnetization is modeled based on single demagnetization curve in the second quadrant, possibly extended to the third quadrant in the user-specified direction. Hysteresis-based demagnetization is able to consider magnetization and demagnetization behavior at the same time in all directions, not just user-specified initial direction.

To enable hysteresis model-based demagnetization, and set the Demagnetization/Magnetization computations for source designs:

- Define the hysteresis loop by inputting the descending branch of a hysteresis loop (not supported by defining Hysteresis Model under Core Loss Model).
- Set the initial demagnetizing direction for the starting point on the hysteresis loop by setting the direction in the field for the Unit Vector input of Magnetic Coercivity on the panel of View/Edit Material during material setup. (The initial value is inherently described by the point (Hc = 0 and B = Br).

If a problem setting does not satisfy the above two conditions at the same time, the transient solver will use the “classic” approach to simulate demagnetization process if possible. Note that the valid descending branch of a hysteresis loop should be input with the same absolute values (H and B) for the 1st point and the last point. This modeling approach is not supported for TDM since the modeling of hard hysteresis material cannot be supported by Time decomposition Method (TDM).

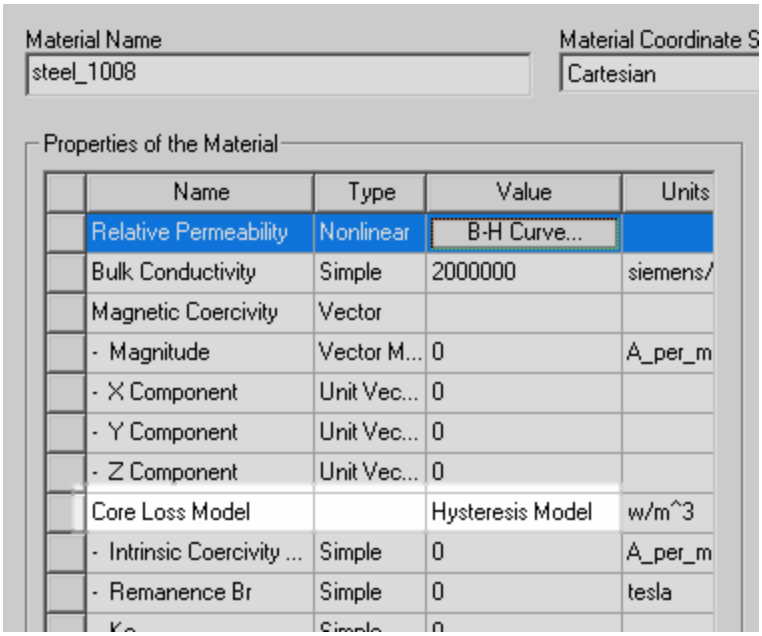
Using the Hysteresis Model-Based Magnetization Approach

(Transient Solver Only)

> **Note:** Transient A-Phi Formulation does not support the Hysteresis Model option.

With the introduction of vector hysteresis modeling capability in the transient solver, it is now possible to simulate the hysteresis model-based magnetization process if a user defines a normal nonlinear BH curve in the 1st quadrant together with the input of Hc (or additional Br for better accuracy of hysteresis loop construction) by selecting **Hysteresis Model** under **Core Loss Model** for the material.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



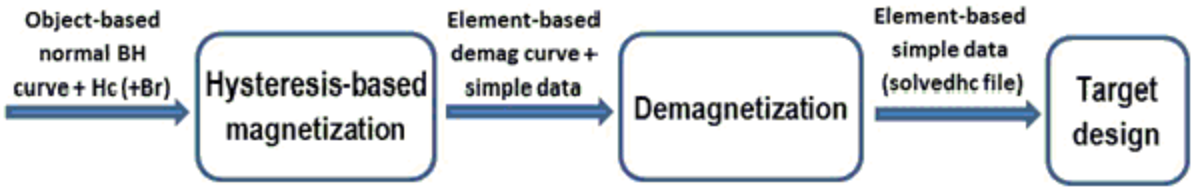
In such a case, magnetization is always considered as isotropic magnetization, that is, magnetization direction is determined by the orientation of the computed field. If a user defines only a normal nonlinear BH curve in the 1st quadrant without defining the Hysteresis model, the transient solver will use the “classic” approach to simulate magnetization process.

Classic Magnetization Approach

Using the classic magnetization approach, if the geometry in the source and target designs is the same, Maxwell maps the computed magnetization data element-by-element. However, the geometry in the source and target designs need not be identical, in which case the geometries in the two designs will be mapped by the solver using the object name. For example, magnetization data on “Box1” in the source design will be applied to the object called “Box1” in the target design. The user must ensure that the source and target object names are identical. The source design will compute magnetization data on the selected objects, while the target design will use the dynamic data in its simulation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Hysteresis Model-Based Magnetization Approach



Using the hysteresis model-based magnetization approach, after the completion of the last time step before stop, the hysteresis model for each element will generate the element-based demagnetization curve originated at the last operating point. As a result, the output from the magnetization modeling process is an element-based demagnetization curve, rather than a constant. Consequently, an additional demagnetization process is required based on the element-based demagnetization curve. In this approach, use of the same mesh between the source and the two target designs is required due to the use of the element-based demagnetization curve as the output of magnetization simulation.

Note:

- If an object is defined as hysteresis model, but not selected for magnetization, the object is no longer considered as a hysteresis material because magnetization and hysteresis modeling cannot be conducted in the same design.
- Hysteresis based magnetization is always considered as isotropic magnetization, that is, its direction is determined by the orientation of the computed field.
- For objects that have been selected for magnetization, but not defined as hysteresis models, the existing classic approach (*simple data* in the diagram above) will be used. Mixing hysteresis model-based magnetization method with the classic magnetization method is supported.
- For the classic approach, no demagnetization step is required; while for the hysteresis model-based magnetization approach, a demagnetization step is normally required. For the demagnetization step, the input is the element-based demagnetization curve derived from the magnetization run.
- When **Clean Stop** is invoked for projects in which a TDM transient source design is linked to a transient target design, after completion of the source design the solution process can continue from the target design. In such a case, the target design must be non-TDM.

Design Criteria

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

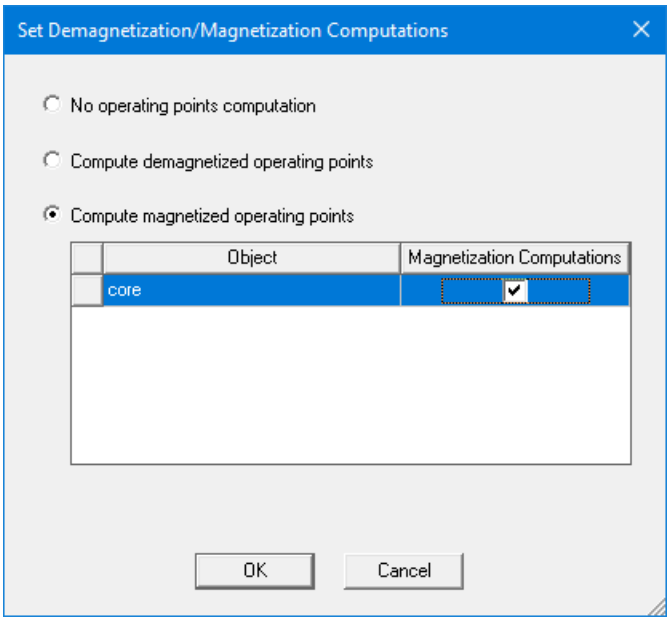
- The source design must have objects assigned with nonlinear, non-permanent-magnet material properties.
- The same design cannot be the source for both magnetization and demagnetization links.
- The source design cannot also be a target of a demagnetization link.
- The target design cannot also be a target or source of a demagnetization link.

The hysteresis model-based magnetization approach requires three designs, one for each step in the hysteresis model-based magnetization approach.

First Design: Set up a Hysteresis-Based Magnetization Computation Design (source design)

1. Create a Maxwell 2D or 3D magnetic transient design (the *Hysteresis-based magnetization* design in the figure shown above).
2. Click **Maxwell** > **Excitations** > **Set Magnetization Computation** to open the **Set Demagnetization/Magnetization Computations** dialog box.

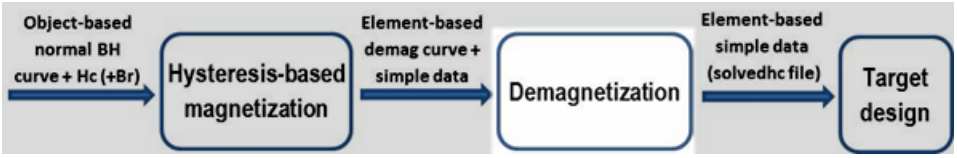
   You can also open the **Set Demagnetization/Magnetization Computations** dialog box by right-clicking **Excitations** in the Project Manager, then selecting **Set Magnetization Computation** on the context menu; or by right-clicking in the Modeler window and selecting **Assign Excitation** > **Set Magnetization Computation**.
3. Select **Compute magnetized operating points**, and select the objects for magnetization and/or hysteresis-based magnetization computation, then click **OK** to accept the settings and close the dialog box.



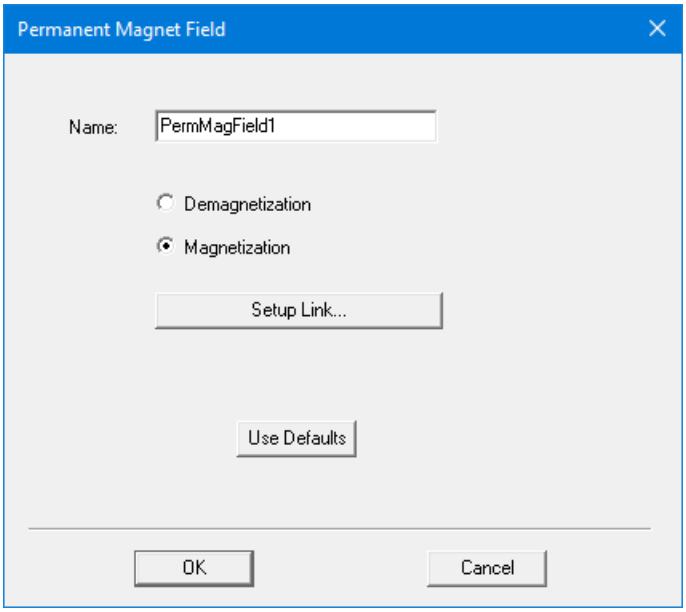
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Solve the design and the solution will include the element-based magnetization data, and/or hysteresis-based demagnetization curve if the design uses nonlinear materials with non-zero Hc defined.

Second Design: Set up a *Demagnetization* design with Output of the Hysteresis-Based Magnetization Computation

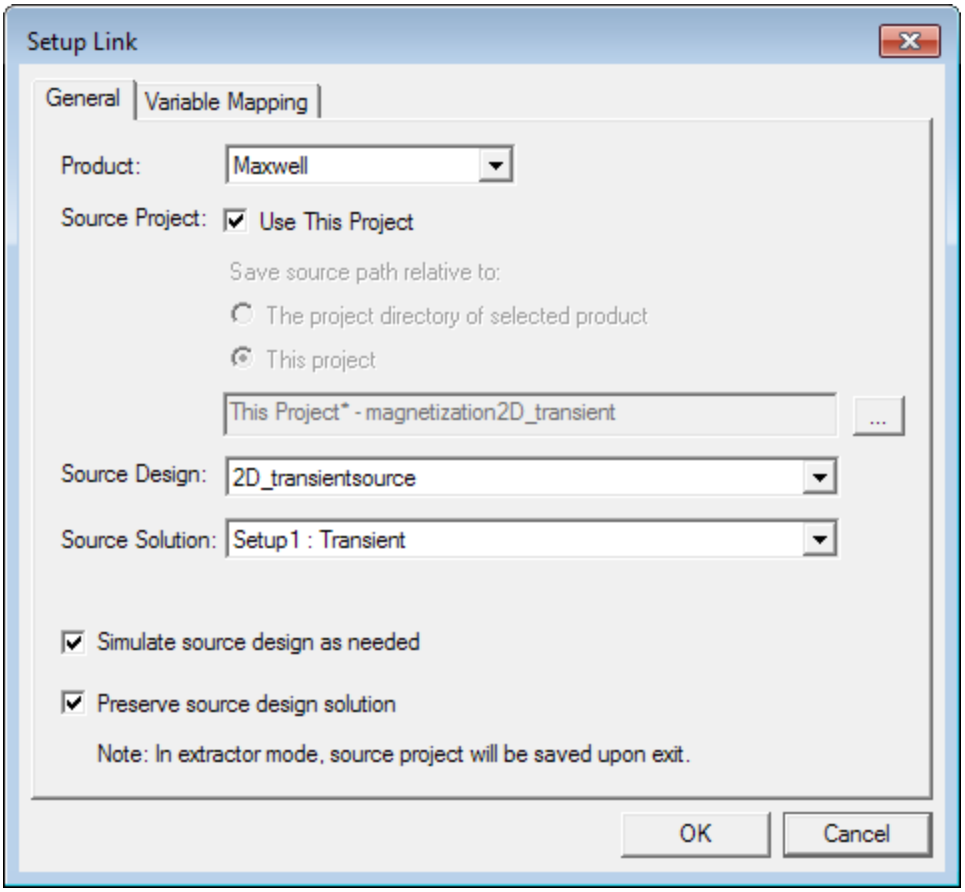


1. Create another Maxwell 2D or 3D magnetic transient design to be used as the *Demagnetization* design (highlighted above).
2. In the Project tree, right-click **Excitations** and select **Assign** > **Permanent Magnet Field**.
3. In the **Permanent Magnet Field** dialog box, select **Magnetization**, then click **Setup Link.**



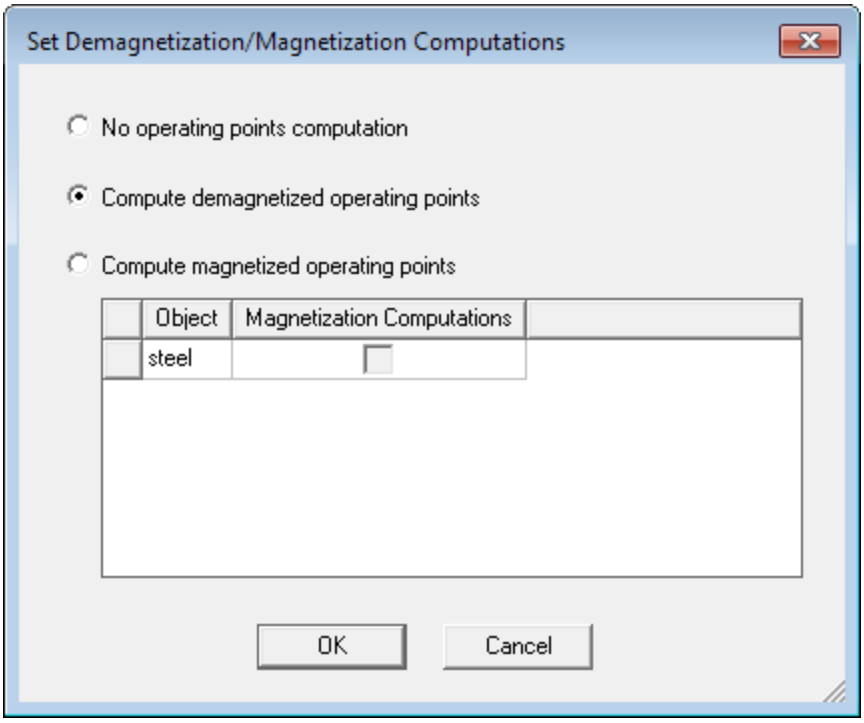
4. Set up a link to the *Hysteresis-based magnetization computation* source design (the first design you set up), which provides the hysteresis-based magnetization solution data. Check **Simulate source design as needed** and **Preserve source design solution**. If the source design is not set up to **Compute magnetized operating points**, an error message will be issued.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Right-click **Excitations** in the Project tree and select **Set Magnetization Computation** on the context menu to open the **Set Demagnetization/Magnetization Computations** dialog box, and select **Compute demagnetized operating points**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



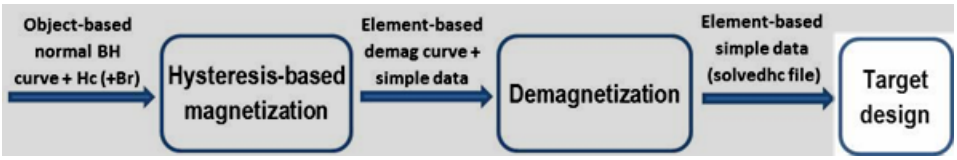
> **Note:** Steps 6 and 7 are applicable only for *object-based* magnetization/demagnetization links. Omit these steps if *design-based* magnetization/demagnetization links are used.

6. Open the **Solve Setup** dialog and select **Import mesh** on the **Advanced** tab to open the **Setup Link** dialog box.
7. In the **Setup Link** dialog box, for the **Source Design**, choose the *Hysteresis-based magnetization computation* (the first design you set up).
8. Solve the design and the solution will include the element-based demagnetization distribution curve (which is based on the Hysteresis-based magnetization data). If the source design has not been solved, solving the current (demagnetization) design causes the source design to be solved first.

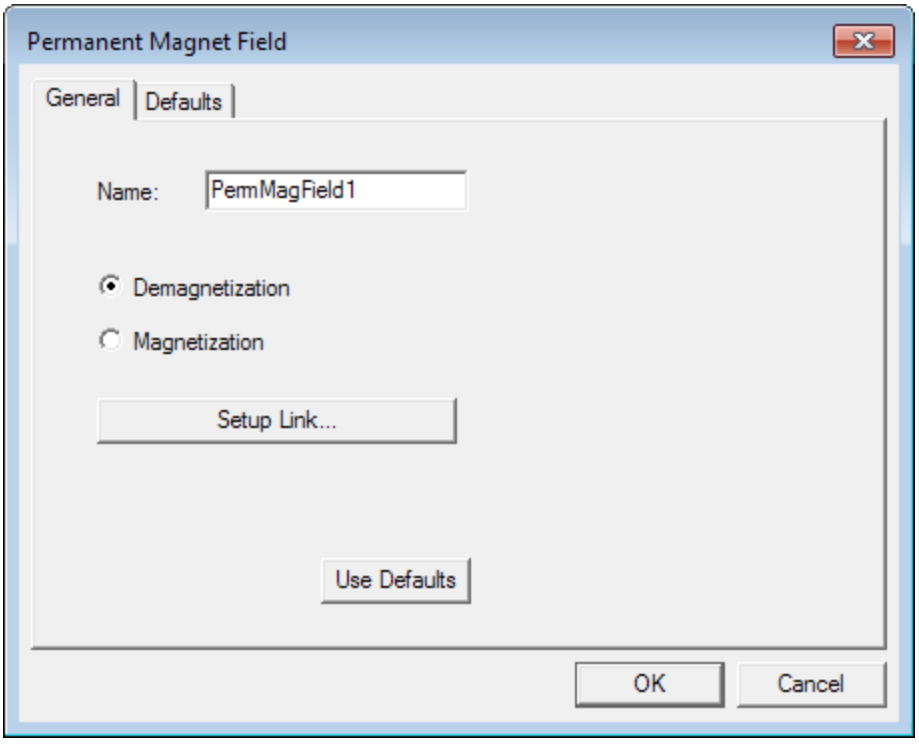
> **Note:** To ensure that new data from the source design is used if the source design has been modified, right-click the Analysis icon of the source design in the project tree, and click **Clear Linked Data**. This flushes existing linked data so that solving the current design will force the source design to be solved first.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Third Design: Set up the *Target* Design

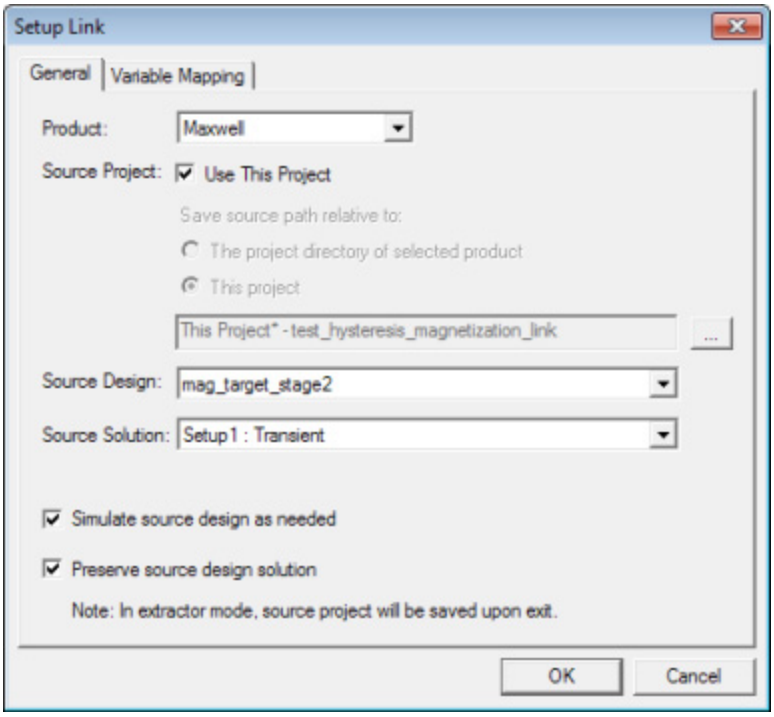


1. Create a third Maxwell 2D or 3D magnetic transient design to be used at the *Target* design (highlighted above).
2. In the Project Manager tree, right-click **Excitations** and select **Assign** > **Permanent Magnet Field**.
3. In the **Permanent Magnet Field** dialog box, select **Demagnetization**, then click **Setup Link.**



4. Set up a link to the *Demagnetization* design (the second design created above), which provides the dynamic demagnetization distribution solution data (based on the *Hysteresis-based magnetization* solution data from the first design). Be sure to check **Simulate source design as needed** and **Preserve source design solution**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:** Steps 5 and 6 are applicable only for *object-based* magnetization/demagnetization links. Omit these steps if *design-based* magnetization/demagnetization links are used.

5. Open the **Solve Setup** dialog and select **Import mesh** on the **Advanced** tab to open the **Setup Link** dialog box.
6. In the **Setup Link** dialog box, for the **Source Design**, choose the *Demagnetization* design (the second design created above).
7. Solve the *Target* design. If the source designs for hysteresis-based magnetization computation and demagnetization are not yet solved, solving the current (target) design causes the two source designs to be solved in sequence first.

> **Note:** To ensure that new data from the source designs is used if the source designs have been modified, right-click the Analysis icon of each source design in the Project Manager tree, and click **Clear Linked Data**. This flushes existing linked data so that solving the current design will force the source designs to be solved first.

Showing and Verifying Conduction Paths

To show the conduction paths in the geometry:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Maxwell** > **Excitations** > **Conduction Paths** > **Show Conduction Paths**.

   The **Conduction Path Visualization** window appears.

2. Click a path in the list.
   The path is highlighted in the model.
3. Click **Close** when you are done viewing conduction paths.

To verify conduction paths:

- Click **Maxwell** > **Excitations** > **Conduction Paths** > **Verify Conduction Paths**.

  The paths are traced in the model diagram.

Recalculating Conduction Paths

Conduction paths are not recalculated unless a design changes results in a different outcome.

You can force a recalculation by using the **Maxwell** > **Excitations** > **Conduction Paths** > **Recalculate Conduction Paths** command.

The **Recalculate Conduction Paths** command is similar to the **Verify Conduction Paths** command. The only difference is that the **Recalculate** command recalculates every time, even if there are no changes that influence the conduction paths.

> **Note:** The **Recalculate Conduction Paths** command does not apply for electrostatic solution types.

To recalculate conduction paths for the active project, click **Maxwell** > **Excitations** > **Conduction Paths** > **Recalculate Conduction Paths**.

Modifying Boundary Conditions and Excitations

Save your project before modifying boundary conditions and excitations. You can modify boundary conditions and excitations after a solution has been generated, but you may lose some solution data.

Duplicating Boundaries and Excitations

To duplicate a boundary or excitation when its geometry is pasted or duplicated:

1. Click **Tools** > **Options** > **General Options** to open the **Options** dialog box, displaying a hierarchical list of options. Click the + next to **Maxwell 2D** and select **Boundary Assignment**.
2. Select **Duplicate boundaries/mesh operations with geometry**.
3. Click **OK**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

All boundaries and excitations are duplicated with their associated geometries until you clear this option.

Tip: Use this option to copy and paste boundaries. For example, do the following to use the same boundary on multiple objects:

1. Select the face to which you want to assign the boundary.
2. Click **Modeler** > **Surface** > **Create Object From Face** to create a new object from the existing face.
3. Assign the boundary to the new face object.
4. Copy and paste the new face object to copy and paste the boundary.

Showing and Hiding Boundaries and Excitations

You can choose to show or hide a boundary or excitation's geometry, name, or vectors in the active view window or in all view windows.

Showing and Hiding Boundaries and Excitations in Every View Window

1. Click **Maxwell** > **Boundaries** > **Visualization** or **Maxwell** > **Excitations** > **Visualization**. The **Visualization Options** window appears.
2. Select one of the following from the **Select** drop-down menu:
    - All
    - All Boundaries
    - All Excitations
    - All Ports
    - By Name
3. If you select **By Name**, enter a name in the **Select By Name** window, and click **OK**.
4. To clear the selection of all boundaries and excitations, click **Deselect All**.
5. Click **Close** to close the **Visualization Options** window.

Setting Default Values for Boundaries and Excitations

When assigning a boundary or excitation, many of the fields in the boundary and excitation dialog boxes have default values associated with them. These default values are initially set by Maxwell, but can be overridden.

To modify the default values associated with a specific boundary or excitation type:

1. Assign a boundary or excitation.
2. Modify any default values.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Close the boundary or excitation's dialog box.
4. Re-open the new boundary or excitation's dialog box. It now includes a **Defaults** tab.
5. Under the **Defaults** tab, click **Save Defaults**.

   The values assigned to this boundary are saved as the default values and are assigned when new boundaries of this type are created.

6. Optionally, click **Revert to Standard Defaults**.

The default values you set for this boundary type are cleared and will revert to the default values set by Maxwell.

Reassigning Excitations

You can reassign an excitation to another surface. This is useful when you have modified objects with assigned excitations, invalidating the excitations. For example, if you unite two objects with assigned excitations, the second object's excitation will become invalid because united objects maintain the characteristics of the first object selected. In this case, you would need to reassign the excitation or delete it. You can use a **Reassign Excitation** dialog box accessed through the **Maxwell** > **Excitation** > **Reassign** menu, or right-click on the Excitation of interest in the Project tree, and select from the shortcut menu to **Reassign**, **Add to Assignment**, or **Remove from Assignment**. You can also determine a current assignment by right-clicking on the excitation in the **Project** tree and by using the **Select Assignment** command on the short cut menu.

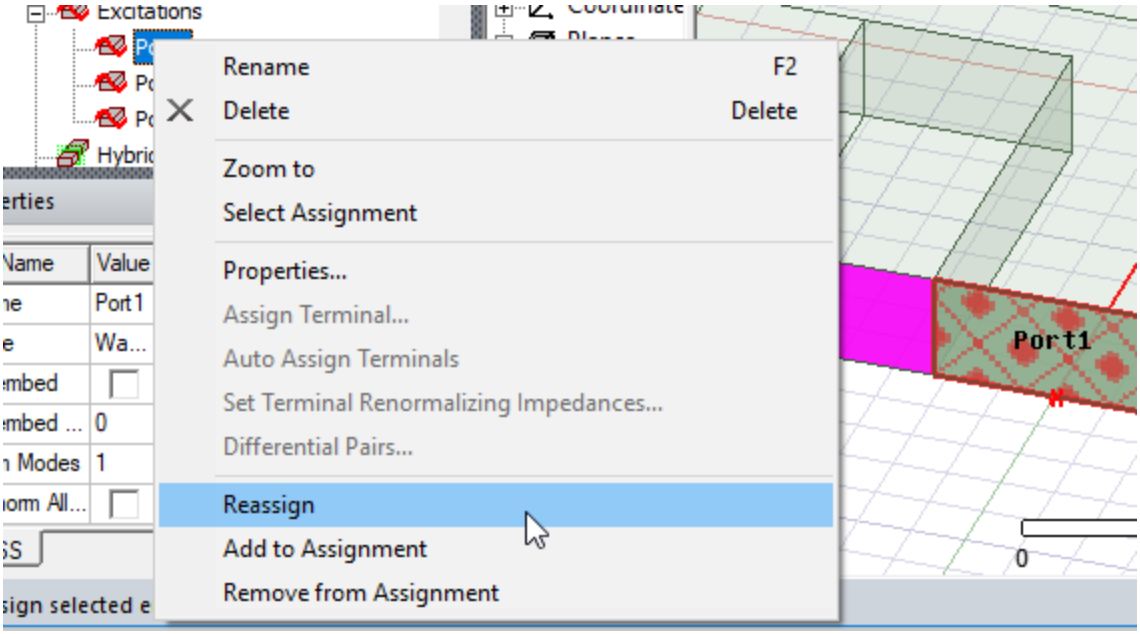
Prerequisites:

- The surface to which you want to **Reassign**, or **Add to Assignment** must be on the same plane as existing assignment. It does not have to be in contact to **Reassign**, but must be in contact to **Add to Assignment**.
- You must first select the surface that you want to reassign or add to an assignment.

To use the short-cut menu to reassign or add an excitation:

1. With the target surface selected, right-click on the Excitation of interest to invoke the menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select **Reassign** to change the assignment to the selected surface, or **Add to Assignment** to include surface.

To use the shortcut menu to Remove a selected surface from an assignment:

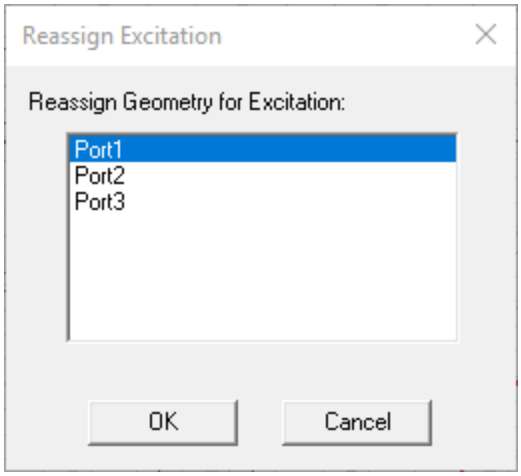
1. With the target surface selected, right-click on the Excitation to invoke the menu.
2. Click **Remove from Assignment**.

To use the Maxwell Menu and the Reassign Excitation dialog box:

1. Select the object or object face to which you want to assign an existing excitation.
2. Click **Maxwell** > **Excitations** > **Reassign**.

The **Reassign Excitation** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Select an existing excitation from the list, and then click **OK**.

The excitation is reassigned to the object or object face.

> **Tip:** When reassigning an excitation that includes vectors in its definition, Maxwell attempts to preserve the vectors with the new assignment, but this is not always possible.

Deleting All Boundaries and/or Excitations

To delete all boundaries or excitations:

- Click **Maxwell** > **Boundaries** > **Delete All** or **Maxwell** > **Excitations** > **Delete All**.

Reprioritizing Boundaries

Each boundary you assign overwrites any existing boundary which it overlaps. You can change the priority of a previously assigned boundary to be greater than a more recently assigned boundary.

The order of boundaries and excitations is important because, for any given triangle of the mesh, only one boundary or excitation can be visible to the solvers. The highest priority boundary condition applied on a certain surface overwrites the lower priority one(s) applied on the same surface. If any surfaces with boundary conditions have a partial overlap, then in the overlap region, the precedence rule applies. The highest priority boundary condition takes precedence over the lower one(s) such that for all mesh nodes on any surface carrying a boundary condition, only one boundary condition is actually enforced and visible to the solver.

Excitations cannot be reprioritized. Only boundaries can be reprioritized.

1. Click **Maxwell** > **Boundaries** > **Reprioritize** to reprioritize boundaries.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Reprioritize Boundaries** window appears. The order the boundaries and excitations appear in the list indicates the order in which they were defined. The lowest priority assignment appears at the top of the list.

Ports are automatically placed at the bottom (highest priority) of the list; you cannot move a boundary to a higher priority than a port. Magnetic Bias Excitations (if any) have the lowest priority. Other boundaries and excitations appear between these two extremes.

2. Drag the boundary you want to change to the desired order of priority.

> **Note:**
>
> The order of boundaries and excitations in the project tree is alphabetical. The order does not correspond to the order of boundaries and excitations visible to the solvers.

Viewing and Editing Boundary or Excitation Properties

To open the **Properties** window for a boundary or excitation that is already assigned:

1. In the project tree, under either **Boundaries** or **Excitations**, right-click a specific boundary or excitation, and select **Properties**.
   The **Properties** window for that boundary/excitation appears.
2. Make the desired changes, and click **OK**.

Reviewing Boundaries and Excitations in the Solver View

After you have assigned all the necessary boundaries and excitations to a model, you should review their order of priority according to the Ansys Electronics Desktop solver. Reviewing the solver's view of the model's boundaries and excitations enables you to verify that their order during the solution process will be as you intended.

To check the solver's view of boundaries and excitations:

1. On the **Maxwell** menu, click **Boundary Display (Solver View)**.

   Ansys Electronics Desktop generates an initial mesh and determines the locations of the boundaries and excitations on the model.

   The **Solver View of Boundaries** window appears, which lists all the boundaries and excitations for the active model in the order specified in the **Reprioritize Boundaries and Excitations** dialog box.

2. Select the **Visibility** option for the boundary or excitation you want to review.

   The selected boundary or excitation will appear in the **3D Modeler** window in the color it has been assigned.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

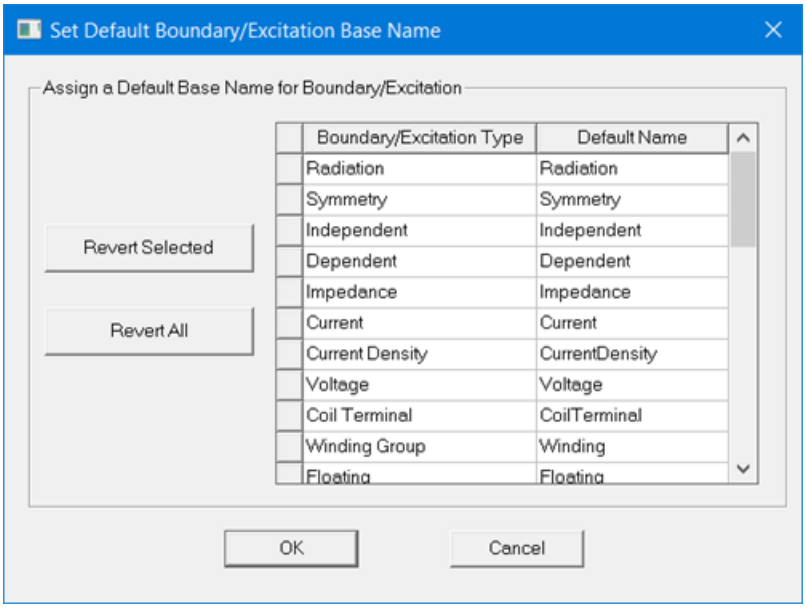
- **Visible to Solver** will appear in the **Solver Visibility** column for each boundary or excitation that is valid.
- **Overridden** will appear in the **Solver Visibility** column for each boundary or excitation that will be ignored by the solver as a result of it overlapping an existing boundary or excitation with a higher priority.

3. Verify that the boundaries or excitations you assigned to the model are being displayed as you intended for solving purposes.
4. If the order of priority is not as you intended, reprioritize the boundaries and excitations.

Set Default Boundary/Excitation Base Names

When setting boundaries and excitations on multiple selected surfaces, separate boundary or excitation definitions may be made automatically. Individual entries in the **Project Tree** will be identified with a **Base Name** and assigned a sequential numerical value appended to the base name.

The **Set Default Boundary/Excitation Base Name** dialog box allows the base names for all boundary and excitation type to be specified for the current project.



To set the base name for a boundary or excitation type:

1. Right click the **Excitations** or **Boundaries** entry in the Project Manager window, and select **Set Default Base Name**.
2. Scroll the table to the boundary or excitation of interest.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select the value in the **Default Name** column in the row of interest and enter the new base name. Names must be less than sixty characters and cannot include spaces. Letters, numbers, and special characters are permitted. Illegal names are not accepted and generate a warning message.
4. Click **OK** to accept the changes and dismiss the dialog box.

All base names may be reset to the factory default list by clicking **Revert All**, or individuals may be reset by first selecting the **Default Name** to be reset and clicking the **Revert Selected** button.

Excitation Name Ordering Conventions

Ansys Electronics Desktop tries to sort excitations by name, using a sort of "improved" dictionary (lexicographic) ordering that takes into account trailing numerals as a special case.

So if you create some ports in the following chronological order, Port3, Port1A, Port10, Port1, Port3A, Port1A1, Port10A, you will find that they appear in the list as:

Port1, Port3, Port10, Port10A, Port1A, Port1A1, Port3A

Explanation:

- Port1, Port3, and Port10 start with the same string but end in trailing numerals, so they are sorted according to the numeric value of the trailing numerals: 1 < 3 < 10. This is different from the standard dictionary sorting order, which would order them as Port1, Port10, Port3.
- Port10A and Port1A both start with 'Port1' but don't end with trailing numerals, so standard dictionary sorting applies. Numbers come before letters, thus Port10A comes before Port1A (0 is before A in position 6).
- Shorter string ABC comes before longer string ABCD (ABC is a "prefix" of ABCD), so Port10 is before Port10A, and Port1A is before Port1A1.

If the leading strings contain punctuation characters (. and _). The ordering is by the character's ASCII code, so period (ASCII 46) is before underscore (ASCII 95.)

Magnetostatic Boundaries and Excitations

Define at least one of the following as a source of static magnetic fields:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The current in a conduction path, which can be either stranded or solid.

  > **Note:** For stranded conductors, it is assumed that the current density is uniform on the cross-section of the respective conductors. Thus, the stranded option for current excitations should mostly be used only in cases where the cross-section of conductors is constant, consistently with the assumption that the respective object is a coil built with strands of wire. For this reason, conductors (coils) could be created by either sweeping around an axis using zero segments or sweeping along a path having only true surface segments in order to create a smooth surface coil having a completely uniform cross-sectional area over its entire length.

- The current density in a conductor.
- The voltage differential across a conduction path.
- The magnetic field on an outside surface.
- A permanent magnet.

If currents or current densities are the only sources of static magnetic fields in your model, set at least one of the following as the outer boundary:

- The default boundary conditions.
- An odd symmetry (flux tangential) boundary.
- An even symmetry (flux normal) boundary.

Magnetostatic Boundaries

The magnetostatic field solver allows you to define the following types of boundaries:

Boundary Type	H-Field Behavior	Used to model...
Default Boundary Conditions (Natural and Neumann)	Field behaves as follows: • Natural boundaries — **H** is continuous across the boundary. • Neumann boundaries — **H** is tangential to the boundary and flux cannot cross it.	Ordinary field behavior. Initially, object interfaces are natural boundaries; outer boundaries, and excluded objects are Neumann boundaries.
Zero Tangential H Field	Flux is perpendicular if tangent components are zero or if a zero tangential H-field boundary was applied.	External magnetic fields.
Tangential H	The tangential components of **H** are set to	External magnetic

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Field	predefined values.	fields.
Integrated Zero Tangential H Field	For applications such as a motor with the shaft excluded, there remains a hole in the middle. In such cases, on the hole's boundary, neither the Tangential H Field boundary (where integration of the tangential H field is non-zero), nor the Zero Tangential H Field boundary (where the tangential H field is zero everywhere) can be applied because – for the hole boundary case – integration of the tangential H field is zero, but the tangential H field is not zero everywhere. Integrated Zero Tangential H Field can be used for such applications.	Cases such as the boundary around the hole that remains when a motor's shaft is excluded.
Insulating	Same as Neumann, except that current cannot cross the boundary.	Thin, perfectly insulating sheets between touching conductors.
Symmetry	Field behaves as follows: • Odd Symmetry (Flux Tangential) — **H** is tangential to the boundary; its normal components are zero. • Even Symmetry (Flux Normal) — **H** is normal to the boundary; its tangential components are zero.	Planes of geometric and magnetic symmetry.
Matching (Independent and Dependent)	The H-field on the dependent boundary is forced to match the magnitude and direction (or the negative of the direction) of the H-field on the independent boundary.	Planes of symmetry in periodic structures where **H** is oblique to the boundary.
Thin Layer	When two parts of the domain are separated by a thin volume representing an air gap, it can be modeled by a surface with an appropriate boundary condition. This boundary creates a field discontinuity between the two sides of the selected surface based on its thickness. The relative magnetic permeability of the air gap is assumed to be equal to one.	Narrow air gaps

Default Boundary Conditions for a Magnetostatic Solver

These boundary conditions are automatically defined for a magnetostatic model:

- Natural boundaries are assigned to the surfaces between objects.
- Neumann boundaries are assigned to the outside edges of the problem region.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To leave a surface set to its default boundary condition, do nothing. Deleted boundary conditions and excitations automatically reset to the default boundary conditions.

Assigning a Magnetic H Field for a Magnetostatic Solver

This type of boundary defines external magnetic fields in a model. Assign it only to the outer surfaces of the problem region.

> **Warning:** When using tangential magnetic field boundary conditions, always double-check that Ampere's law is not violated.

> **Note:**
> The specific boundary types you can define depend on the solver you have chosen for your project.
> - Electrostatic Boundary Conditions
> - DC Conduction Boundary Conditions
> - Magnetostatic Boundary Conditions
> - AC Magnetic Boundary Conditions

To define a zero tangential H field boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).
2. Click **Maxwell** > **Boundaries** > **Assign** > **Zero Tangential H Field**.
   The **Zero Tangential H Field** window appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Click **OK** to assign the boundary to the selected object.

To define a tangential H field boundary:

1. Select the section of the geometry on which you want to apply the boundary condition.
2. Click **Maxwell** > **Boundaries** > **Assign** > **Tangential H Field**.
   The **Tangential H Field** window appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the value of the field component in the x-direction in the **U(X)** box and the value of the field component in the y-direction in the **V(Y)** box.
5. In the **Coordinate System** section, do the following to define the vectors:
   a. Select **New Vector** from the **U Vector** drop-down menu.
      The **Create Line** message appears asking you to draw the U vector of the coordinate system in the plane of the selected face.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Click two points to specify the vector.
   The vector is set, and the **Tangential H Field** window reappears, with **Defined** listed for **U Vector**.
c. To reverse the direction of the vector, click the **Reverse Direction** check box.

6. Optionally, click **Use Defaults** to revert to the default values in the window.
7. Click **OK** to assign the boundary to the selected object.

The new boundary is added to the boundary list in the Project Manager tree.

Assigning an Integrated Zero Tangential H Field Boundary

This type of boundary is applied when the integral of the tangential H field is zero.

> **Warning:** When using integrated zero tangential magnetic field boundary conditions, always double-check that Ampere's law is not violated!

To define an integrated zero tangential H field boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).
2. Click **Maxwell 3D** > **Boundaries** > **Assign** > **Integrated Zero Tangential H Field**.

   The **Integrated Zero Tangential H Field** window appears.

3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Click **OK** to assign the boundary to the selected object.

The new boundary is added to the boundary list in the project tree.

Assigning an Insulating Boundary for a Magnetostatic Solver

This boundary condition is used to model very thin sheets of perfectly Insulating material between touching conductors. Current cannot cross an insulating boundary.

To set an insulating boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).
2. Click **Maxwell** > **Boundaries** > **Assign** > **Insulating**.

   The **Insulating Boundary** window appears.

3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Click **OK** to assign the boundary to the selected object.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning a Symmetry Boundary Condition for a Magnetostatic Solver

This boundary condition defines a plane of geometric or magnetic symmetry in a structure. Assign it only to the outer surfaces of the problem region.

To set a symmetry boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).
2. Click **Maxwell** > **Boundaries** > **Assign** > **Symmetry**.
   The **Symmetry Boundary** window appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Select one of the following as the type of symmetry:

Odd (Flux Tangential)	**H** is tangential to the boundary; its normal components are zero.
Even (Flux Normal)	**H** is normal to the boundary; its tangential components are zero.

5. Optionally, click **Use Defaults** to revert to the default values in the window.
6. Click **OK** to assign the boundary to the selected object.

> **Warning:** When using even symmetry boundaries, double-check that Ampere's law is not violated.

Assigning an Independent Boundary for a Magnetostatic Solver

Independent and dependent boundaries enable you to model planes of periodicity where the H-field at every point on the dependent boundary surface is forced to match the H-field of every corresponding point on the independent boundary surface. The transformation used to map the H-field from the independent to the dependent is determined by specifying a coordinate system on both the independent and dependent boundaries.

To set an independent boundary:

1. Select the face to which you want to assign the independent boundary.
2. Click **Maxwell** > **Boundaries** > **Assign** > **Matching** > **Independent**.
   The **Independent Boundary** window appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. In the **Coordinate System** section, do the following to define the coordinate system:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** You must specify the coordinate system in the plane on which the boundary exists. First draw the U vector of the coordinate system. Maxwell uses the U vector you draw and the normal vector of the boundary face to calculate the v-axis. Then specify the direction of the V vector.

   a. Select **New Vector** from the **U Vector** drop-down menu.
      The **Create Line** message appears, asking you to draw the U vector of the coordinate system in the plane of the selected face, and the **Independent Boundary** dialog box disappears while you draw the U vector.
   b. Select the U vector's origin, which must be on the boundary's surface, in one of the following ways:
      - Click the point.
      - Type the point's coordinates in the in the **X**, **Y**, and **Z** boxes.
   c. Select a point on the u-axis.
      The **Independent Boundary** dialog box reappears
5. To reverse the direction of the vector, select the **Reverse Direction** check box.
6. Click **OK**.

The Maxwell solver computes the H-field on this boundary and maps it to the dependent boundary using the transformation defined by the independent and dependent coordinate systems.

Assigning a Dependent Boundary for a Magnetostatic Solver

Assigning a dependent boundary is the second step in creating matching boundaries. The field on the independent boundary is mapped to the dependent boundary.

> **Note:** You must define an independent boundary before creating the dependent boundaries that are associated with it. Assign dependent boundaries only to the outer surfaces of the problem region.

To set a dependent boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).
2. Click **Maxwell** > **Boundaries** > **Assign** > **Matching** > **Dependent**.

   The **Dependent Boundary** window appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Select the independent boundary to which the dependent is to be assigned from the **Independent** drop-down menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. To specify the coordinate system, do the following in the Coordinate System section:

   Note: You must specify the coordinate system in the plane on which the boundary exists. First draw the U vector of the coordinate system. Maxwell uses the U vector you draw and the normal vector of the boundary face to calculate the v-axis. Then specify the direction of the V vector.

   a. Select **New Vector** from the **U Vector** drop-down menu.
   The **Create Line** message appears asking you to draw the U vector of the coordinate system in the plane of the selected face, and the **Dependent Boundary** dialog box disappears while you draw the U vector.
   b. Select the U vector's origin in one of the following ways:
      - Click the point.
      - Type the point's coordinates in the in the **X**, **Y**, and **Z** boxes.

      Note: The U vector's origin must be on the boundary's surface.

   c. Select a point on the u-axis in the same way.
   When the second point is identified, the vector is set, and the **Dependent Boundary** window reappears, with **Defined** listed in the **U Vector** drop-down menu.
   d. To specify the direction of the V vector, select or clear the **Reverse Direction** check box.
6. In the **Relation** section, select one of the following to specify the field behavior on the boundary:

Hdep = Hind	Click this radio button if the dependent and independent boundaries have the same magnitude and direction.
Hdep = -Hind	Click this radio button if the dependent boundary field has the same magnitude as but the opposite direction from the independent boundary field.

7. Optionally, click **Use Defaults** to revert to the default values in the window.
8. Click **OK** to assign the boundary to the selected object.

Note: The origin must be a vertex point of one of the objects.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning a Thin Layer (Air Gap) Boundary for a Magnetic Solver

When two parts of the domain are separated by a thin volume representing an air gap, it can be modeled using a **Thin Layer** boundary condition. This boundary creates a scalar potential discontinuity across the two sides of the surface based on its thickness. When the **Air Gap** option is selected, the relative magnetic permeability is assumed to be equal to one. The thin layer is defined purely in terms of its thickness in this case.

This boundary condition can be used to model thin ferromagnetic structures such as shields and narrow air gaps in power transformers, inductors, or any other electric device without needing to explicitly model (and mesh) the thin volume.

Note:

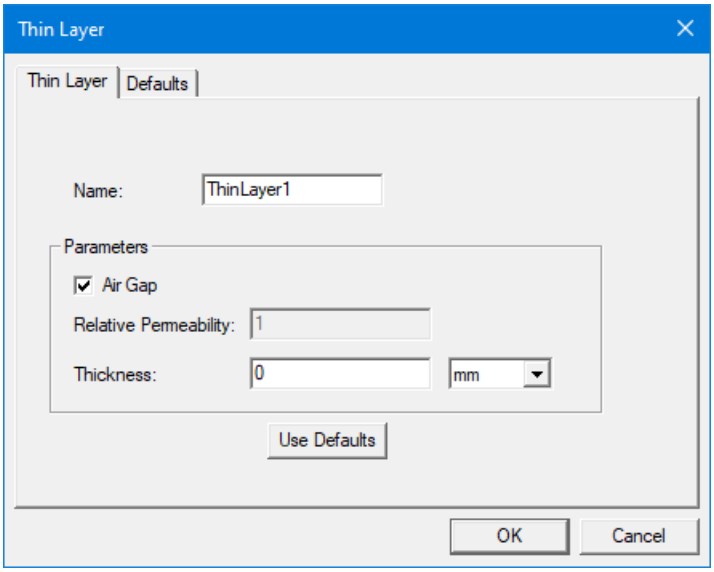
- The conductivity of the thin layer in the magnetostatic solution is assumed to be zero. This is a valid assumption as no currents flow inside the thin layer, which is guaranteed by the thin layer not touching conductors and by the fact that the magnetostatic solution cannot represent induced current effects. For the induced current analysis, the AC Magnetic solution is recommended.
- The thin layer cannot be a closed shell encompassing a conductor.
- The thin layer does not support a mechanical movement, *i.e.*, the boundary condition cannot be applied to the interface of moving and stationary bodies.
- The thin layer cannot be defined on a conductor boundary or as a sheet object that touches or crosses any conducting body. In such cases, the Maxwell solver displays an error message indicating the name of the conducting body that the thin layer is touching.

For the 3D Magnetostatic solver, the thin layer is defined in terms of its thickness and magnetic permeability.

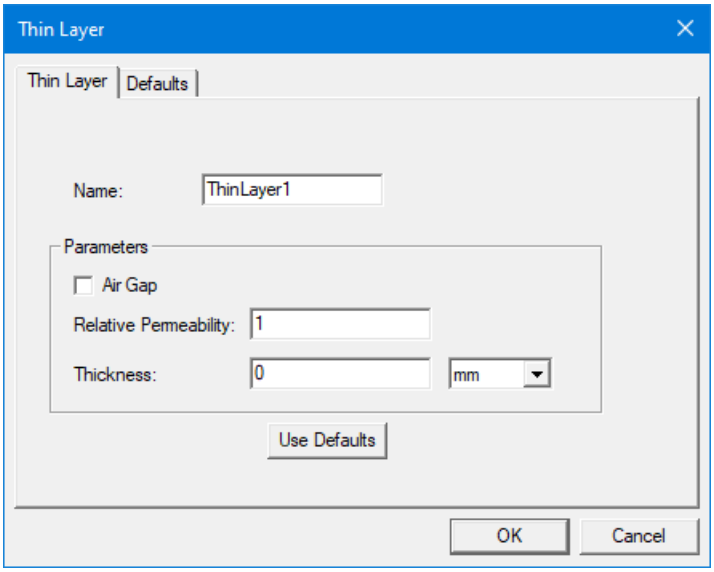
To define a thin layer boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a sheet object or face of a 3D body).
2. Click **Maxwell 3D** > **Boundaries** > **Assign** > **Thin Layer** to open the **Thin Layer** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Enter a name for the boundary in the **Name** box, or accept the default.
4. If an air gap is being modeled, keep the **Air Gap** option checked. The **Relative Permeability** field will be grayed out (non-editable) and set to 1.
5. If **Air Gap** is checked, enter the **Thickness** of the air gap and the unit of measure. It must be a value greater than zero. The use of a variable is supported, but it cannot be an intrinsic variable.
6. If the **Air Gap** is unchecked, the thin layer can represent a thin ferromagnet. In this case, enter the thickness of the layer and the material's Relative Permeability in the respective fields. The use of a variable is supported, but it cannot be an intrinsic variable.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Optionally, click **Use Defaults** to revert to the default values, which are defined in the **Defaults** tab.

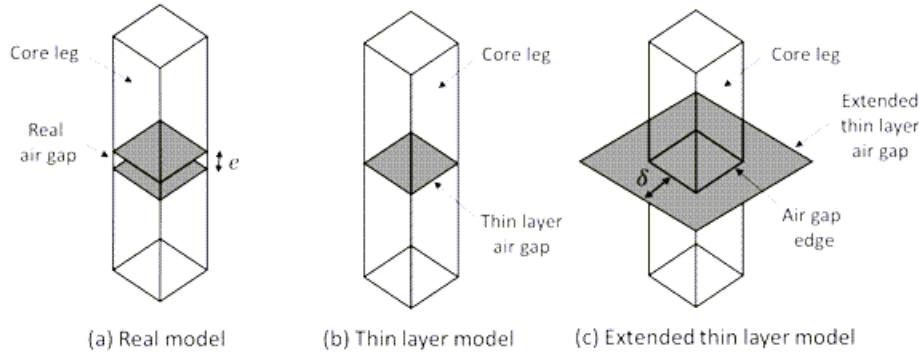
   From the **Defaults** tab, the **Save Defaults** button saves the values currently defined on the **Thin Layer** tab as the defaults to be assigned to new impedance boundaries. **Revert to Standard Defaults** clears existing user-defined values and replaces them with the standard default values.

8. Click **OK** to assign the boundary to the selected object.

The Project Manager tree lists the newly assigned thin layer boundary in the tree. You can select the boundary in the tree to view and edit its thickness in the Properties window. You can also double-click the boundary entry in the tree to open it for editing in the Thin Layer window.

Getting Better Convergence with the Thin Layer Model

For a magnetostatic simulation, the thin layer model for air gaps requires the duplication of the scalar potential to create a field discontinuity beyond the dimension of the sheet object and correctly represent the fringing flux effects. The Maxwell solver automatically duplicates the potential in one layer of elements from the gap edges. However, during the mesh adaptive process, the lateral elements may become too small and thus too close to the gap sides. This results in a low convergence of the mesh adaptive steps as the magnetic energy on such elements becomes more important. To improve convergence, the user can extend the sheet object manually (see figure below). Better convergence is observed by raising its lateral sides from one to three times the air gap thickness ($\delta \cong 3e$) depending on the model geometry.



Assigning a Resistive Sheet Boundary for the 3D Magnetostatic Solver

> **Note:** For transient designs, refer to Assigning a Resistive Sheet Boundary for the Transient Solver.
> For AC Magnetic designs, refer to Assigning a Resistive Sheet Boundary for the AC Magnetic Solver.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Maxwell 3D magnetostatic solution can include a nonlinear resistive sheet boundary condition. The linear resistive sheet boundary condition is equivalent to a resistive sheet in the AC magnetic solver and the transient solver (see links above). The nonlinear relationship between conductivity σ and current density J is as follows:

$$\sigma\left(J\right) = L\frac{c+J^d}{a+bJ^{d-1}}$$

where L is the resistive sheet thickness, and a, b, c, and d are constants. For electric arc simulation, the nonlinear relationship is applied to both the anode and the cathode, so a total of eight constants are required – four for the anode and four for the cathode.

The ohmic loss on the nonlinear resistive sheet can be calculated using

$$P = \lim_{L\to 0}\int_V \frac{J^2}{\sigma(\mathrm{J})}dV = \lim_{L\to 0}\int_S \frac{J^2}{f(\mathrm{J})}dS$$

where V is the resistive sheet volume, S is the resistive sheet surface, and

$$\left(J\right) = \frac{c+J^d}{a+bJ^{d-1}}$$

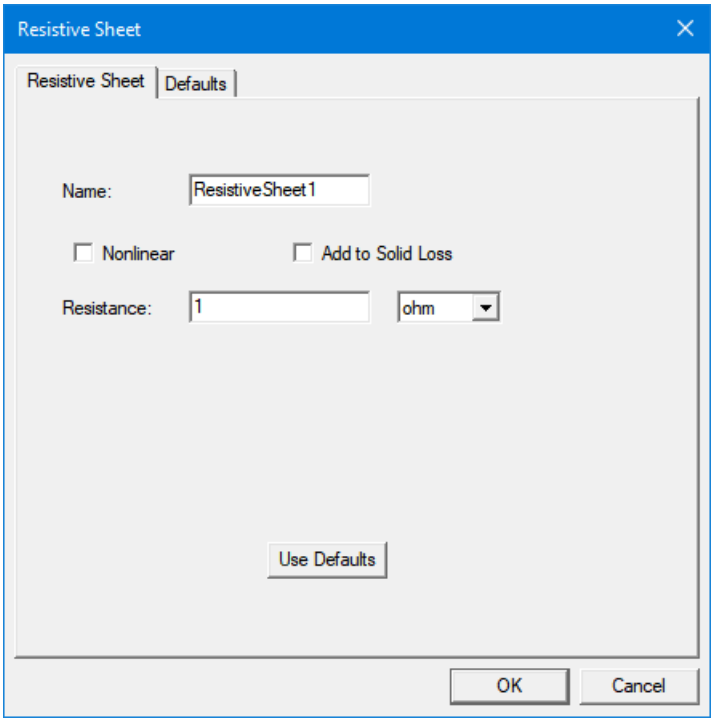
To define a resistive sheet boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a sheet object or face of a 3D conducting body touching another conducting body or face of a 3D conducting body touching the Region).

   > **Note:**
   > - The sheet must be completely inside a conductor or on its surface.
   > - The sheet must have conductors touching both of its faces.
   > - The resistance value cannot be dependent on an intrinsic variable.

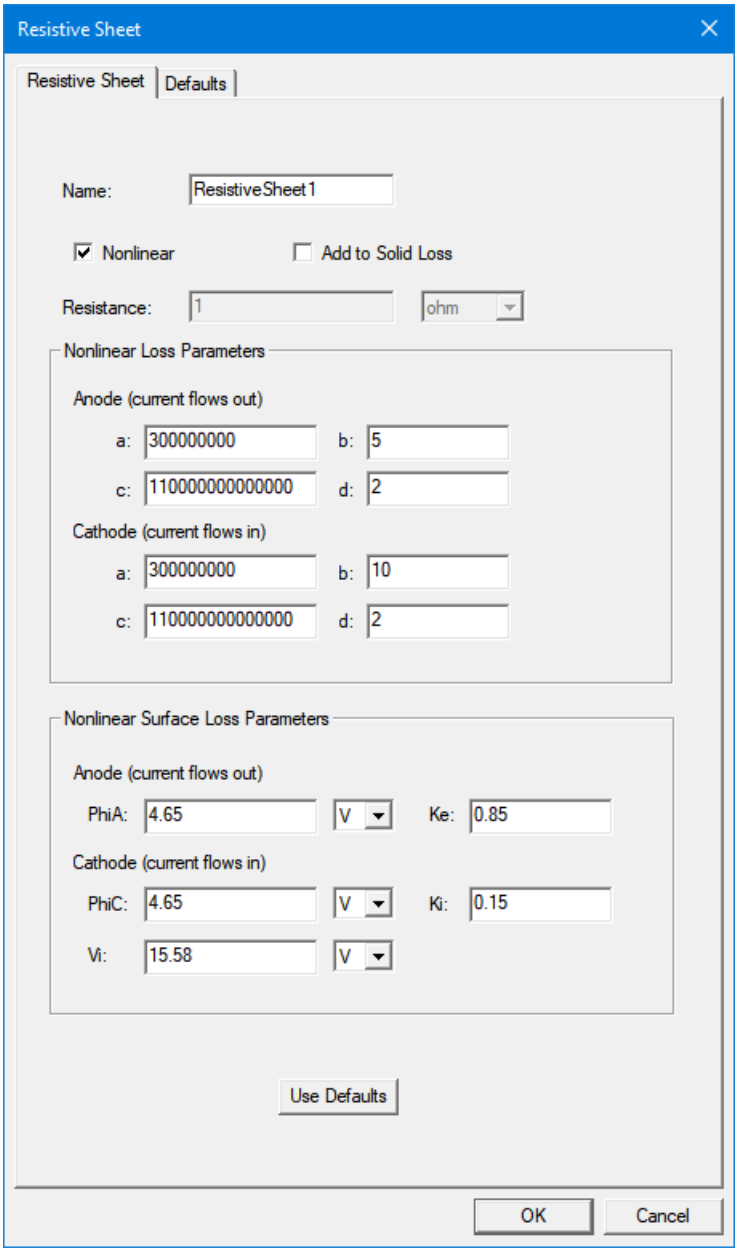
2. Click **Maxwell 3D** > **Boundaries** > **Assign** > **Resistive Sheet** to open the **Resistive Sheet** dialog box. Alternatively, you can select the sheet/surface, right-click, select **Assign Boundary** > **Resistive Sheet**.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. If **Nonlinear** is unchecked, enter a constant resistance value in the **Resistance** field, and select a unit of measure. Use of variables is supported. Constant resistance cannot be a negative value.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. If **Nonlinear** is checked, additional fields display for entering **Anode** and **Cathode** values. These values correspond to the *a*, *b*, *c*, and *d* coefficients in the equations above. The *a* and *b* values cannot both be 0.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Nonlinear Surface Loss Parameters** also become active when the **Nonlinear** option is checked. These parameters are used to calculate the surface loss on the specified boundary.

The surface loss is calculated using the following formulas:

$$\textbf{Cathode}: \ Q_c = k_i P + k_i J(V_i - \phi_c)$$

$$\textbf{Anode}: \ Q_a = k_e P + k_e J \phi_a$$

User can input all five coefficients. The default values are

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- k_i = 0.15
- k_e=0.85
- V_i = 15.58 V
- ϕ_c = 4.65 V
- ϕ_a = 4.65 V

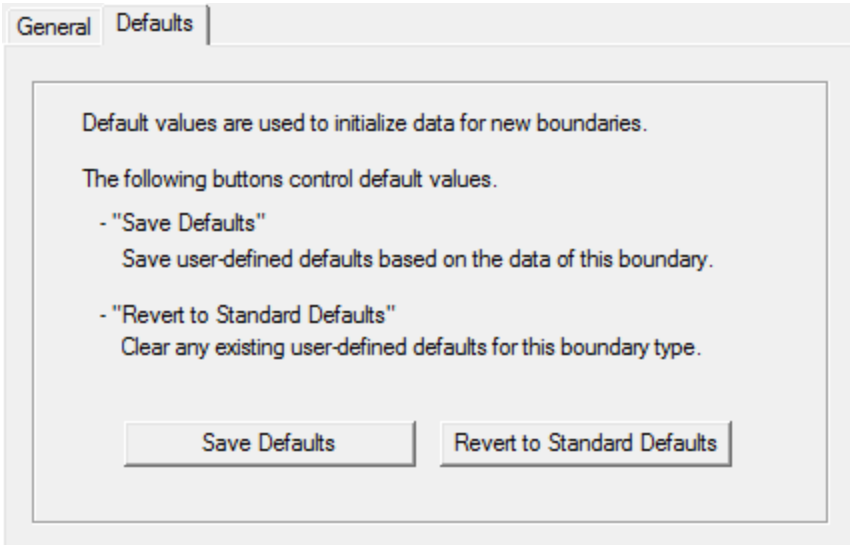
For copper material:

- k_i is the ion current ratio at the cathode, and k_e is the electron current at anode; these values are usually determined experimentally.
- V_i is the gas ionization potential.
- ϕ_c is copper material work function (material property) at the cathode.
- ϕ_a is copper material work function (material property) at the anode.

This surface loss is available in the Surface Loss Density plot in Maxwell and is output to System Coupling for electric arc simulation.

6. **Add to Solid Loss**: When this is checked, the resistive sheet boundary loss is added to the Solid Loss quantity that is available in the Maxwell application and used to evaluate electric arc voltage drop for the Twin Builder FMU. Note that the sheet loss is not added to the Volume Loss quantity that is output to System Coupling. This option applies to both linear and nonlinear resistive sheet boundaries.
7. Optionally, click **Use Defaults** to revert to the default value (1 ohm) in the window.

   The **Defaults** tab allows you to control default values. The **Save Defaults** button saves the values currently defined on the **General** tab as the defaults to be assigned to new impedance boundaries. **Revert to Standard Defaults** clears existing user-defined values and replaces them with the standard default values.



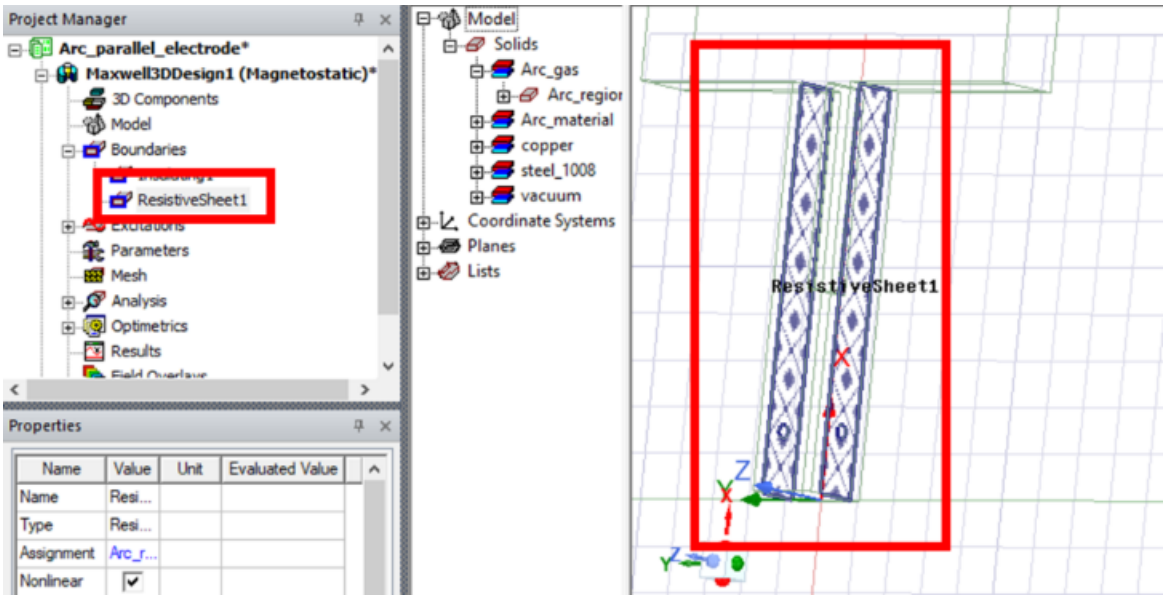
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Click **OK** to assign the boundary to the selected object.

   The Project Manager lists the newly assigned resistive sheet boundary in the tree. You can select the boundary in the tree to view and edit its properties in the Properties Window. You can also double-click the boundary entry in the tree to open it for editing in the **Resistive Sheet** dialog box.

> **Note:** When switching the solution type between transient and AC Magnetic, resistive sheet boundaries will be kept.

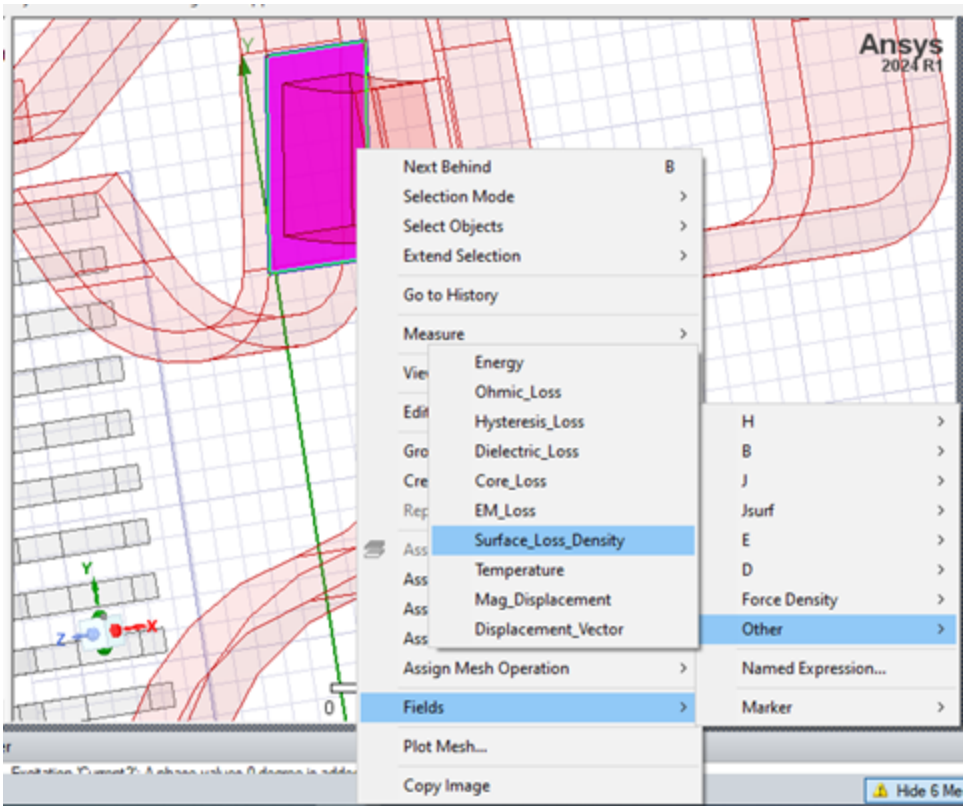
Note that multiple surfaces can be selected, and then one nonlinear resistive sheet boundary can be applied to all selected surfaces:



Resistive Sheet Output

The losses incurred due to a Resistive Sheet boundary are output as **Surface Loss Density**. These losses are counted as losses in solid conductors and are reported together with the losses in other solid conductors under Solid Loss. If the Surface Loss Density is mapped to **Mechanical**, it should be assigned to a face as Heat Flux. The Surface Loss Density plot is accessed from **Fields** > **Other**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Magnetostatic Excitations

The following excitations of magnetic fields are available for magnetostatic problems:

Excitation	Type of Excitation
Voltage	DC voltage on a surface or a sheet object
Voltage Drop	Voltage drop across a sheet object
Current Density	Known current density distribution in a conductor. In this case, the current density terminal must also be defined
Current	Total current in a conductor
Winding With Current	Current for both a stranded and solid conductor
Winding With Resistance Voltage Drop	Resistance Voltage Drop for both a stranded and solid conductor

In addition, permanent magnets serve as sources of magnetic fields.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Warning: If voltage sources are used as excitations, for current to flow, you must define a minimum of two voltage excitations or a voltage drop per conduction path. Each excitation must be set to a different voltage. Current flows from surfaces at higher voltages to surfaces at lower voltages.

Note: When a source conductor touches an outer boundary, Maxwell assumes current will flow perpendicular to the boundary. If you do not want current to cross the outer boundary, an insulating boundary should be used.

Assigning a Voltage Excitation for a Magnetostatic Solver

This type of excitation sets the voltage on a surface to a specific value. Use it to set up a voltage drop across a conduction path to cause current to flow.

To set a voltage excitation:

1. Select the section of the geometry on which you want to apply the excitation (typically a 2D object).
2. Click **Maxwell** > **Excitations** > **Assign** > **Voltage**.
   The **Voltage Excitation** window appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, enter a value for the voltage in the **Value** box, and select the units from the drop-down menu.
5. Optionally, click **Use Defaults** to revert to the default values in the window.
6. Click **OK** to assign the excitation to the selected object.

Warning: For current to flow, you must define a minimum of two voltage excitations or a voltage drop. Each excitation must be set to a different voltage. Current flows from surfaces at higher voltages to surfaces at lower voltages.

Assigning a Voltage Drop Excitation for a Magnetostatic Solver

This type of excitation sets the voltage drop across a sheet object to a specific value. The voltage drop applies only to sheet objects:

To set a voltage drop:

1. Select the section of the geometry on which you want to apply the excitation (typically a 2D object).
2. Click **Maxwell** > **Excitations** > **Assign** > **Voltage Drop**.
   The **Voltage Drop Excitation** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the voltage drop on the surface in the **Value** box, and select the units from the drop-down menu.
5. To change the direction of the voltage drop, click **Swap Direction**.
6. Optionally, click **Use Defaults** to revert to the default values in the window.
7. Click **OK** to assign the excitation to the selected object.

Assigning a Current Density Excitation for a Magnetostatic Solver

This command specifies the x-, y-, and z-components of the current density in a conduction path. If the current density is a function of position, the value is entered in ampere/m^2, even if you change the units in the problem.

To define the current density:

1. Select the section of the geometry (*i.e.*, the conductor) on which you want to apply the excitation (typically a 3D object).
2. Click **Maxwell 3D** > **Excitations** > **Assign** > **Current Density**.
   The **Current Density Excitation** window appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, specify the following options:
   a. Enter the **X Component**, **Y Component**, and **Z Component** values to define the respective components of the current density vector.
   b. Do the following in the two **Coordinate System** drop-down menus:
      - From the first drop-down menu, select either **Global** or **RelativeCSx** (where x = 1,2,3… if any relative coordinate systems have been defined).
      - From the second drop-down menu, select **Cartesian**, **Cylindrical**, or **Spherical**.
5. Optionally, click **Use Defaults** to revert to the default values in the window.
6. Click **OK** to assign the excitation to the selected object.

To complete the current density definition, you also need to specify any related current density terminals, which must be assigned to a 2D object or face of the respective conduction path.

To define a current density terminal:

1. Select the 2D object or face to which to assign the terminal.
2. Click **Maxwell** > **Excitations** > **Assign** > **Current Density Terminal**.
   The **Current Density Terminal Excitation** window appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. Click **OK** to assign the excitation to the selected object.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning a Current Excitation for a Magnetostatic Solver

Specifies the total current in a conduction path. The conduction path may be contained completely within the problem region (for example, a coil), or may touch the edges of the problem region.

To set a current excitation:

1. Select the section of the geometry (i.e., the conductor) on which you want to apply the excitation (typically a face or other 2D planar object).
2. Click **Maxwell 3D** > **Excitations** > **Assign** > **Current**.
   The **Current Excitation** window appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, specify the following options:
   a. Enter a value for the current in the **Value** box, and select the units from the drop-down menu.
   b. Select **Solid** or **Stranded** as the **Type** of conductor. For a stranded conductor, a uniformly distributed current density is assumed.
   c. Click **Swap Direction** to change the direction of the current flow.
5. Optionally, click **Use Defaults** to revert to the default values in the window.
6. Click **OK** to assign the excitation to the selected object.

Assigning a Winding Setup for a Magnetostatic Solver

> **Note:**
>
> - If you wish to define a winding for an AC Magnetic solution type, refer to Assigning a Winding Setup for an AC Magnetic Solver.
> - If you wish to define a winding for a Transient solution type, refer to Assigning a Winding Setup for a Transient Solver.

You can use the predefined coil terminal(s) to define one or more current or voltage windings.

To define a winding for your model:

1. Click **Maxwell 3D** > **Excitations** > **Add Winding**. to open the **Winding** dialog box.
2. Enter a name for the winding in the **Name** box, or accept the default.
3. In the **Parameters** section:
   a. Select **Current**, or **Resistance Voltage Drop** from the **Type** drop-down menu.
   b. Select the **Solid** or **Stranded** radio button to specify the type of conductor.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Enter values in the following fields (enabled according to winding type), and select the desired units.

For a current winding	For a resistance voltage drop winding[1]
Current	Resistance
	Voltage

> **Note:** You can also type a function as an expression for any of these fields.

d. Enter a value in the **Number of parallel branches** text box.

> **Note:** Parallel branches for solid windings assume there are no circulating currents in the parallel branches.

4. Optionally, click **Use Defaults** to revert to the default values in the window.
5. Click **OK** to assign the excitation to the selected object.

A winding is now defined for your model.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: To add a coil terminal to this winding, do the following:

1. In the project tree, right-click the winding, and select **Add Terminals**.
   The **Add Terminals** dialog box appears, listing all coil terminals that do not already belong to that winding.
2. Select the coil terminal (s) you want to add. To select multiple terminals, press **Ctrl** and click each terminal.
3. Click **OK**.

To assign a coil terminal excitation and add it to this winding, do the following:

1. Select the section of the geometry on which you want to apply the coil terminal excitation.
2. In the project tree, right-click the winding, and select **Assign Coil Terminal**.
   The **Coil Terminal Excitation** dialog box appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the number of conductors for the coil in the **Number of Conductors** box.

   Note: This value represents only the number of conductors inside of the selected geometry. If the Coil Terminal is cut due to symmetry, then only enter the Number of Conductors in the portion modeled.

5. Click **Swap Direction** to change the reference direction for the coil terminal. You can also change this direction in the coil terminal's Properties window by selecting either **Point into terminal**, or **Point out of terminal** for the **Direction** parameter.
6. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
7. Click **OK** to assign the coil terminal excitation. The coil terminal excitation is assigned and is added to the winding.

To delete all coil terminal excitations that belong to this winding, do the following:

- In the project tree, right-click the winding, and select **Delete All Terminals**.
  All coil terminal excitations are removed from the winding and deleted from the excitations.

---

1. For a solid winding, the resistance term can represent: the resistance of a portion of the winding, which is not modeled (for instance, end-effects), the leads connecting the winding to the source, or the source resistance. (The main winding resistance is calculated directly by the solver.) For a stranded winding, the resistance term is the complete DC resistance of the winding (since the solver does not determine resistance of a stranded winding) as well as the resistance of the end-effects, leads, source, etc. For both solid and stranded windings, the inductance term can represent: the extra inductance for a portion of the winding of the winding, which is not modeled (for instance, end-effects),

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the leads connecting the winding to the source, or the source inductance. (The main winding inductance itself is calculated directly by the solver.)

Assigning a Coil Terminal for a Magnetostatic Solver

To assign a coil terminal as a magnetostatic excitation:

1. Select the section of the geometry on which you want to apply the excitation (typically a 2D planar object).
2. Click **Maxwell** > **Excitations** > **Assign** > **Coil Terminal**. (You can also right-click an existing winding in the project tree, and select **Assign Coil Terminal**.)
3. In the **Coil Terminal Excitation** dialog box that appears, enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the number of conductors for the coil terminal in the **Number of Conductors** box.

   > **Note:** This value represents only the number of conductors inside of the selected geometry. If the Coil Terminal is cut due to symmetry, then only enter the Number of Conductors in the portion modeled.

5. If you want to switch the direction for the coil terminal, click **Swap Direction**.

   > **Note:** The direction is shown in the model by a bold red arrow.

6. Optionally, click **Use Defaults** to revert to the default values in the window.
7. Click **OK** to assign the coil terminal to the selected object.

> **Note:**
>
> To add this coil terminal to an existing winding:
>
> 1. In the project tree, right-click the coil terminal, and select **Add to Winding**.
> 2. In the **Add to Winding** dialog box, select the winding to which you want to add the coil terminal, and click **OK**.

> **Note:** To remove a coil terminal from an existing winding: In the project tree, right-click the coil terminal, and select **Remove from Winding**.
> The terminal is removed from the winding and moved up one level in the project tree (directly beneath **Excitations**).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Electrostatic Boundaries and Excitations

Specify at least one of the following excitations as a source of electric fields:

- The charge on a surface or object
- The volume charge density inside an object
- The voltage difference between conducting surfaces and objects. Define the electric potential on each surface and object using a voltage excitation.

Include at least one of the following as a reference for computing the electric potential:

- A voltage excitation
- An odd symmetry (flux normal) boundary

> **Note:** If a conductor is not assigned any excitation, it is automatically assigned a floating excitation with zero charge.

Electrostatic Boundaries

The following boundary conditions are available for 3D electrostatic problems:

Boundary Type	E-Field Behavior	Used to model...
Default Boundary Conditions (Natural and Neumann)	Field behaves as follows: • Natural boundaries – The normal component of **D** changes by the amount of surface charge density. No special conditions are imposed. • Neumann boundaries – **E** is tangential to the boundary. Flux cannot cross a Neumann boundary.	Ordinary E-field behavior on boundaries. Object interfaces are initially set to natural boundaries; outer boundaries are initially set to Neumann boundaries.
Symmetry	Field behaves as follows: • Even Symmetry (Flux Tangential) – **E** is tangential to the boundary; its normal components are zero. • Odd Symmetry (Flux Normal) – **E** is normal to the boundary; its tangential components are zero.	Planes of geometric and electrical symmetry.
Matching	The E-field on the dependent	Planes of symmetry in periodic

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(Independent and Dependent)	boundary is forced to match the magnitude and direction (or the negative of the direction) of the E-field on the independent boundary.	structures where **E** is oblique to the boundary.
Insulating	E-fields can be discontinuous across the insulating boundary.	Thin layer with two input values: **relative permittivity** and **thickness** of the thin layer.

Default Boundary Conditions for an Electrostatic Solver

These boundary conditions are automatically defined for an electrostatic model:

- Natural boundaries are assigned to the surfaces between dielectrics.
- Neumann boundaries are assigned to the outside edges of the problem region.

To leave a surface set to its default boundary condition, do nothing. Deleted boundary conditions and excitations automatically revert to the default boundary conditions.

Assigning a Symmetry Boundary Condition for an Electrostatic Solver

This boundary condition defines a plane of geometric or electric symmetry in a structure. Assign it only to the outer surfaces of the problem region.

To set a symmetry boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).
2. Click **Maxwell** > **Boundaries** > **Assign** > **Symmetry**.
   The **Symmetry Boundary** window appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Select one of the following as the type of symmetry:

Odd (Flux Normal)	**E** is normal to the boundary; its tangential components are zero.
Even (Flux Tangential)	**E** is tangential to the boundary; its normal components are zero.

5. Optionally, click **Use Defaults** to revert to the default values in the window.
6. Click **OK** to assign the boundary to the selected object.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning an Independent Boundary for an Electrostatic Solver

Independent and dependent boundaries enable you to model planes of periodicity where the E-field at every point on the dependent boundary surface is forced to match the E-field of every corresponding point on the independent boundary surface. The transformation used to map the E-field from the independent to the dependent is determined by specifying a coordinate system on both the independent and dependent boundaries.

To set an independent boundary:

1. Select the face to which you want to assign the independent boundary.
2. Click **Maxwell** > **Boundaries** > **Assign** > **Matching** > **Independent**.
   The **Independent Boundary** window appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. In the **Coordinate System** section, do the following to define the coordinate system:

   > **Note:** You must specify the coordinate system in the plane on which the boundary exists. First draw the U vector of the coordinate system. Maxwell uses the U vector you draw and the normal vector of the boundary face to calculate the v-axis. Then specify the direction of the V vector.

   a. Select **New Vector** from the **U Vector** drop-down menu.
      The **Create Line** message appears, asking you to draw the U vector of the coordinate system in the plane of the selected face, and the **Independent Boundary** dialog box disappears while you draw the U vector.
   b. Select the U vector's origin, which must be on the boundary's surface, in one of the following ways:
      - Click the point.
      - Type the point's coordinates in the in the **X**, **Y**, and **Z** boxes.
   c. Select a point on the u-axis.
      The **Independent Boundary** dialog box reappears
5. To reverse the direction of the vector, select the **Reverse Direction** check box.
6. Click **OK**.

The Maxwell solver computes the H-field on this boundary and maps it to the dependent boundary using the transformation defined by the independent and dependent coordinate systems.

Assigning a Dependent Boundary for an Electrostatic Solver

Assigning a dependent boundary is the second step in creating matching boundaries. The field on the Independent boundary is mapped to the dependent boundary.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** You must define an Independent boundary before creating the dependent boundaries that are associated with it. Assign dependent boundaries only to the outer surfaces of the problem region.

To set a dependent boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).
2. Click **Maxwell** > **Boundaries** > **Assign** > **Matching** > **Dependent**.
   The **Dependent Boundary** window appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Select the independent boundary to which the dependent is to be assigned from the **Independent** drop-down menu.
5. To specify the coordinate system, do the following in the Coordinate System section:

   > **Note:** You must specify the coordinate system in the plane on which the boundary exists. First draw the U vector of the coordinate system. Maxwell uses the U vector you draw and the normal vector of the boundary face to calculate the v-axis. Then specify the direction of the V vector.

   a. Select **New Vector** from the **U Vector** drop-down menu.
      The **Create Line** message appears asking you to draw the U vector of the coordinate system in the plane of the selected face, and the **Dependent Boundary** dialog box disappears while you draw the U vector.
   b. Select the U vector's origin in one of the following ways:
      - Click the point.
      - Type the point's coordinates in the in the **X**, **Y**, and **Z** boxes.

      > **Note:** The U vector's origin must be on the boundary's surface.

   c. Select a point on the u-axis in the same way.
      When the second point is identified, the vector is set, and the **Dependent Boundary** window reappears, with **Defined** listed in the **U Vector** drop-down menu.
   d. To specify the direction of the V vector, select or clear the **Reverse Direction** check box.
6. In the **Relation** section, select one of the following to specify the field behavior on the boundary:

   Edep = Click this radio button if the dependent

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Eind	and independent boundaries have the same magnitude and direction.
Edep = -Eind	Click this radio button if the dependent boundary field has the same magnitude as but the opposite direction from the independent boundary field.

7. Optionally, click **Use Defaults** to revert to the default values in the window.
8. Click **OK** to assign the boundary to the selected object.

> **Note:** The origin must be a vertex point of one of the objects.

Assigning an Insulating Boundary for an Electrostatic Solver

To set an insulating boundary:

1. Select the geometry face on which you want to apply the insulating boundary condition.
2. Click **Maxwell 3D** > **Boundaries** > **Assign** > **Insulating**.

   The **Insulating Boundary** dialog box appears.
3. On the **Insulating Boundary** tab, enter a name for the boundary in the **Name** box, or accept the default.
4. The **Parameters** section of the **Insulating Boundary** tab allows you to set the relative permittivity and thickness of the insulating boundary. Values can be defined using variables and also support Optimetrics.
   a. Enter the desired value for **Relative Permittivity**, or accept the default value of 1.
   b. Enter the desired value for **Thickness**, or accept the default value of 0.

   Note that changing data values will invalidate the solutions.
5. Optionally, click **Use Defaults** to revert to the default values in the window.
6. Click **OK** to assign the boundary to the selected object.

Electrostatic Excitations

The following excitations are available for electrostatic problems:

Excitation	Type of Excitation
Voltage	The DC voltage on a surface or object. This excitation type is available for both electrostatic and DC conduction Electric solvers.
Charge	The total charge on a surface or object (dielectric). This excitation type is available for electrostatic solvers only.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Floating	Used to model conductors at unknown potentials. This excitation type is available for electrostatic solvers only.
Volume Charge Density	The volume charge density in an object. This excitation type is available for electrostatic solvers only.

> **Note:** If a conductor is not assigned any excitation, it is automatically assigned a floating excitation with zero charge.

Assigning a Voltage Excitation for an Electrostatic Solver

Voltage excitations are identical to voltage boundaries in previous versions of Maxwell. Maxwell no longer has voltage boundaries in the electric solvers. Instead, a voltage excitation can be assigned to any surfaces, 2D objects, or solids. These assignments could include spatial functions. This type of excitation sets the electric potential (voltage) on a surface to a specific value.

> **Note:** When calculating capacitance, a voltage source should be assigned to an entire object, not just a single face located on that object.

To set a voltage excitation:

1. Select the section of the geometry on which you want to apply the excitation.
2. Click **Maxwell** > **Excitations** > **Assign** > **Voltage**.
   The **Voltage Excitation** window appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the electric potential in the **Value** box, and select the units from the drop-down menu. You can enter a numerical value or a spatial function.
5. For a spatial function, select a coordinate system from the **Coordinate System** drop-down menu.
6. Optionally, click **Use Defaults** to revert to the default values in the window.
7. Click **OK** to assign the excitation to the selected object.

Assigning a Charge Excitation for an Electrostatic Solver

This type of excitation defines the total charge on a surface or object. The potential on the charge is computed during the solution.

To define a charge excitation:

1. Select the section of the geometry on which you want to apply the excitation.
2. Click **Maxwell** > **Excitations** > **Assign** > **Charge**.
   The **Charge Excitation** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the charge in the **Value** box.
5. Optionally, click **Use Defaults** to revert to the default values in the window.
6. Click **OK** to assign the excitation to the selected object.

Assigning a Floating Excitation for an Electrostatic Solver

This type of excitation models conductors at unknown potentials and specifies the total charge on the conductor.

To define a floating conductor:

1. Select the section of the geometry on which you want to apply the excitation. This can be a 2D surface or a 3D solid.
2. Click **Maxwell** > **Excitations** > **Assign** > **Floating**.
   The **Floating Excitation** window appears.
3. Click the **General** tab.
4. Enter a name for the excitation in the **Name** box, or accept the default.
5. In the **Parameters** section, enter the charge on the boundary in the **Value** box.
6. Optionally, click **Use Defaults** to revert to the default values in the window. You can also click the **Defaults** tab to save new default settings or to revert to the standard default values.
7. Click **OK** to assign the excitation to the selected object.

Assigning a Volume Charge Density Excitation for an Electrostatic Solver

This type of excitation defines the volume charge density on an object.

To define the charge density on a 3D object:

1. Select the 3D section of the geometry on which you want to apply the excitation.
2. Click **Maxwell** > **Excitations** > **Assign** > **Volume Charge Density**.
   The **Volume Charge Density Excitation** window appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the charge density in the **Value** box.
5. In the **Parameters** section, select the type of coordinate system from the **Coordinate System** drop-down menu.
6. Optionally, click **Use Defaults** to revert to the default values in the window.
7. Click **OK** to assign the excitation to the selected object.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3D AC Conduction Boundaries and Excitations

The following boundaries can be specified for this solver:

- Symmetry
- (Matching) Independent/Dependent
- Thin Layer
- Insulating

The following excitations can be specified for this solver:

- Voltage
- Current
- Sink

Include at least one of the following as a reference for computing the electric potential:

- A voltage excitation (with a potential difference)
- An odd symmetry (flux normal) boundary
- A sink excitation

3D AC Conduction Boundaries

The following boundary conditions are available for 3D AC conduction electric problems:

Boundary Type	E-Field Behavior	Used to model...
Default Boundary Conditions (Natural and Neumann)	Field behaves as follows: • Natural boundaries - The normal and tangent components of **J** and **E** are continuous across discontinuity surfaces. • Neumann boundaries - **E** and **J** vectors are tangential to the boundary. Flux cannot cross a Neumann boundary.	Object interfaces are initially set to natural boundaries; outer boundaries are initially set to Neumann boundaries.
Symmetry	Field behaves as follows: • Even Symmetry (Flux Tangential) - **E and J vectors are** tangential to the boundary; its normal components are zero. • Odd Symmetry (Flux Normal) - **E and J vectors are** normal to the boundary; its tangential components are zero.	Planes of geometric and electrical symmetry.
Matching	The E-field on the dependent boundary is forced to	Planes of symmetry

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(Independent and Dependent)	match the magnitude of the E-field on the independent boundary if the plus sign is used. If the minus sign is used, the field on the dependent boundary will oscillate with opposite phase angle (180 degrees phase shift).	in periodic structures where **E** is oblique to the boundary.
Insulating	The current cannot cross the boundary.	Thin, perfectly insulating sheets between touching conductive domains.
Thin Layer	When two parts of the domain are separated by thin volume, it can be modeled by a Thin Layer. This boundary creates a field discontinuity (voltage drop) between the two sides of the selected surface based on the material bulk properties and thickness.	Thin and non-perfect (possibly lossy) domains.

Default Boundary Conditions for a 3D AC Conduction Solver

These boundary conditions are automatically defined for a AC conduction model:

- Natural boundaries are assigned to the surfaces between objects. The normal component of **J** and the tangential component of **E** are continuous across discontinuity surfaces.
- Neumann boundaries are assigned to the outside edges of the problem region. **E** and **J** vectors are tangential to the boundary. Flux cannot cross a Neumann boundary.

To leave a surface set to its default boundary condition, do nothing. Deleted boundary conditions and excitations automatically revert to the default boundary conditions.

Assigning a Symmetry Boundary Condition for a 3D AC Conduction Solver

This boundary condition defines a plane of geometric or electric symmetry in a structure. Assign it only to the outer surfaces of the problem region.

To set a symmetry boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).
2. Click **Maxwell 3D** > **Boundaries** > **Assign** > **Symmetry**.

   The **Symmetry Boundary** dialog box appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Select one of the following as the type of symmetry:

   Odd Voltages have opposite sign on the two sides of the symmetry

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(Flux Normal)	plane.
Even (Flux Tangential)	Voltages have same sign on the two sides of the symmetry plane.

5. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
6. Click **OK** to assign the boundary to the selected object.

Assigning an Independent Boundary for a 3D AC Conduction Solver

Independent and dependent boundaries enable you to model planes of periodicity where the **E**-field at every point on the dependent boundary surface is forced to match the **E**-field of every corresponding point on the independent boundary surface. The transformation used to map the E-field from the independent to the dependent is determined by specifying a coordinate system on both the independent and dependent boundaries.

To set an independent boundary:

1. Select the face to which you want to assign the independent boundary.
2. Click **Maxwell 3D** > **Boundaries** > **Assign** > **Matching** > **Independent**.

   The **Independent Boundary** window appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. In the **Coordinate System** section, do the following to define the coordinate system:

   > **Note:** You must specify the coordinate system in the plane on which the boundary exists. First draw the U vector of the coordinate system. Maxwell uses the U vector you draw and the normal vector of the boundary face to calculate the V-axis. Then specify the direction of the V vector.

   a. Select **New Vector** from the **U Vector** drop-down menu.
   The **Create Line** message appears, asking you to draw the U vector of the coordinate system in the plane of the selected face, and the **Independent Boundary** dialog box disappears while you draw the U vector.
   b. Select the U vector's origin, which must be on the boundary's surface, in one of the following ways:
      - Click the point.
      - Type the point's coordinates in the in the **X**, **Y**, and **Z** boxes.
   c. Select a point on the U-axis.

      The **Independent Boundary** dialog box reappears

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. To reverse the direction of the vector, select the **Reverse Direction** check box.
6. Click **OK**.

The Maxwell solver computes the E-field on this boundary and maps it to the dependent boundary.

Assigning a Dependent Boundary for a 3D AC Conduction Solver

Assigning a dependent boundary is the second step in creating matching boundaries. The field on the Independent boundary is mapped to the dependent boundary.

> **Note:** You must define an Independent boundary before creating the dependent boundaries that are associated with it. Assign dependent boundaries only to the outer surfaces of the problem region.

To set a dependent boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).
2. Click **Maxwell 3D** > **Boundaries** > **Assign** > **Matching** > **Dependent**.

   The **Dependent Boundary** window appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Select the independent boundary to which the dependent is to be assigned from the **Independent** drop-down menu.
5. To specify the coordinate system, do the following in the Coordinate System section:

   > **Note:** You must specify the coordinate system in the plane on which the boundary exists. First draw the U vector of the coordinate system. Maxwell uses the U vector you draw and the normal vector of the boundary face to calculate the v-axis. Then specify the direction of the V vector.

   a. Select **New Vector** from the **U Vector** drop-down menu.
   The **Create Line** message appears asking you to draw the U vector of the coordinate system in the plane of the selected face, and the **Dependent Boundary** dialog box disappears while you draw the U vector.

   b. Select the U vector's origin in one of the following ways:

      > **Note:** The U vector's origin must be on the boundary's surface.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click the point.
- Type the point's coordinates in the in the **X**, **Y**, and **Z** boxes.

c. Select a point on the u-axis in the same way.
When the second point is identified, the vector is set, and the **Dependent Boundary** window reappears, with **Defined** listed in the **U Vector** drop-down menu.

d. To specify the direction of the V vector, select or clear the **Reverse Direction** check box.

6. In the **Relation** section, select one of the following to specify the field behavior on the boundary:

Edep = Eind	Click this radio button if the dependent and independent boundaries have the same magnitude and direction.
Edep = -Eind	Click this radio button if the dependent boundary field has the same magnitude as but the opposite direction from the independent boundary field.

7. Optionally, click **Use Defaults** to revert to the default values in the window.
8. Click **OK** to assign the boundary to the selected object.

Note: The origin must be a vertex point of one of the objects.

Assigning an Insulating Boundary for the 3D AC Conduction Solver

This boundary condition is generally used to model very thin sheets of perfectly insulating material between touching conductors, as well as infinitely thin cracks inside conductors. Current cannot cross an insulating boundary, that is, $\boldsymbol{n} \cdot \boldsymbol{J} = \mathbf{0}$.

To set an insulating boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).
2. Click **Maxwell 3D** > **Boundaries** > **Assign** > **Insulating**.

   The **Insulating Boundary** window appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Click **OK** to assign the boundary to the selected object.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** If you apply the insulating boundary condition on sheet objects (inside conductors), always make sure that the respective entities are selected as faces and not as objects before attempting to assign the insulating boundary condition to them.

Assigning a Thin Layer Boundary for a 3D AC Conduction Solver

When two parts of the domain are separated by a thin volume, it can be modeled using a **Thin Layer** boundary condition. This boundary creates a field discontinuity (voltage drop) across the two sides of the surface based on the material bulk parameters and thickness.

This boundary condition can be used to model thin domains, for example: the contact resistance between two conductive domains, the capacitive effect of thin insulating materials or thin lossy domains in electronic boards without needing to explicitly model (and mesh) a 3D object having a finite thickness.

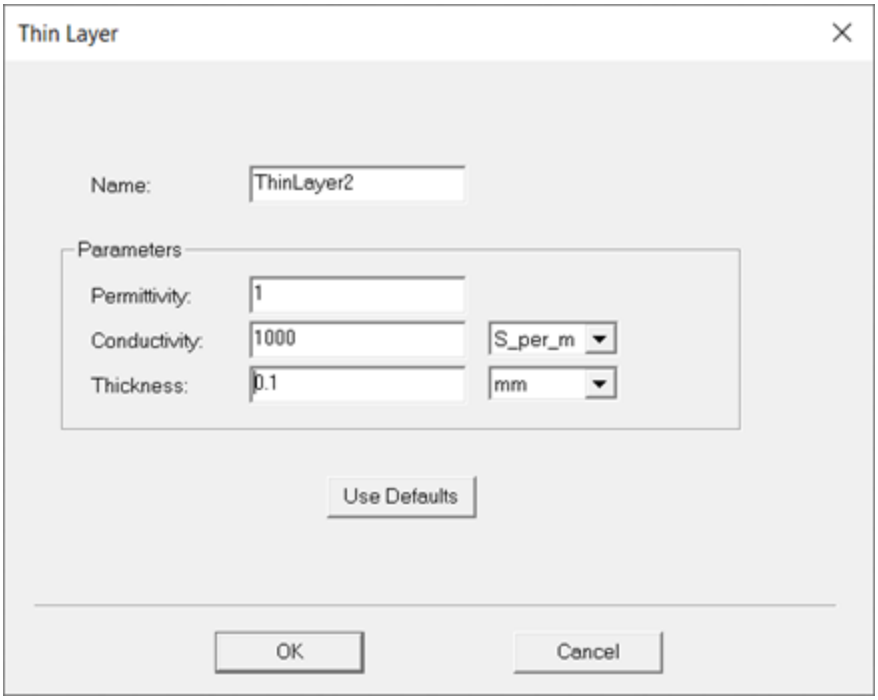
> **Note:** The thin layer can be only applied to a coincident surface shared by two different bodies. It cannot be applied to a 2D cross-sectional sheet inside of a single body.

For the 3D AC conduction solver, the thin layer is defined by its bulk conductivity, permittivity, and thickness.

To define a resistive sheet boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a sheet object or face of a 3D body).
2. Click **Maxwell 3D** > **Boundaries** > **Assign** > **Thin Layer** to open the Thin Layer dialog box.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Enter the thin layer parameters in the **Conductivity**, **Permittivity**, and **Thickness** fields, and select the units of measure. Use of variables is supported.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Optionally, click **Use Defaults** to revert to the default values in the window.

   The **Defaults** tab allows you to control default values. The **Save Defaults** button saves the values currently defined on the **General** tab as the defaults to be assigned to new boundaries. **Revert to Standard Defaults** clears existing user-defined values and replaces them with the standard default values.

6. Click **OK** to assign the boundary to the selected object.

The Project Manager lists the newly assigned thin layer boundary in the tree. You can select the boundary in the tree to view and edit its properties in the Properties Window. You can also double-click the boundary entry in the tree to open it for editing in the thin layer dialog box.

3D AC Conduction Excitations

The following excitations are available for 3D electric AC conduction problems:

Excitation	Type of Excitation
Voltage	The AC voltage on a surface or object.
Current	The current across a surface.
Sink	Applied only to surfaces which are grounded with reference potential value. It is the same as applying zero voltage excitation on those surfaces.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** All field quantities oscillate with the same frequency, equal to the frequency specified in the solution setup. The phase of quantities computed in the solution space however will be different in general.

Assigning a Voltage Excitation for a 3D AC Conduction Solver

This excitation sets the electric potential (voltage) on a surface or object to a specific value.

To set a voltage excitation:

1. Select the section of the geometry on which you want to apply the excitation.
2. Click **Maxwell 3D** > **Excitations** > **Assign** > **Voltage**.
   The **Voltage Excitation** dialog box appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section:
   a. Enter the electric potential in the **Value** box, and select the unit from the drop-down menu. You can enter a numerical value or a spatial function. For a spatial function, select a coordinate system from the **Coordinate System** drop-down menu.
   b. Enter a value in the **Phase** text box, and select a unit from the drop-down menu.
5. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
6. Click **OK** to assign the excitation to the selected object.

Assigning a Current Excitation for a 3D AC Conduction Solver

To define a current across a surface:

1. Select the 2D section of the geometry on which you want to apply the excitation.
2. Click **Maxwell 3D** > **Excitations** > **Assign** > **Current**.
   The **Current Excitation** window appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section:
   a. Enter the current strength in the **Value** box, and select the unit from the drop-down menu.
   b. Enter the phase angle in the **Phase** box, and select the unit from the drop-down menu.
   c. Select either **Solid** or **Stranded** to define the **Type** for the conductor. For a stranded conductor, a uniformly distributed current density is assumed without considering eddy effects.
   d. Optionally, to change the direction of the current flow, click **Ref. Direction**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Optionally, click **Use Defaults** to revert to the default values in the window.
6. Click **OK** to assign the excitation to the selected object.

Assigning a Sink Excitation for a 3D AC Conduction Solver

To define a sink excitation on a surface:

1. Select the section of the geometry on which you want to apply the excitation (typically a 2D object).
2. Click **Maxwell 3D** > **Excitations** > **Assign** > **Sink**.
   The **Sink Excitation** window appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. Click **OK** to assign the excitation to the selected object.

DC Conduction Boundaries and Excitations

The following excitations can be specified as the source of electric fields:

- The voltage difference between surfaces and objects. Define the electric potential on each surface and object using a voltage excitation.
- Using the current excitation, define the total current flowing through a surface or a 2D object.
- Using the sink excitation, define a collection of surfaces or 2D objects as a sink. Sink excitations are applied when only current excitations are defined in a conduction path and there is no voltage excitation. In fact, in this case, a sink excitation has to be defined and it ensures that the total current flowing through the outside surface of a conduction path is exactly zero.

Include at least one of the following as a reference for computing the electric potential:

- A voltage excitation.
- A sink excitation.
- An odd symmetry (flux normal) boundary.

DC Conduction Boundaries

The following boundary conditions are available for DC conduction electric problems:

Boundary Type	E-Field Behavior	Used to model...
Default Boundary Conditions (Natural and	Field behaves as follows: • Natural boundaries — The normal component of **D** changes by the amount of surface charge density. No special	Ordinary E-field behavior on boundaries. Object interfaces are initially set to natural boundaries; outer

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Neumann)	conditions are imposed. • Neumann boundaries — **E** is tangential to the boundary. Flux cannot cross a Neumann boundary.	boundaries are initially set to Neumann boundaries.
Insulating	Same as Neumann, except that current cannot cross the boundary. An insulating boundary is *only* available for electrostatic solutions that include a DC conduction analysis.	Thin, perfectly insulating sheets between touching conductors.
Symmetry	Field behaves as follows: • Even Symmetry (Flux Tangential) — **E** is tangential to the boundary; its normal components are zero. • Odd Symmetry (Flux Normal) — **E** is normal to the boundary; its tangential components are zero.	Planes of geometric and electrical symmetry.
Matching (Independent and Dependent)	The E-field on the dependent boundary is forced to match the magnitude and direction (or the negative of the direction) of the E-field on the independent boundary.	Planes of symmetry in periodic structures where **E** is oblique to the boundary.
Thin Layer	When two parts of the domain are separated by a thin volume, it can be modeled by a Thin Layer. This boundary creates a field discontinuity (voltage drop) between the two sides of the selected surface based on the material bulk properties and thickness.	Thin conducting domains

Default Boundary Conditions for a DC Conduction Solver

These boundary conditions are automatically defined for a DC conduction model:

- Natural boundaries are assigned to the surfaces between objects.
- Neumann boundaries are assigned to the outside edges of the problem region.

To leave a surface set to its default boundary condition, do nothing. Deleted boundary conditions and excitations automatically revert to the default boundary conditions.

Assigning an Insulating Boundary for a DC Conduction Solver

This boundary condition is used to model very thin sheets of perfectly insulating material between touching conductors. Current cannot cross an insulating boundary.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

An insulating boundary is *only* available for electrostatic solutions that include a DC conduction analysis.

To set an insulating boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).
2. Click **Maxwell** > **Boundaries** > **Assign** > **Insulating**.

   The **Insulating Boundary** window appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Click **OK** to assign the boundary to the selected object.

Assigning a Symmetry Boundary Condition for a DC Conduction Solver

This boundary condition defines a plane of geometric or electric symmetry in a structure. Assign it only to the outer surfaces of the problem region.

To set a symmetry boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).
2. Click **Maxwell** > **Boundaries** > **Assign** > **Symmetry**.
   The **Symmetry Boundary** window appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Select one of the following as the type of symmetry:

Odd (Flux Normal)	Current flows in opposite directions on either side of the symmetry plane.
Even (Flux Tangential)	Current flows in the same direction on both sides of the symmetry plane.

5. Optionally, click **Use Defaults** to revert to the default values in the window.
6. Click **OK** to assign the boundary to the selected object.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Warning: When using even symmetry boundaries, double-check the following two points:

- Do not violate Ampere's law!
- All magnetic field boundaries must be connected to each other. Defining disconnected magnetic field boundaries and even symmetry boundaries can produce unexpected results, as there is no unique solution to such problems.

Assigning a Independent Boundary for a DC Conduction Solver

Independent and dependent boundaries enable you to model planes of periodicity where the H-field at every point on the dependent boundary surface is forced to match the H-field of every corresponding point on the independent boundary surface. The transformation used to map the H-field from the independent to the dependent is determined by specifying a coordinate system on both the independent and dependent boundaries.

To set an independent boundary:

1. Select the face to which you want to assign the independent boundary.
2. Click **Maxwell** > **Boundaries** > **Assign** > **Matching** > **Independent**.
   The **Independent Boundary** window appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. In the **Coordinate System** section, do the following to define the coordinate system:

   Note: You must specify the coordinate system in the plane on which the boundary exists. First draw the U vector of the coordinate system. Maxwell uses the U vector you draw and the normal vector of the boundary face to calculate the V-axis. Then specify the direction of the V vector.

   a. Select **New Vector** from the **U Vector** drop-down menu.
      The **Create Line** message appears, asking you to draw the U vector of the coordinate system in the plane of the selected face, and the **Independent Boundary** dialog box disappears while you draw the U vector.
   b. Select the U vector's origin, which must be on the boundary's surface, in one of the following ways:
      - Click the point.
      - Type the point's coordinates in the in the **X**, **Y**, and **Z** boxes.
   c. Select a point on the u-axis.
      The **Independent Boundary** dialog box reappears
5. To reverse the direction of the vector, select the **Reverse Direction** check box.
6. Click **OK**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Maxwell solver computes the E-field on this boundary and maps it to the dependent boundary using the transformation defined by the independent and dependent coordinate systems.

Assigning a Dependent Boundary for a DC Conduction Solver

Assigning a dependent boundary is the second step in creating matching boundaries. The field on the Independent boundary is mapped to the dependent boundary.

> **Note:** You must define a Independent boundary before creating the dependent boundaries that are associated with it. Assign dependent boundaries only to the outer surfaces of the problem region.

To set a dependent boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).
2. Click **Maxwell** > **Boundaries** > **Assign** > **Matching** > **Dependent**.
   The **Dependent Boundary** window appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Select the independent boundary to which the dependent is to be assigned from the **Independent** drop-down menu.
5. To specify the coordinate system, do the following in the Coordinate System section:

   > **Note:** You must specify the coordinate system in the plane on which the boundary exists. First draw the U vector of the coordinate system. Maxwell uses the U vector you draw and the normal vector of the boundary face to calculate the v-axis. Then specify the direction of the V vector.

   a. Select **New Vector** from the **U Vector** drop-down menu.
      The **Create Line** message appears asking you to draw the U vector of the coordinate system in the plane of the selected face, and the **Dependent Boundary** dialog box disappears while you draw the U vector.
   b. Select the U vector's origin in one of the following ways:

      > **Note:** The U vector's origin must be on the boundary's surface.

      - Click the point.
      - Type the point's coordinates in the in the **X**, **Y**, and **Z** boxes.
   c. Select a point on the u-axis in the same way.
      When the second point is identified, the vector is set, and the **Dependent**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Boundary window reappears, with **Defined** listed in the **U Vector** drop-down menu.

d. To specify the direction of the V vector, select or clear the **Reverse Direction** check box.

> **Note:** When entering current, current density terminals, or coil terminals, the arrow associated with current in the selected object shows the direction as if a positive value is entered for the current. If a negative value is entered the actual current flow direction is opposite to what the arrow shows. Click **Reverse Direction** if you want to reverse the direction of the arrow.

6. In the **Relation** section, select one of the following to specify the field behavior on the boundary:

Edep = Eind	Click this radio button if the dependent and independent boundaries have the same magnitude and direction.
Edep = -Eind	Click this radio button if the dependent boundary field has the same magnitude as but the opposite direction from the independent boundary field.

7. Optionally, click **Use Defaults** to revert to the default values in the window.
8. Click **OK** to assign the boundary to the selected object.

> **Note:** The origin must be a vertex point of one of the objects.

Assigning a Thin Layer Boundary for a 3D DC Conduction Solver

When two parts of the domain are separated by a thin volume, it can be modeled using a **Thin Layer** boundary condition. This boundary creates a field discontinuity (voltage drop) across the two sides of the surface based on the material bulk parameters and thickness.

This boundary condition can be used to model thin domains, for example, the contact resistance between two conductive domains without needing to explicitly model (and mesh) a 3D object having a finite thickness.

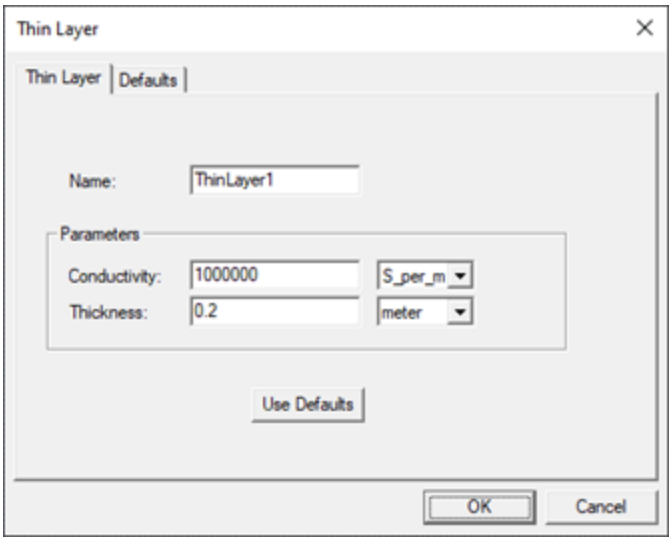
> **Note:** The thin layer can be only applied to a coincident surface shared by two different bodies. It cannot be applied to a 2D cross-sectional sheet inside of a single body.

For the 3D DC conduction solver, the thin layer is defined by its bulk conductivity, and thickness.

To define a thin layer boundary:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the section of the geometry on which you want to apply the boundary condition (typically a sheet object or face of a 3D body).
2. Click **Maxwell 3D** > **Boundaries** > **Assign** > **Thin Layer** to open the Thin Layer dialog box.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Enter the thin layer parameters in the **Conductivity** and **Thickness** fields, and select the units of measure. Use of variables is supported.



5. Optionally, click **Use Defaults** to revert to the default values in the window.

   The **Defaults** tab allows you to control default values. The **Save Defaults** button saves the values currently defined on the **Thin Layer** tab as the defaults to be assigned to new boundaries. **Revert to Standard Defaults** clears existing user-defined values and replaces them with the standard default values.

6. Click **OK** to assign the boundary to the selected object.

The Project Manager tree lists the newly assigned thin layer boundary in the tree. You can select the boundary in the tree to view and edit its properties in the Properties Window. You can also double-click the boundary entry in the tree to open it for editing in the thin layer dialog box.

DC Conduction Excitations

The following excitations are available for electric DC conduction problems:

Excitation	Type of Excitation
Voltage	The DC voltage on a surface or object. This excitation type is available for both electrostatic and DC conduction solvers.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Current	The current across a 2D object. This excitation type is available for DC conduction solvers only. A current excitation must be applied across a 2D surface.
Sink	This excitation type is available for DC conduction solvers only.

For a DC conduction solution, if a current excitation is assigned, at least one sink or voltage excitation must also be defined. However, you cannot define both a voltage excitation and a sink excitation at the same time.

Assigning a Voltage Excitation for a DC Conduction Solver

This type of excitation sets the electric potential (voltage) on a surface or object to a specific value.

To set a voltage excitation:

1. Select the section of the geometry on which you want to apply the excitation.
2. Click **Maxwell** > **Excitations** > **Assign** > **Voltage**.
   The **Voltage Excitation** window appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the electric potential in the **Value** box, and select the units from the drop-down menu. You can enter a numerical value or a spatial function.
5. For a spatial function, select a coordinate system from the **Coordinate System** drop-down menu.
6. Optionally, click **Use Defaults** to revert to the default values in the window.
7. Click **OK** to assign the excitation to the selected object.

Assigning a Current Excitation for a DC Conduction Solver

To define a current across a surface:

1. Select the 2D section of the geometry on which you want to apply the excitation.
2. Click **Maxwell 3D** > **Excitations** > **Assign** > **Current**.
   The **Current Excitation** window appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the current strength in the **Value** box.
5. Optionally, to change the direction of the current flow, click **Swap Direction**.
6. Optionally, click **Use Defaults** to revert to the default values in the window.
7. Click **OK** to assign the excitation to the selected object.

Assigning a Sink Excitation for a DC Conduction Solver

To define a sink excitation on a surface:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the section of the geometry on which you want to apply the excitation (typically a 2D object).
2. Click **Maxwell** > **Excitations** > **Assign** > **Sink**.
   The **Sink Excitation** window appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. Click **OK** to assign the excitation to the selected object.

DC Conduction + Electrostatic Boundaries and Excitations

In the case of a problem when you are interested in the electric solution in both the conductors and the insulators, you can define a **DC Conduction + Electrostatic** solution type. The Insulator/Conductor threshold defines which objects are conductors or insulators. The solver first solves the conduction problem in all conduction paths and then, using the calculated voltage on the conductors as a voltage excitation, computes the electrostatic field in the insulators. As a consequence, you can define the DC-conduction excitations and boundary conditions in the conducting regions and the electrostatic excitations and boundary conditions in the insulators.

AC Magnetic T-Omega Boundaries and Excitations

Specify at least one of the following as a source of AC magnetic fields in your model:

- The current in a conduction path, which can be either stranded or solid.

  > **Note:** For stranded conductors, it is assumed that the current density is uniform on the cross-section of the respective conductors. Thus, the stranded option for current excitations should mostly be used only in cases where the cross-section of conductors is constant, consistent with the assumption that the respective object is a coil built with strands of wire. For this reason, conductors (coils) could be created by either sweeping around an axis using zero segments or sweeping along a path having only true surface segments in order to create a smooth surface coil having a completely uniform cross-sectional area over its entire length.

- The current density in a conductor.
- The magnetic field on an outside surface.

If currents or current densities are the only sources of AC magnetic fields in your model, set at least one outer boundary to the following:

- The default boundary conditions.
- An odd symmetry (flux tangential) boundary.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- An even symmetry (flux normal) boundary.

AC Magnetic T-Omega Boundaries

The AC Magnetic T-Omega field solver allows you to define the following types of boundaries:

Boundary Type	H-Field Behavior	Used to model…
Default Boundary Conditions (Natural and Neumann)	Field behaves as follows: • Natural boundaries – **H** is continuous across the boundary. • Neumann boundaries – **H** is tangential to the boundary and flux cannot cross it.	Ordinary field behavior. Initially, object interfaces are natural boundaries; outer boundaries and excluded objects are Neumann boundaries.
Zero Tangential H Field	Flux is perpendicular if tangent components are zero or if a zero tangential H-field boundary condition was applied.	External AC magnetic fields.
Tangential H Field	The tangential components of **H** are set to predefined values.	External AC magnetic fields
Integrated Zero Tangential H Field	For applications such as a motor with the shaft excluded, there remains a hole in the middle. In such cases, on the hole's boundary, neither the Tangential H Field boundary (where integration of the tangential H field is non-zero), nor the Zero Tangential H Field boundary (where the tangential H field is zero everywhere) can be applied because – for the hole boundary case – integration of the tangential H field is zero, but the tangential H field is not zero everywhere. Integrated Zero Tangential H Field can be used for such applications.	Cases such as the boundary around the hole left when a motor's shaft is excluded.
Insulating	Same as Neumann, except that current cannot cross the boundary.	Perfectly insulating sheets between conductors.
Symmetry	Field behaves as follows: • Odd Symmetry (Flux Tangential) – **H** is tangential to the boundary; its normal components are zero.	Planes of geometric and magnetic symmetry.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Boundary Type	H-Field Behavior	Used to model…
	• Even Symmetry (Flux Normal) – **H** is normal to the boundary; its tangential components are zero.	
Matching (Independent and Dependent)	The H-field on the dependent boundary is forced to match the magnitude and direction (or the negative of the direction) of the H-field on the independent boundary.	Planes of symmetry in periodic structures where **H** is oblique to the boundary.
Radiation	No restrictions on the field behavior.	Unbounded eddy currents.
Impedance	Includes the effect of induced currents beyond the boundary surface.	Conductors with very small skin depths.
Resistive Sheet	When two conductors are in contact within a conduction path, a voltage drop normally exists across the contact surface due to the imperfection of the contact between the two conductors. This boundary condition assigns a resistive sheet to consider the impact of this kind of voltage drop. The resistive sheet is defined by a lumped resistance in ohms.	Voltage drop and loss on the contact surface between two different objects within a conduction path.
Thin Layer	This boundary creates a field discontinuity between the two sides of the selected surface based on its thickness and the given electric and magnetic material proprieties. When the air gap option is selected, the conductivity is assumed to be zero and the relative magnetic permeability is assumed to be equal to one.	Thin conductors and ferromagnets, and narrow air gaps

Default Boundary Conditions

F or the AC Magnetic T-Omega solver, these boundary conditions are automatically defined for an AC Magnetic model:

- Natural boundaries are assigned to the surfaces between objects.
- Neumann boundaries are assigned to the outside edges of the problem region.

To leave a surface set to its default boundary condition, do nothing. Deleted boundary conditions and excitations automatically reset to the default boundary conditions.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning a Magnetic H Field

This type of boundary defines external magnetic fields in a model. Assign it only to the outer surfaces of the problem region. Regardless of the model's drawing units, magnetic field values are entered in s/m.

> **Warning:** When using tangential magnetic field boundary conditions, always double-check that Ampere's law is not violated!

To define a zero tangential H field boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).
2. Click **Maxwell** > **Boundaries** > **Assign** > **Zero Tangential H Field**.

   The **Zero Tangential H Field** window appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Click **OK** to assign the boundary to the selected object.

To define a tangential H field boundary:

1. Select the section of the geometry on which you want to apply the boundary condition.
2. Click **Maxwell** > **Boundaries** > **Assign** > **Tangential H Field**.

   The **Tangential H Field** window appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the value of the field component in the x-direction in the **U(X)** box and the value of the field component in the y-direction in the **V(Y)** box.
5. In the **Coordinate System** section, do the following define the vectors:
   a. Select **New Vector** from the **U Vector** drop-down menu.
   The **Create Line** message appears asking you to draw the U vector of the coordinate system in the plane of the selected face.
   b. Click two points to specify the vector.
   The vector is set, and the **Tangential H Field** window reappears, with **Defined** listed for **U Vector**.
   c. To reverse the direction of the vector, click the **Reverse Direction** check box.
6. Optionally, click **Use Defaults** to revert to the default values in the window.
7. Click **OK** to assign the boundary to the selected object.

The new boundary is added to the boundary list in the Project Manager tree.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning an Integrated Zero Tangential H Field Boundary

This type of boundary is applied when the integral of the tangential H field is zero.

> **Warning:** When using integrated zero tangential magnetic field boundary conditions, always double-check that Ampere's law is not violated!

To define an integrated zero tangential H field boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).
2. Click **Maxwell 3D** > **Boundaries** > **Assign** > **Integrated Zero Tangential H Field**.

   The **Integrated Zero Tangential H Field** window appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Click **OK** to assign the boundary to the selected object.

The new boundary is added to the boundary list in the project tree.

Assigning an Insulating Boundary

This boundary condition is generally used to model very thin sheets of perfectly insulating material between touching conductors, as well as infinitely thin cracks inside conductors. Current cannot cross an insulating boundary.

To set an insulating boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).
2. Click **Maxwell** > **Boundaries** > **Assign** > **Insulating**.

   The **Insulating Boundary** window appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Click **OK** to assign the boundary to the selected object.

> **Note:** An insulating boundary condition can be used to control the flow of induced eddy currents inside conductors that contain sheet objects with an applied insulating boundary condition. In case you apply the insulating boundary condition on sheet objects (inside conductors), always make sure that the respective entities are selected as faces and not as objects before attempting to assign the insulating boundary condition to them.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning a Symmetry Boundary

This boundary condition defines a plane of geometric or magnetic symmetry in a structure. Assign it only to the outer surfaces of the problem region.

To set a symmetry boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).
2. Click **Maxwell** > **Boundaries** > **Assign** > **Symmetry**.

   The **Symmetry Boundary** window appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Select one of the following as the type of symmetry:

Odd (Flux Tangential)	H is tangential to the boundary; its normal components are zero.
Even (Flux Normal)	H is normal to the boundary; its tangential components are zero.

5. Optionally, click **Use Defaults** to revert to the default values in the window.
6. Click **OK** to assign the boundary to the selected object.

> **Warning:** When using even symmetry boundaries, always double-check that Ampere's law is not violated!

Assigning an Independent Boundary

Independent and dependent boundaries enable you to model planes of periodicity where the H-field at every point on the dependent boundary surface is forced to match the H-field of every corresponding point on the independent boundary surface. The transformation used to map the H-field from the independent to the dependent is determined by specifying a coordinate system on both the independent and dependent boundaries.

To set an independent boundary:

1. Select the face to which you want to assign the independent boundary.
2. Click **Maxwell** > **Boundaries** > **Assign** > **Matching** > **Independent**.

   The **Independent Boundary** window appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. In the **Coordinate System** section, do the following to define the coordinate system:

   > **Note:** You must specify the coordinate system in the plane on which the boundary exists. First draw the U vector of the coordinate system. Maxwell uses the U vector you draw and the normal vector of the boundary face to calculate the v-axis. Then specify the direction of the V vector.

   a. Select **New Vector** from the **U Vector** drop-down menu.
      The **Create Line** message appears, asking you to draw the U vector of the coordinate system in the plane of the selected face, and the **Independent Boundary** dialog box disappears while you draw the U vector.
   b. Select the U vector's origin, which must be on the boundary's surface, in one of the following ways:
      - Click the point.
      - Type the point's coordinates in the in the **X**, **Y**, and **Z** boxes.
   c. Select a point on the u-axis.
      The **Independent Boundary** dialog box reappears.
5. To reverse the direction of the vector, select the **Reverse Direction** check box.
6. Click **OK**.

The Maxwell solver computes the H-field on this boundary and maps it to the dependent boundary using the transformation defined by the independent and dependent coordinate systems.

Assigning a Dependent Boundary

Assigning a dependent boundary is the second step in creating matching boundaries. The field on the independent boundary is mapped to the dependent boundary.

> **Note:** You must define a independent boundary before creating the dependent boundaries that are associated with it. Assign dependent boundaries only to the outer surfaces of the problem region.

To set a dependent boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).
2. Click **Maxwell** > **Boundaries** > **Assign** > **Matching** > **Dependent**.

   The **Dependent Boundary** window appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Select the independent boundary to which the dependent is to be assigned from the **Independent** drop-down menu.
5. To specify the coordinate system, do the following in the Coordinate System section:

> **Note:** You must specify the coordinate system in the plane on which the boundary exists. First draw the U vector of the coordinate system. Maxwell uses the U vector you draw and the normal vector of the boundary face to calculate the v-axis. Then specify the direction of the V vector.

   a. Select **New Vector** from the **U Vector** drop-down menu.
   The **Create Line** message appears asking you to draw the U vector of the coordinate system in the plane of the selected face, and the **Dependent Boundary** dialog box disappears while you draw the U vector.
   b. Select the U vector's origin in one of the following ways:

> **Note:** The U vector's origin must be on the boundary's surface.

      - Click the point.
      - Type the point's coordinates in the in the **X**, **Y**, and **Z** boxes.

   c. Select a point on the u-axis in the same way.
   When the second point is identified, the vector is set, and the **Dependent Boundary** window reappears, with **Defined** listed in the **U Vector** drop-down menu.
   d. To specify the direction of the V vector, select or clear the **Reverse Direction** check box.

> **Note:** When entering current, current density terminals, or coil terminals, the arrow associated with current in the selected object shows the direction as if a positive value is entered for the current. If a negative value is entered the actual current flow direction is opposite to what the arrow shows. Click **Reverse Direction** if you want to reverse the direction of the arrow.

6. In the **Relation** section, select one of the following to specify the field behavior on the boundary:

Hdep = Hind	Click this radio button if the dependent and independent boundaries have the same magnitude and direction.
Hdep = -Hind	Click this radio button if the dependent boundary field has the same magnitude as but the opposite direction from the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

independent boundary field.

7. Optionally, click **Use Defaults** to revert to the default values in the window.
8. Click **OK** to assign the boundary to the selected object.

> **Note:** The origin must be a vertex point of one of the objects.

Assigning a Radiation Boundary

To simulate problems that allow fields to radiate infinitely far into space, you can define surfaces to be radiation boundaries. The system absorbs the field at the radiation boundary, essentially ballooning the boundary infinitely far away from the structure. In a far field region, field components are expressed by

$$\boldsymbol{E} = -\boldsymbol{Z}\boldsymbol{n} \times \boldsymbol{H}$$

where

$$\boldsymbol{Z} = \sqrt{\frac{\boldsymbol{\mu}}{\boldsymbol{\varepsilon}}}$$

and *Z* is the component of the E-field that is tangential to the surface.

Using the field impedance, the equation becomes

$$\boldsymbol{n} \times \boldsymbol{E} = -\boldsymbol{Z}\boldsymbol{n} \times (\boldsymbol{n} \times \boldsymbol{H})$$

which is used as a radiation boundary. The radiation boundary condition should be placed far enough from the source of radiation so that the approximation of the far field in the region of the boundary holds.

The second-order radiation boundary condition is an approximation of free space. The accuracy of the approximation depends on the distance between the boundary and the object from which the radiation emanates.

A radiation surface does not have to be spherical. However, it should be exposed to the background, convex with regard to the radiation source, and located at least one-quarter of a wavelength away from the radiating sources. In some cases you may want to use smaller distances.

To assign a radiation boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).
2. Click **Maxwell** > **Boundaries** > **Assign** > **Radiation**.

   The **Radiation Boundary** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Optionally, click **Use Defaults** to revert to the default values in the window.
5. Click **OK** to assign the boundary to the selected object.

Assigning an Impedance Boundary

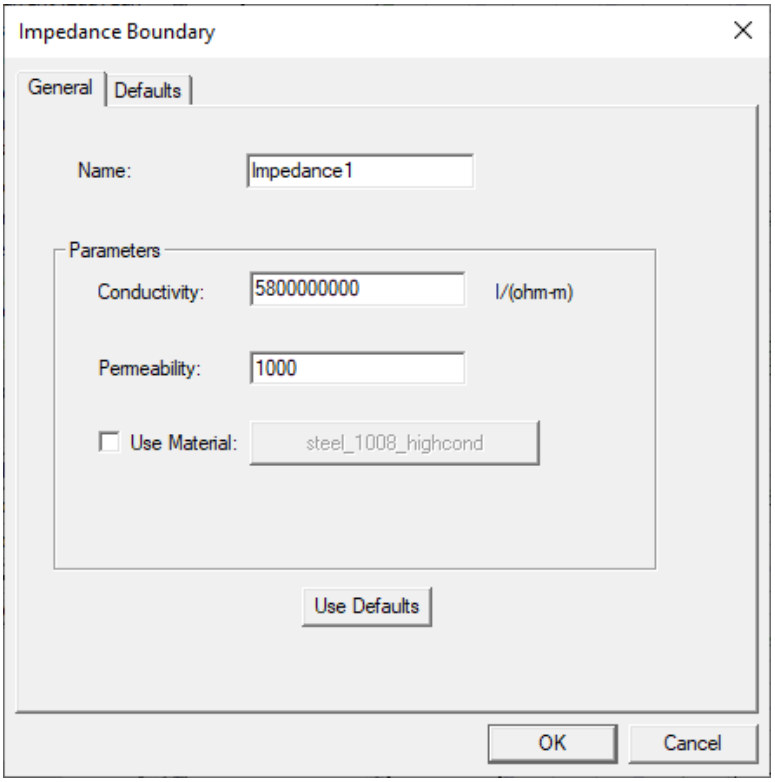
> **Note:** To define an impedance boundary for the transient solver, refer to Assigning an Impedance Boundary for the Transient Solver.

This boundary condition is used to simulate the effect of induced currents in a conductor without explicitly computing them. The conductor must be excluded from the model (saving time needed to mesh and solve for currents), so assign the impedance boundary condition to an outside edge of the problem region or to an excluded object.

To define an impedance boundary:

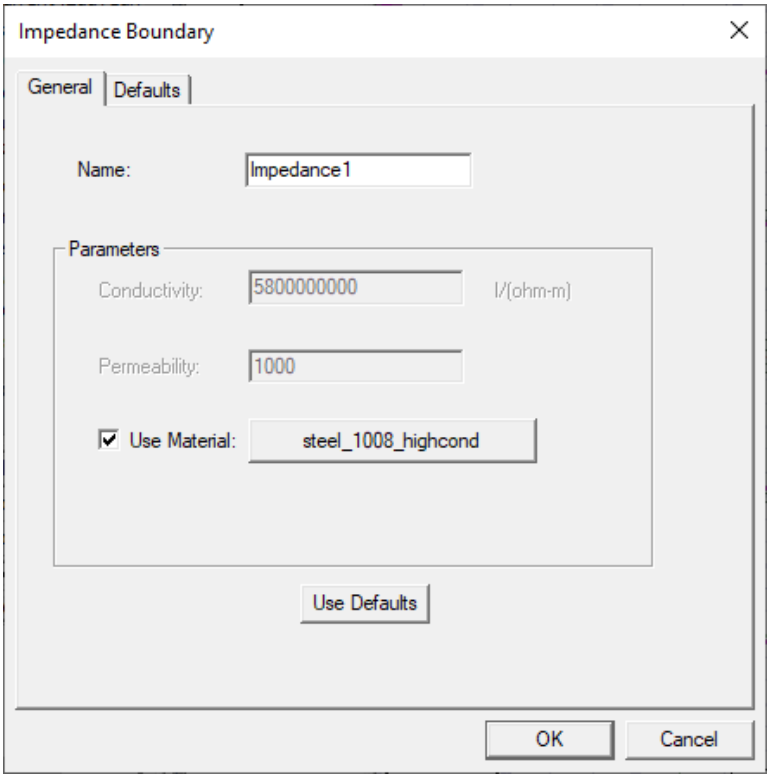
1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).
2. Click **Maxwell 3D** > **Boundaries** > **Assign** > **Impedance** to open the **Impedance Boundary** dialog box.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Do one of the following:
    - If the **Use Material** check box is unchecked, enter the conductivity (in inverse ohm-meters) in the **Conductivity** field, and the conductor's relative permeability in the **Permeability** field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- If you wish to select a material from the library, check the **Use Material** check box, then use the button to select the desired material.

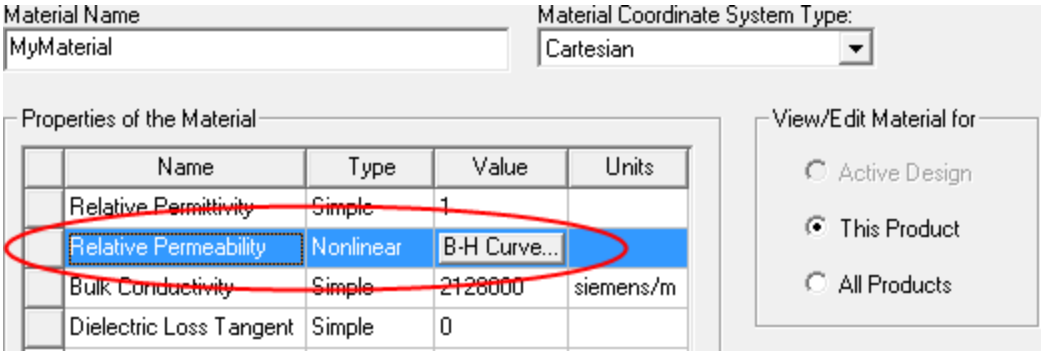
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Nonlinear Permeability**

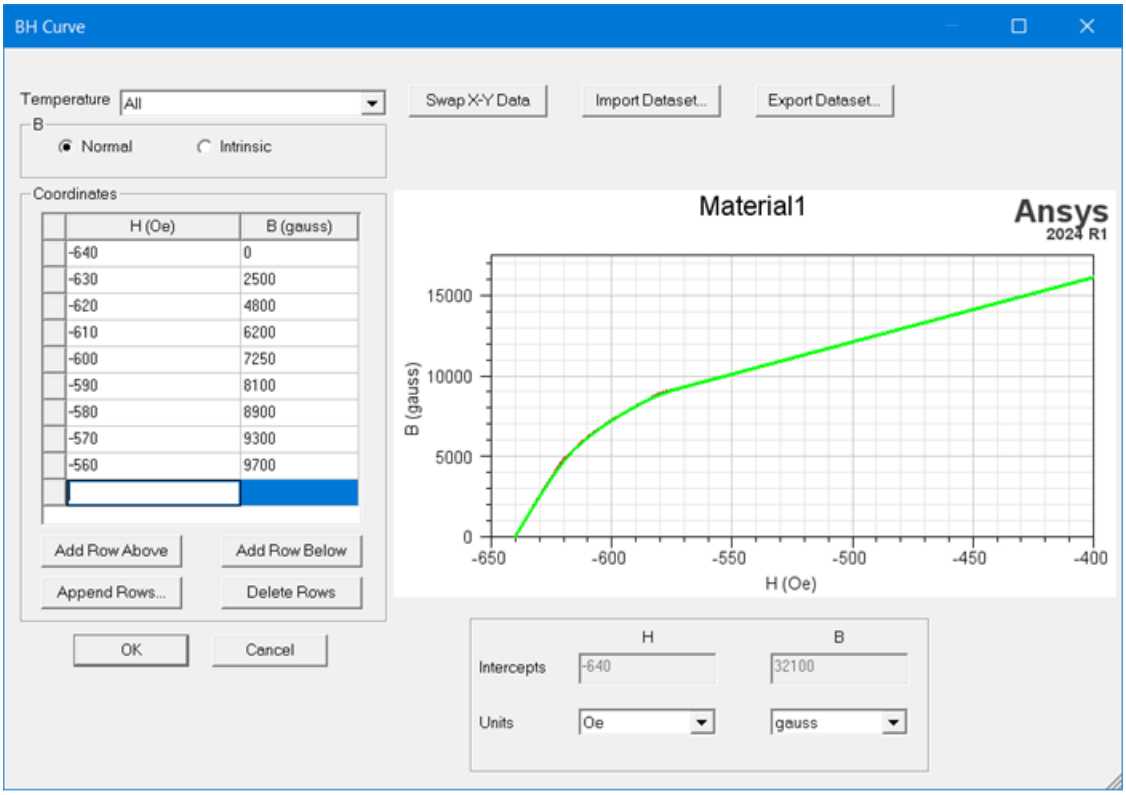
  If you select a material that has nonlinear permeability, the nonlinear coefficient is computed automatically.

  The nonlinear relative permeability of the material is determined by the B-H curve. If the desired B-H curve of the permeability belongs to an existing material, you can directly use that existing material.



  Otherwise, you can edit a new material with the desired B-H curve.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Refer to Relative Permeability for a Maxwell or RMxprt Material for details on using nonlinear relative permeability.

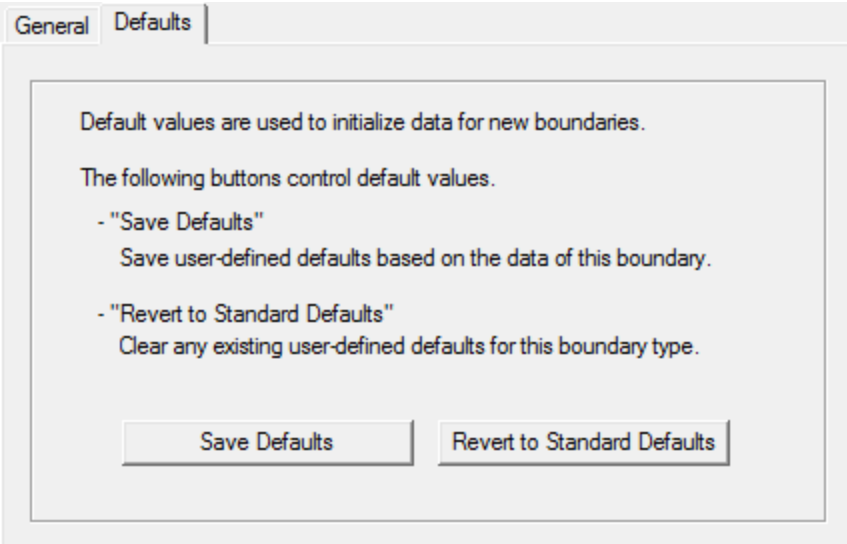
- **Temperature-Dependent Impedance Boundary**

  The electrical conductivity and permeability of a material used for an impedance boundary condition can be temperature dependent. The Maxwell 3D eddy solver provides an option to allow you to input functions as thermal modifiers to consider the effect of temperature on those properties. To enable this feature, check the **Use Material** check box as above. You can then click **Material** to open the material properties panel in which you can add a thermal modifier. (Refer to Setting the Temperature of Objects for additional information on using temperature-dependent impedance boundaries).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Optionally, click **Use Defaults** to revert to the default values in the window.

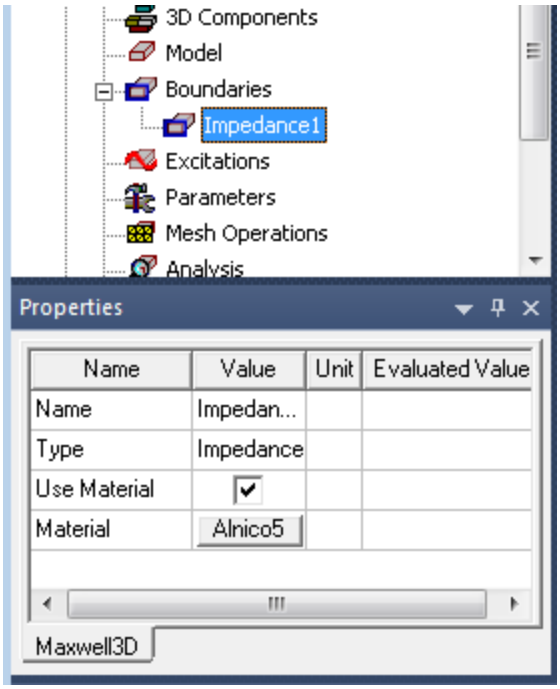
The **Defaults** tab allows you to control default values. The **Save Defaults** button saves the values currently defined on the **General** tab as the defaults to be assigned to new impedance boundaries. **Revert to Standard Defaults** clears existing user-defined values and replaces them with the standard default values.



6. Click **OK** to assign the boundary to the selected object.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Project Manager lists the newly assigned impedance boundary in the tree. You can select the boundary in the tree to view and edit its properties in the Properties Window. You can also double-click the boundary entry in the tree to open it for editing in the Impedance Boundary dialog box.



Impedance Boundary

Impedance boundaries allow you to simulate the effect of induced currents in a conductor without explicitly computing them. Use this boundary condition for models where

- the skin depth in the conductor is less than two orders of magnitude smaller than the dimensions of the structure. In models like this, the mesher may not be able to create a fine enough mesh in the conductor to compute eddy currents.
- the magnetic field decays much more rapidly inside the conductor in the direction that's normal to the surface than it does in directions that are tangential to the surface.
- the AC current source is relatively far away from the surface where eddy currents occur, compared to the size of the skin depth.

The conductor itself must be excluded from the solution region. When setting up the model, do one of the following:

- For external boundaries, when drawing the model, make the surface along which eddy currents are to be computed an outer surface of the problem region.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For internal impedance boundaries, assign perfect conductor to the object in the Material Manager or uncheck **Solve Inside** to exclude the object from the solution domain.

Then, when defining boundaries, assign an impedance boundary to the individual surfaces of the problem region (for external boundaries) or to the entire object (for internal boundaries). By entering the conductivity, s, and the relative permeability, mr, of the object, you specify the skin depth of induced eddy currents. The simulator uses this skin depth value when computing the electromagnetic field solution. It assumes that the H-field falls off exponentially inside the conductor. The ohmic loss due to induced currents can then be computed from the tangential components of the H-field along the impedance boundary — the surface of the object that you are interested in.

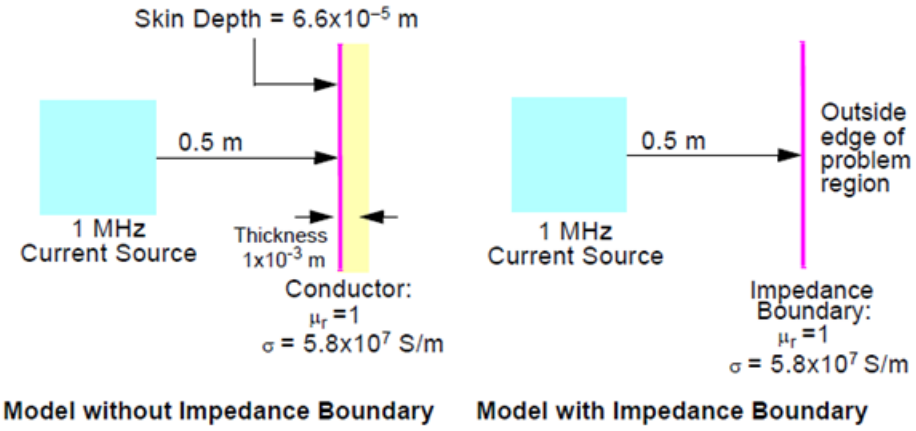
> **Note:** An impedance boundary only approximates the effect of eddy currents acting at a shallow skin depth. It does not directly compute them. In general, the fields modeled using an impedance boundary will closely match the field patterns that would actually occur in the structure. However, at discontinuities in the surface (such as corners), the field patterns may be different.

> **Note:** For the current version of the Maxwell solver, when a project includes all three of the following at the same time: an impedance boundary condition, a material with anisotropic permeability, and a winding, the project cannot be solved by the Maxwell AC Magnetic solver and the Maxwell 3D transient solver.

When to Use Impedance Boundaries

A typical situation where impedance boundaries can be used to reduce the complexity of a model is shown below. Suppose you want to compute eddy current losses in the conductor next to the current source shown below on the left. If the source carries AC current at a frequency of 1 MHz, the skin depth in the conductor is 6.6×10^{-5} meters. This is several orders of magnitude smaller than the conductor's thickness. Because the conductor where currents are induced is also relatively far away from the current source, an impedance boundary can be used to model the induced currents — as shown on the right.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Model without Impedance Boundary **Model with Impedance Boundary**

The conductor itself must be excluded from the model. Instead, the outside boundary of the model is moved to the inside surface of the conductor. This outside surface is defined as an impedance boundary, using the conductivity and permeability specified previously. Since the simulator does not have to actually compute a solution inside the conductor, the field solution is computed more quickly and uses less memory. After solving, you can compute the ohmic loss for the surface using the solution calculator and plot the loss density on the boundary.

Assigning a Resistive Sheet Boundary

> **Note:**
>
> This topic describes assigning a resistive sheet boundary for the 3D AC Magnetic T-Omega solver. For Transient designs refer to Assigning a Resistive Sheet Boundary for the Transient Solver.
> For 3D Magnetostatic designs refer to Assigning a Resistive Sheet Boundary for the 3D Magnetostatic Solver.

When two conductors are in contact within a conduction path, a voltage drop normally exists across the contact surface due to the imperfection of the contact between the two conductors. This boundary condition assigns a resistive sheet to consider the impact of this kind of voltage drop. The resistive sheet is defined by a lumped resistance in ohms.

The following are usages of the resistive sheet boundary:

- **Typical usage**: Modeling contact resistance by applying Resistive Sheet boundary to a contacting surface (face) of two conductive objects.
- **Other possible usages**: Resistive Sheet boundary can be applied to any sheet object inside a conductive object or on a conductive object surface in order to increase the total

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

resistance of the conducting path and to introduce localized loss density to the place of the sheet location.

This boundary can also be used to model terminal contact resistance (for stranded or solid conductors). For example, by applying it to the conductor cross section touching the region. In this case the Excitation Terminal and Resistive Sheet boundary are not allowed to be on the same sheet object.

The losses incurred due to a Resistive Sheet boundary are output as Surface Loss Density. These losses are counted as losses in solid conductors and are reported together with the losses in other solid conductors under Solid Loss. If the Surface Loss Density is mapped to Mechanical, it should be assigned to a face as Heat Flux.

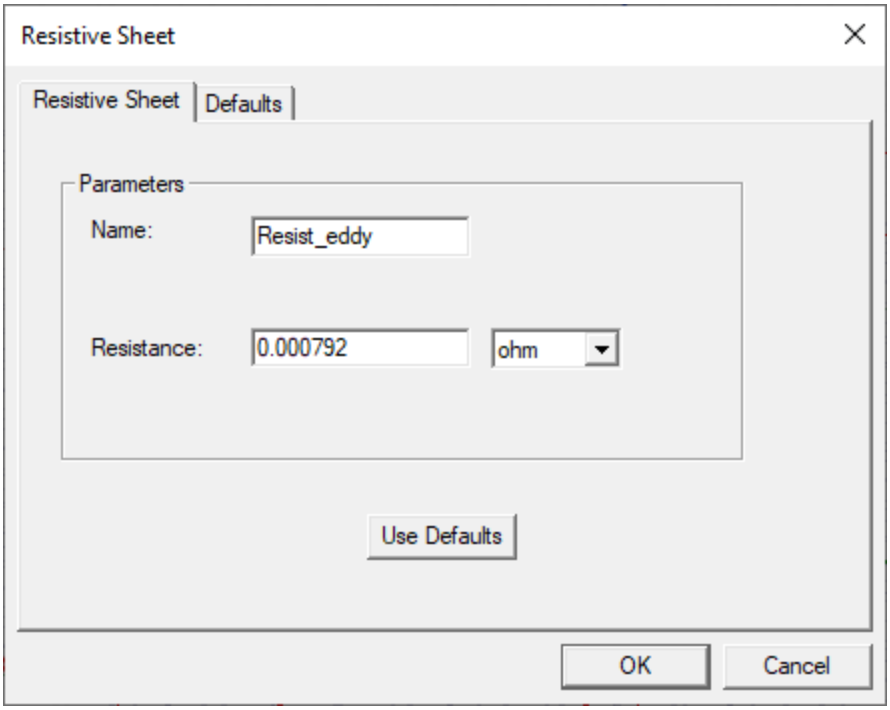
To define a resistive sheet boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a sheet object or face of a 3D conducting body touching another conducting body or face of a 3D conducting body touching the region).

   Note:
   - The sheet must be completely inside a conductor or on its surface.
   - The sheet must have conductors touching both of its faces.
   - The resistance value cannot be dependent on an intrinsic variable.

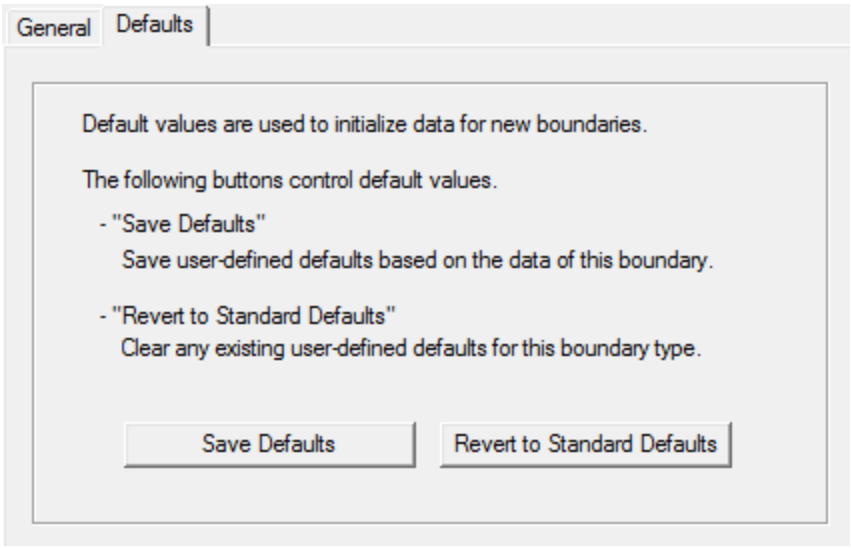
2. Click **Maxwell 3D** > **Boundaries** > **Assign** > **Resistive Sheet** to open the **Resistive Sheet** dialog box.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Enter the resistance value in the **Resistance** field, and select a unit of measure. Use of variables is supported.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Optionally, click **Use Defaults** to revert to the default value (1 ohm) in the window.

   The **Defaults** tab allows you to control default values. The **Save Defaults** button saves the values currently defined on the **General** tab as the defaults to be assigned to new impedance boundaries. **Revert to Standard Defaults** clears existing user-defined values and replaces them with the standard default values.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Click **OK** to assign the boundary to the selected object.

   The Project Manager tree lists the newly assigned resistive sheet boundary in the tree. You can select the boundary in the tree to view and edit its properties in the Properties window. You can also double-click the boundary entry in the tree to open it for editing in the **Resistive Sheet** dialog box.

> **Note:** When switching the solution type between transient and AC Magnetic, resistive sheet boundaries will be kept.

Assigning a Thin Layer Boundary

For the AC Magnetic (Eddy Current) solver, this boundary creates a field discontinuity between the two sides of the selected surface based on its thickness and the given electric and magnetic material properties. This allows representing eddy currents flowing inside the thin layer. When the air gap option is selected, the conductivity is assumed to be zero and the relative magnetic permeability is assumed to be equal to one. The thin layer is defined purely in terms of its thickness in this case.

This boundary condition can be used to model thin conductors and ferromagnets such as shields and narrow air gaps, as those in power transformers, inductors, or any other electric devices without needing to explicitly model (and mesh) the thin structure.

> **Note:**
> - The thin layer cannot be a closed shell encompassing a conductor.
> - The thin does not support a mechanical movement, *i.e.*, the boundary condition cannot be applied to the interface of moving and stationary bodies.
> - The thin layer cannot be defined on a conductor boundary or as a sheet object that touches or crosses any conducting body. In such cases, the Maxwell solver displays an error message indicating the name of the conducting body that the thin layer is touching.

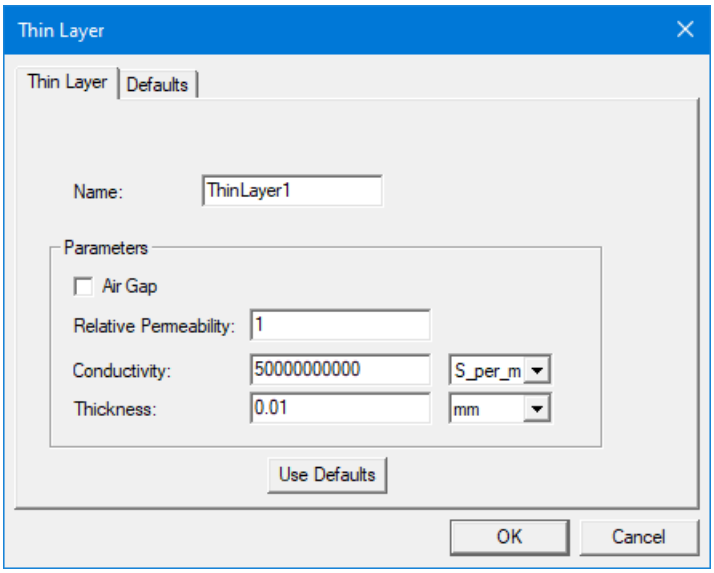
To define a thin layer boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a sheet object or face of a 3D body).
2. Click **Maxwell 3D** > **Boundaries** > **Assign** > **Thin Layer** to open the **Thin Layer** dialog box.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. If air gap is being modeled, check the **Air Gap** checkbox. The **Conductivity** and **Relative Permeability** fields will be grayed out (non-editable); the **Conductivity** of the thin layer will automatically be set to zero; and the **Relative Permeability** set to one.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Enter the **Thickness** of the air gap and the unit of measure. It must be a value greater than zero. The use of a variable is supported, but it cannot be an intrinsic variable.
6. If the **Air Gap** checkbox is unchecked, the thin layer can represent a thin conductor or ferromagnet. In this case, enter the material **Conductivity** with unit, and the **Relative Permeability** in the respective fields.

> **Note:** If AC Magnetic with DC Fields is the active solver, the Conductivity parameter will be used in the AC solve, but not the DC solve.



7. Optionally, click **Use Defaults** to revert to the default values in the window.
8. The **Defaults** tab allows you to control default values. The **Save Defaults** button saves the values currently defined on the **Thin Layer** tab as the defaults to be assigned to new impedance boundaries. **Revert to Standard Defaults** clears existing user-defined values and replaces them with the standard default values.
9. Click **OK** to assign the boundary to the selected object.

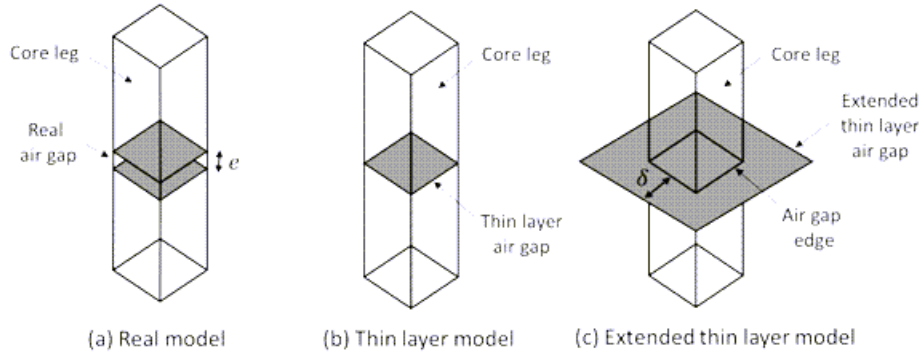
The Project Manager tree lists the newly assigned thin layer boundary in the tree. You can select the boundary in the tree to view and edit its properties in the Properties window. You can also double-click the boundary entry in the tree to open it for editing in the Thin Layer dialog box.

Getting Better Convergence with the Thin Layer Model

For an AC Magnetic simulation, the thin layer model for air gaps requires the duplication of the scalar potential to create a field discontinuity beyond the dimension of the sheet object and correctly represent the fringing flux effects. The Maxwell solver automatically duplicates the potential in one layer of elements from the gap edges. However, during the mesh adaptive

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

process, the lateral elements may become too small and thus too close to the gap sides. This results in a low convergence of the mesh adaptive steps as the magnetic energy on such elements becomes more important. To improve convergence, the user can extend the sheet object manually (see figure below). Better convergence is observed by raising its lateral sides from one to three times the air gap thickness ($\delta \cong 3e$) depending on the model geometry.



AC Magnetic T-Omega Excitations

For the AC Magnetic T-Omega solver, the sources of AC magnetic fields listed below are available. Excitations are assumed to be peak quantities (not RMS).

Excitation	Type of Excitation
Current Density	The current density in a conductor. In this case, you must also define the current density terminals for the conductor.
Current Density Terminal	Specifies the cross-section of the conductor where current density is specified, to be used as a terminal object. Current Density and Current Density Terminal are required to be used in conjunction.
Current	Total current in a conductor
Coil Terminal	Used to define one or more model windings
Winding With Current	Current for both a stranded and solid conductor
Winding With Voltage	Voltage for both a stranded and solid conductor

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Winding With External Circuit Connection	External circuit connection for both a stranded and solid conductor

Note: When entering current, the arrow associated with current in the selected object shows the direction as if a positive value is entered for the current. If a negative value is entered the actual current flow direction is opposite to what the arrow shows. Click **Reverse Direction** if you want to reverse the direction of the arrow.

Note: When a source conductor touches an outer boundary, Maxwell assumes current will flow perpendicular to the boundary. If you do not want current to cross the outer boundary, an insulating boundary should be used.

Note: The 3D AC Magnetic T-Omega solver can be used to get harmonic losses on nonlinear systems and can be an alternative to a transient simulation. See Using the AC Magnetic Solver to Generate Harmonic Losses for more details.

Assigning a Current Density Excitation

This command specifies the x-, y-, and z-components of the AC current density in a conduction path. If the current density is a function of position, the value is entered in A/m^2, even if you change the units in the problem.

To define the current density:

1. Select the section of the geometry (*i.e.*, the conductor) in which you want to apply the excitation (typically a 3D object).
2. Click **Maxwell 3D** > **Excitations** > **Assign** > **Current Density**.

   The **Current Density Excitation** window appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, specify the following options:
   a. Enter the **X Component**, **Y Component**, and **Z Component** values to define the respective components of the current density vector.
   b. Do the following in the two **Coordinate System** drop-down menus:
      - From the first drop-down menu, select either **Global** or **RelativeCS1** (where x = 1,2,3… if any relative coordinate systems have been defined).
      - From the second drop-down menu, select **Cartesian**, **Cylindrical**, or **Spherical**.
   c. Enter the phase angle, θ, of the current density in the **Phase** field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Optionally, click **Use Defaults** to revert to the default values in the window.
6. Click **OK** to assign the excitation to the selected object.

To complete the current density definition, you also need to specify any related current density terminals, which must be assigned to a 2D object or face of the respective conduction path.

> **Note:** Select the names of the sheet objects to serve as the current density terminals. This object must form an exact cross-section of the current density conduction path. You must create 2D objects which represent locations where current flows into and out of the problem region, or branches at any location in the conduction path. In current loops, any exact 2D cross-section may serve as a terminal.

Assigning a Current Density Terminal Excitation

To define the current density terminal:

1. Select the section of the geometry (*i.e.*, the conductor) in which you want to apply the excitation (typically a 3D object).
2. Click **Maxwell 3D** > **Excitations** > **Assign** > **Current Density Terminal**.

   The **Current Density Terminal Excitation** window appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. Click **OK** to assign the excitation to the selected target(s).

Assigning a Current Excitation

Specifies the total AC current in a conduction path. The conduction path may be contained completely within the problem region (for example, a coil), or may touch the edges of the problem region.

To specify the total AC current:

1. Select the 2D section of the geometry on which you want to apply the excitation.
2. Click **Maxwell 3D** > **Excitations** > **Assign** > **Current**.

   The **Current Excitation** window appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. Enter the current strength in the **Value** box.
5. Enter the phase for the current in the **Phase** box.
6. Select either **Solid** or **Stranded** to define the **Type** for the conductor. For a stranded conductor, a uniformly distributed current density is assumed without considering eddy effects.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** The Solid/Stranded setting is not available for a 3D AC Magnetic A-Phi design. The Solid type is always used for this type of design.

7. Optionally, click **Swap Direction** to change the direction of the current flow.
8. Optionally, click **Use Defaults** to revert to the default values in the window.
9. Click **OK** to assign the excitation to the selected object.

Assigning a Coil Terminal

To assign a coil terminal as an eddy-current excitation:

1. Select the section of the geometry on which you want to apply the excitation (typically a 2D planar object).
2. Click **Maxwell** > **Excitations** > **Assign** > **Coil Terminal**. (You can also right-click an existing winding in the project tree, and select **Assign Coil Terminal**.)

   The **Coil Terminal Excitation** dialog box appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the number of conductors for the coil terminal in the **Number of Conductors** box. For a solid winding, the number should be 1.

   > **Note:** This value represents only the number of conductors inside of the selected geometry. If the Coil Terminal is cut due to symmetry, then only enter the Number of Conductors in the portion modeled.
5. If you want to switch the direction for the coil terminal, click **Swap Direction**.

   > **Note:** The direction is shown in the model by a bold red arrow.
6. Optionally, click **Use Defaults** to revert to the default values in the window.
7. Click **OK** to assign the coil terminal to the selected object.

> **Note:**
>
> To add a coil terminal to an existing winding:
>
> 1. In the project tree, right-click the coil terminal, and select **Add to Winding**.
> 2. In the **Add to Winding** dialog box that appears, select the winding to which you want to add the coil terminal, and click **OK**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: To remove a coil terminal from an existing winding: In the Project Manager tree, right-click the coil terminal, and select **Remove from Winding**. The terminal is removed from the winding and moved up one level in the Project Manager tree (directly beneath **Excitations**).

Assigning a Winding Setup

Note:

- To define a winding for a Transient solution type, see Assigning a Winding Setup for a Transient Solver.
- To define a winding for a Magnetostatic solution type, see Assigning a Winding Setup for a Magnetostatic Solver.

You can use the predefined coil terminal(s) to define one or more current or voltage windings.

To define a winding for your model:

1. Click **Maxwell** > **Excitations** > **Add Winding** to open the **Winding** dialog box.
2. Enter a name for the winding in the **Name** box, or accept the default.
3. In the **Parameters** section:
   a. Select **Current**, **Voltage**, or **External** from the **Type** drop-down menu. The winding type selected enables only those fields (Initial Current, Resistance, Inductance, Voltage, Phase) applicable for the chosen type.
   b. Select the **Solid** or **Stranded** radio button to specify the type of conductor.
   c. For the 3D AC Magnetic T-Omega solver, there is a **Get Databases from FFT** button. This feature allows the user to get frequency domain signals from transient signals, and solve on the frequencies in AC Magnetic designs. For more information, see Using an AC Magnetic Excitation to Generate Harmonic Losses.
   d. Enter values in the following fields (enabled according to winding type), and select the desired units. You can also type a function as an expression for any of these fields except Initial Current.

For a current winding	For a voltage winding[1]	For an external winding
Current	Resistance	(no settings required)
Phase	Inductance	
	Voltage	
	Phase	

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

e. Enter a value in the **Number of parallel branches** text box.

> **Note:** Parallel branches for solid windings assume there are no circulating currents in the parallel branches.

4. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
5. Optionally, on the Defaults tab, click **Save Defaults** to save the current settings or click **Revert to Standard Defaults** to revert to the default values in the dialog box.
6. Click **OK** to assign the excitation to the selected object.

A winding is now defined for your model.

> **Note:** For an external winding, you must also set up an external circuit connection.

> **Note:**
>
> To add a coil terminal to this winding:
>
> 1. In the project tree, right-click the winding, and select **Add Terminals**.
>
>   The **Add Terminals** dialog box appears, listing all coil terminals that do not already belong to that winding.
>
> 2. Select the coil terminal(s) you want to add. To select multiple terminals, press **Ctrl** and click each terminal.
> 3. Click **OK**.
>
> To assign a coil terminal excitation and add it to this winding:
>
> 1. Select the section of the geometry on which you want to apply the coil terminal excitation.
> 2. In the project tree, right-click the winding, and select **Assign Coil Terminal**. The **Coil Terminal Excitation** dialog box appears.
> 3. Click **OK** to assign the coil terminal excitation.
>   The coil terminal excitation is assigned and is added to the winding.
>
> To delete all coil terminal excitations that belong to this winding:
>
> - In the project tree, right-click the winding, and select **Delete All Terminals**. All coil terminal excitations are removed from the winding and deleted from the excitations.

---

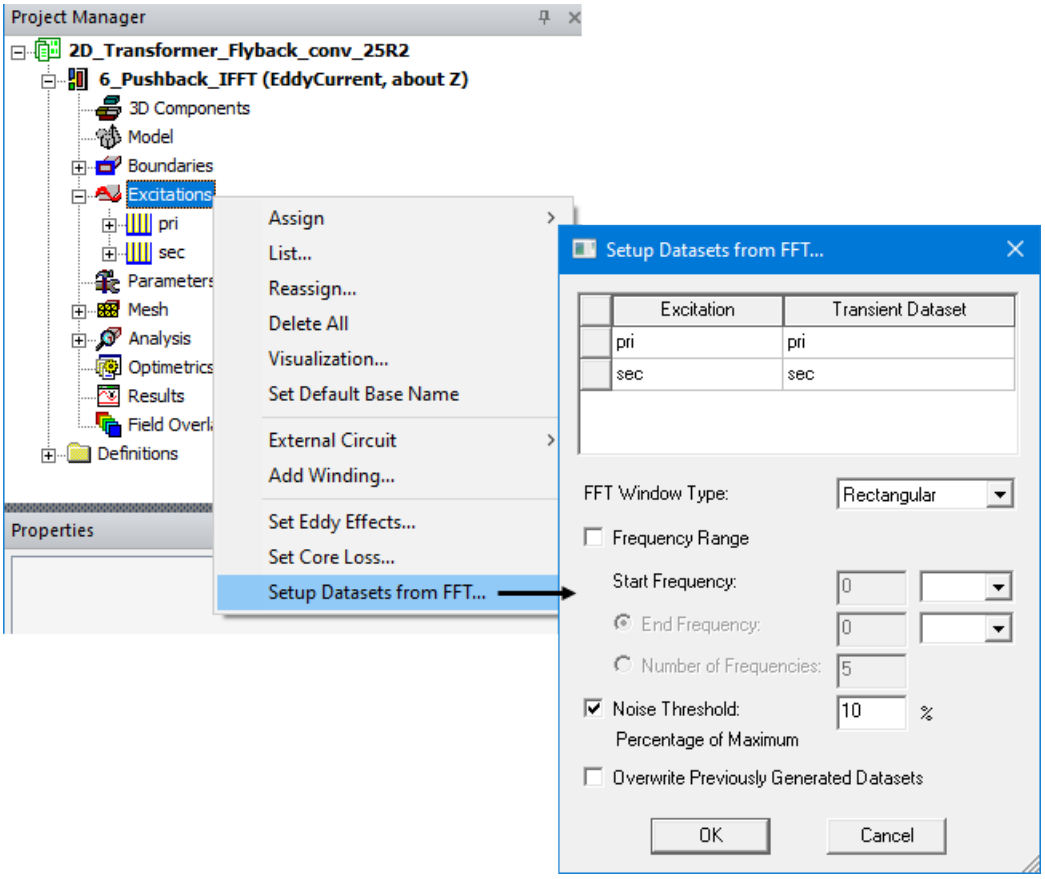
1. For a solid winding, the resistance term can represent: the resistance of a portion of the winding which is not modeled (for instance, end-effects), the leads connecting the winding

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

to the source, or the source resistance. (The main winding resistance is calculated directly by the solver.) For a stranded winding, the resistance term is the complete DC resistance of the winding (since the solver does not determine resistance of a stranded winding) as well as the resistance of the end-effects, leads, source, etc. For both solid and stranded windings, the inductance term can represent: the extra inductance for a portion of the winding of the winding which is not modeled (for instance, end-effects), the leads connecting the winding to the source, or the source inductance. (The main winding inductance itself is calculated directly by the solver.)

Using AC Magnetic Excitation to Generate Harmonic Losses

The 2D AC Magnetic (Eddy Current) and 3D AC Magnetic (Eddy Current) T-Omega solvers can be used to get harmonic losses on nonlinear systems and can be an alternative to a transient simulation. The **Excitations** entry in the Project Manager tree has the right-click menu **Setup datasets from FFT** option that can be used to reconstruct the signal vs time using a certain amount of harmonics from an FFT plot.



To generate the harmonic losses:

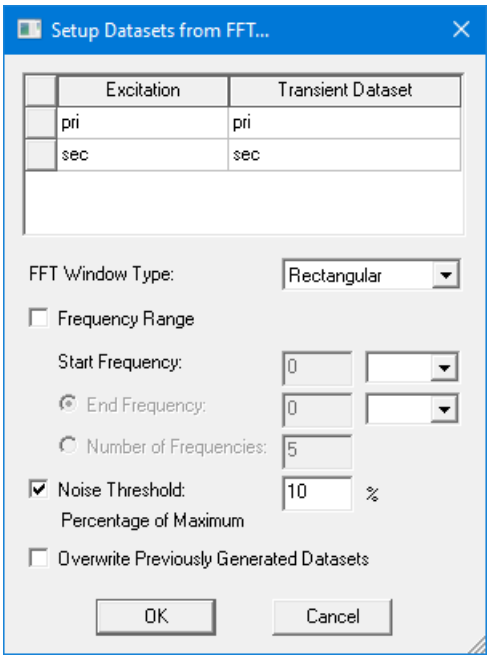
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. For each signal, input the broadband spectrum as datasets of amplitude (A) vs. frequency, phase (ϕ) vs. frequency. While each dataset might have different amplitudes, signals must have the same frequency points.

   Note: To use transient data as an AC Magnetic excitation, you have the following options:

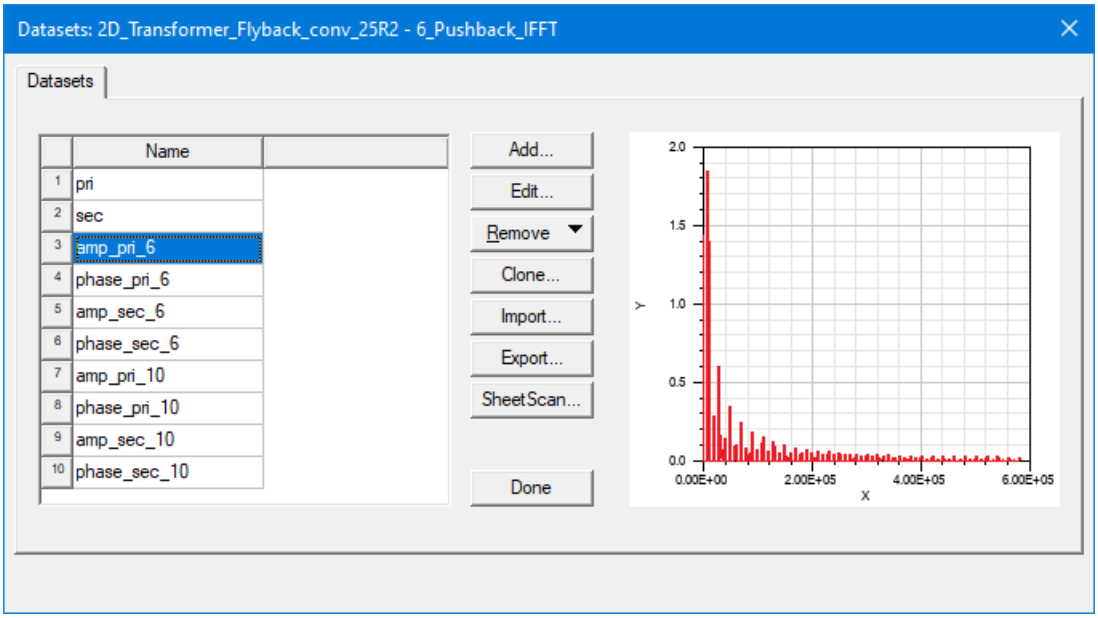
   - Get transient data from a third-party simulation (or from a Maxwell transient simulation) and perform FFT outside of Ansys Electronics Desktop to obtain two datasets as Amplitude vs Frequency and Phase vs Frequency. These two can be imported into Ansys Electronic Desktop as datasets and used as arguments within the pwl function of the excitation definition.
   - Get transient data from a third-party simulation (or from a Maxwell transient simulation) and import them as datasets and perform the built-in FFT function on selected excitations. This action will automatically generate a pwl function with corresponding arguments.
   - Define new frequency-dependent datasets within the excitation window by manually introducing the corresponding data pairs, Amplitude vs Frequency and Phase vs Frequency and manually defining the pwl function within corresponding excitation field definition.

2. Solve the frequency sweep and get EM losses on the broadband frequencies.
3. To retrieve the datasets, right click on the **Excitations** entry in the Project Manager tree, and select **Setup Datasets from FFT**. The following dialog box opens:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Select a transient dataset to use for each excitation listed. The winding or current excitations have to be defined before opening the Setup Datasets from FFT dialog box. The list of available transient datasets is determined by the Design Datasets dialog box, which is accessible from the design's right-click menu:



> **Note:** The FFT result of frequency lists should be the same for all excitations' transient datasets, which means the input transient datasets should have equivalent sampling rates and time windows.

5. Select the FFT window type.
6. Set the **Frequency Range**:
   - Users can input start and end frequency to specify the range or
   - Users can input start frequency and frequency number to specify the range. The Number of Frequencies defines the harmonic content where an upper frequency isn't necessarily the limit of interest. The **Number of Frequencies** is based on multiples of the fundamental frequency, which is defined by the duration of the transient dataset created by the user during pushback.
7. Set the **Noise Threshold**, which automatically drops harmonics that fall below the defined threshold regardless of the method used to create the list of harmonics.
   - The input is a percentage of maximum magnitude value of amplitude datasets.
   - Data of (A, f) whose magnitudes are smaller than the value will not be added to frequency-domain datasets. The data of (ϕ, f) at that frequency will also be discarded.

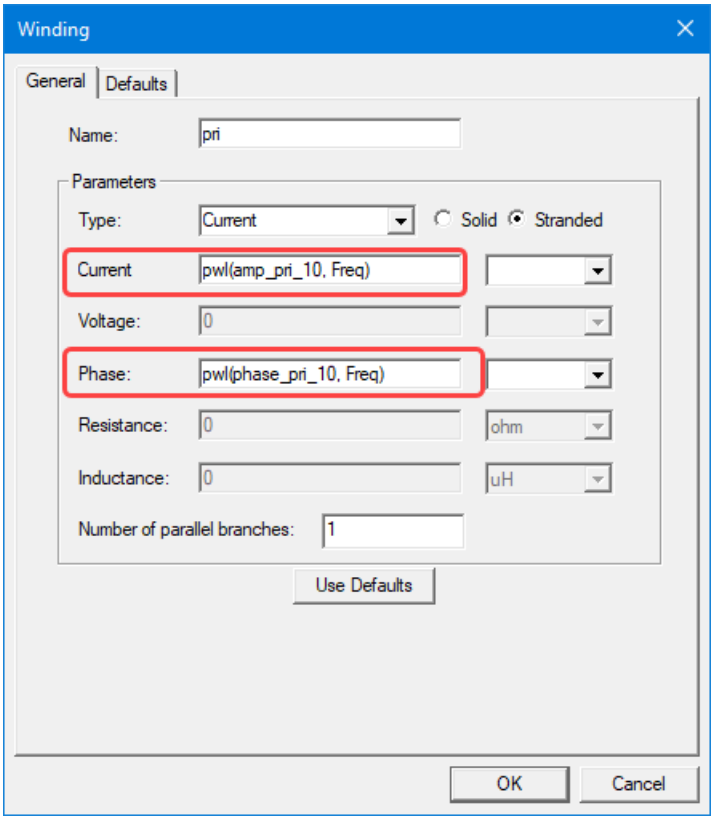
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Note that when the application filters out the frequencies whose amplitude is less than the noise threshold, a frequency is only removed if all excitations' amplitudes on that frequency are less than the noise threshold.

8. Check the **Overwrite Previously Generated Datasets** if you had previously run the Setup Datasets from FFT function and now want to update the results with new parameters. If this option is unchecked, new datasets will be created and appended to the Design Datasets dialog box.
9. Click **OK**.

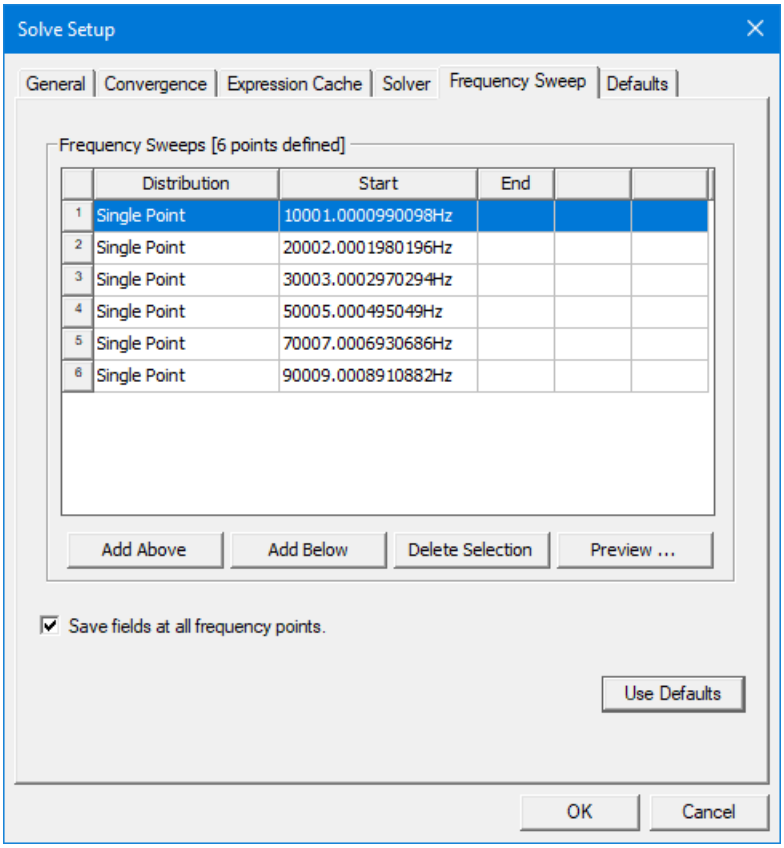
After clicking the **OK** button, the following operations occur:

1. Two datasets (A, f) and (ϕ , f) are generated and included in the list of design datasets. Note that after the datasets are generated, users can modify them.
2. Winding or current values (current/voltage, phase) will be updated with the PWL of these datasets.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Frequencies from the FFT process will be added to **Solve Setup** frequency sweep.



Validation

For each signal, the frequency points of (A, f) and (ϕ , f) datasets should be the same.

For all signals, the frequency points of all datasets should be the same.

> **Note:** This feature also supports harmonic loss coupling with EM Icepak and Ansys WorkBench.

External Circuit Setup

The Maxwell Circuit design editor is used to set up the external circuit excitation for windings in Maxwell 2D and 3D AC Magnetic solutions. Because the AC Magnetic solver is a frequency domain solver, only linear circuit components are supported, which include Resistance, Inductance, Capacitance, and Transformer. In addition, only sinusoidal sources should be used as sources. Sinusoidal Current Source and Sinusoidal Voltage Source, designed for transient

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

solutions, can also be used for AC Magnetic solutions. For example, to use the sinusoidal current source, you should set the parameters as follows:

- Set **I0**, **Td**, and **Df** to zero, where I0 is offset current in Amps, Td is the delay time in seconds, and Df is the damping factor in 1/seconds.
- Set **TIME** as the **Type**.
- Use **Ia** to set the peak amplitude in Amps.
- Use **IFreq** to set the signal frequency. Note that setting IFreq is not necessary as it will be internally set to the frequency defined in the Maxwell design setup.
- Use **Phase** to set the phase delay.

With these specifications, the time signal can be written as

$$I(t) = I_a \sin\left(2\pi I_{freq} - Phase\right)$$

Note that *Phase* is the signal phase delay. The complex source in the AC Magnetic solver is represented in phasor form as

$$I = I_a \; e^{j\theta}$$

and the time signal can be written in terms of (I_a, θ) as

$$I\left(t\right) = Im\left\{I_a \; e^{j(2\pi I_{freq} + \theta)}\right\}$$

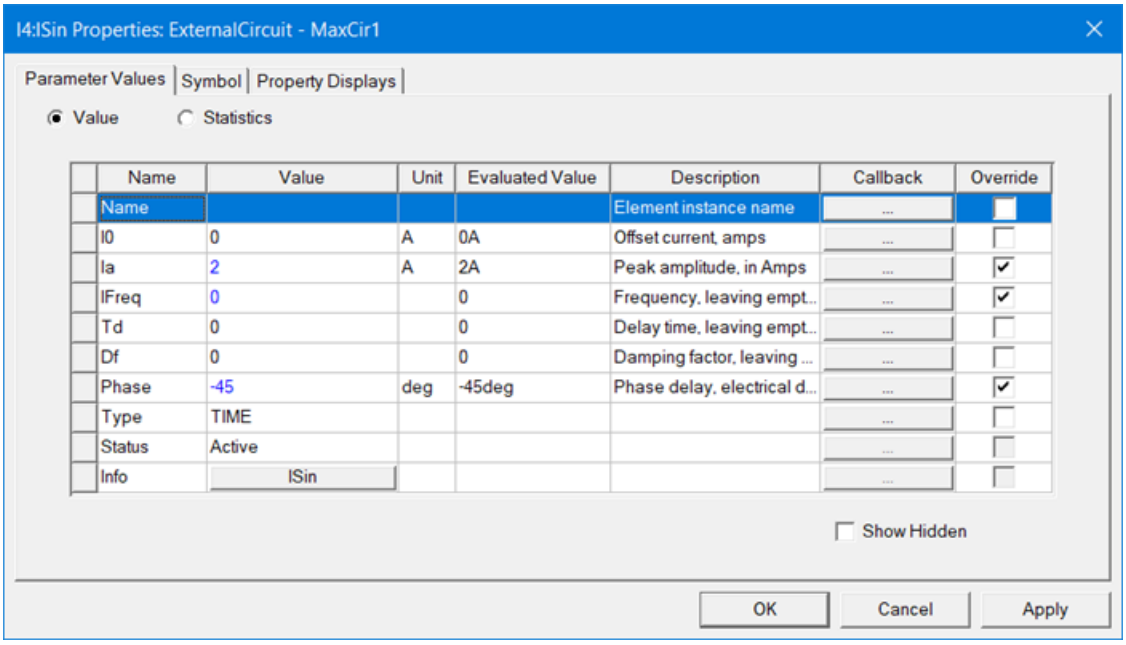
In the complex expression, θ is defined as phase, while in Maxwell Circuit, *Phase* is defined as phase delay. Thus,

$$Phase = -\theta$$

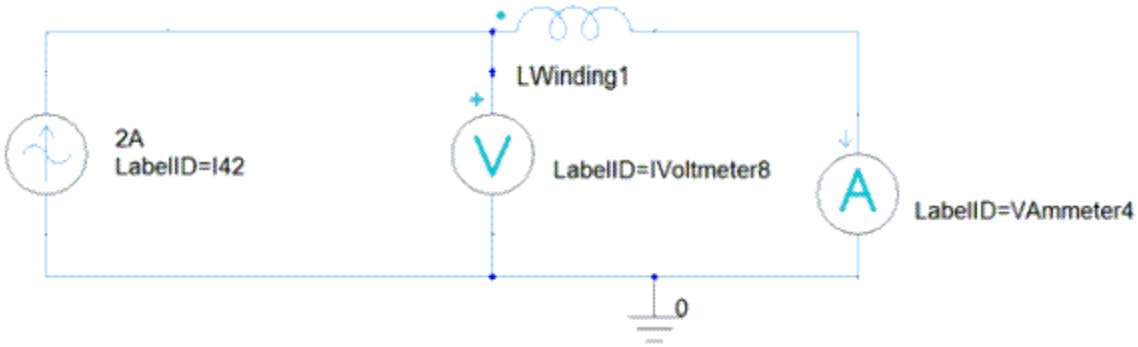
As a result, you should enter the negative of phase of the complex signal as the Phase value.

An example of such a setup for $I = 2e^{j45^\circ}$ is shown below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



An example of an external circuit excitation for the winding for an AC Magnetic solver is shown below.



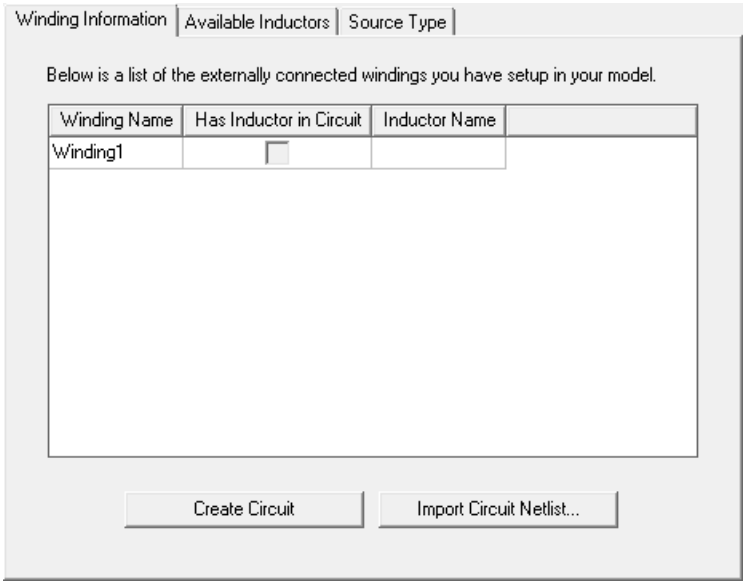
> **Note:** For AC magnetic solutions, only linear circuit components are supported, which include Resistor, Inductor, Capacitor, and Transformer. Only sinusoidal sources should be used as sources

To set up an external circuit connection for an External winding type:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

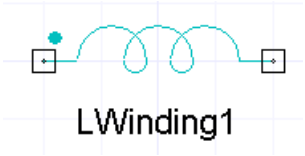
1. Click **Maxwell 3D** > **Excitations** > **External Circuit** > **Edit External Circuit** to open the **Edit External Circuit** dialog box. You can also right-click on **Excitations** in the Project tree and select **External Circuit** > **Edit External Circuit**.

    a. If no external winding connections have been set up for your model, the **Winding Information** tab lists the external winding(s) in the design. Continue with step 2.



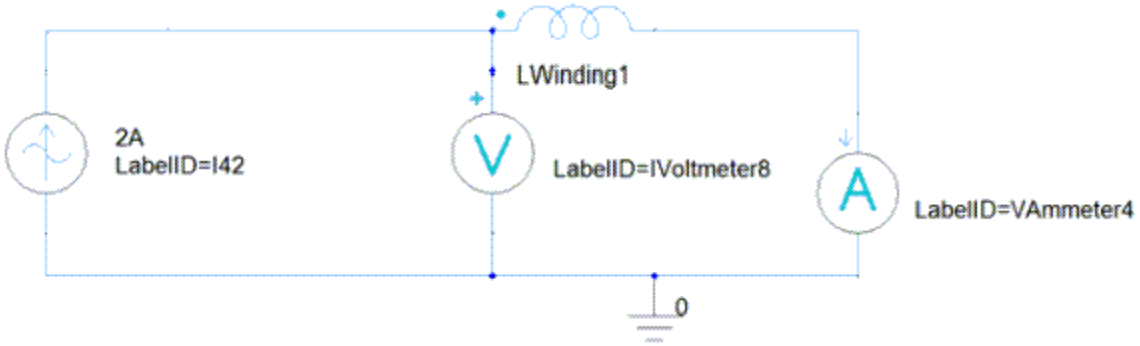
    b. If external windings have already been set up, continue with step 7.

2. On the **Winding Information** tab, click **Create Circuit** to add a circuit design to the project. The new circuit design automatically opens for editing in the Maxwell Circuit Editor. The circuit includes one inductor for each external winding in the original Maxwell transient design. The following example is for a design having one external winding.



3. Add the components needed to complete the external circuit design such as shown the following example.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. When the circuit design is finished, click **Maxwell Circuit** > **Export Netlist** and export the netlist **.sph** file to the desired location.
5. Return to the Maxwell design and click **Maxwell 3D** > **Excitations** > **External Circuit** > **Edit External Circuit** to open the **Edit External Circuit** dialog box. You can also right-click on **Excitations** in the Project tree and select **External Circuit** > **Edit External Circuit**.
6. On the **Winding Information** tab, click **Import Circuit** to import the circuit.
7. To view a list of inductors in the imported circuit, click the **Available Inductors** tab.
8. To view the sources used in the externally connected windings, click the **Source Type** tab.
9. If variables are used in the imported circuit, they and their design values are listed on the **Parameter Values** tab. You can link to variables within the imported circuit as follows:
    a. On the **Parameter Values** tab, click in the **Value** column of the parameter to be mapped.
    b. Enter a new local variable name and press the **Enter** key to open the **Add Variable** dialog.
    c. Select a **Unit Type** and **Unit**, enter a **Value** for the variable, and click **OK**. The new Maxwell variable is now mapped to the variable in the imported circuit, and can be varied directly by Maxwell or used for optimetrics analyses.
10. If the imported file is a .**sph** file, you can click the **Circuit Path** tab to view the original project and design names.

    > **Note:** After you import a circuit netlist, the **Create Circuit** button changes to **Update Netlist from Circuit**. Use **Update Netlist from Circuit** to reimport the netlist to keep it current.

11. Click **OK** to close the **Edit External Circuit** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AC Magnetic A-Phi Boundaries and Excitations

The following boundaries can be specified for this solver:

- Flux Tangential

The following excitations can be specified for this solver:

- Voltage
- Current
- 3D Layout Port

Include at least a voltage excitation (with a potential difference) as a reference for computing the electric potential.

AC Magnetic A-Phi Boundaries

The AC Magnetic(Eddy Current) A-Phi field solver allows you to define the following types of boundaries:

Boundary Type	**H-Field Behavior**	**Used to model…**
Default Boundary Conditions (Natural and Neumann)	For natural boundaries, the fields behave as follows: • B: normal component is continuous across the boundary • J: tangent component is continuous For Neumann boundaries, the fields behave as follows: • B: normal to the boundary • J: tangential on the boundary	Object interfaces are initially set to natural boundaries; outer boundaries are initially set to Neumann boundaries.
Flux Tangential	• B: tangential to the boundary and flux cannot cross it. • J: normal to the boundary. Problems with flux tangential at outer boundary.	Problems with flux tangential at outer boundary.

AC Magnetic A-Phi Excitations

The following excitations are available for AC Magnetic A-Phi solver. Excitations are assumed to be peak quantities (not RMS).

Excitation	**Type of Excitation**
Voltage	This excitation sets the electric

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Excitation	Type of Excitation
	potential (voltage) on a surface or object to a specific value.
Current	The AC current flowing across a surface (2D sheet).
3D Layout Port	See Simulating a 3D Layout Component with an A-Phi Solver

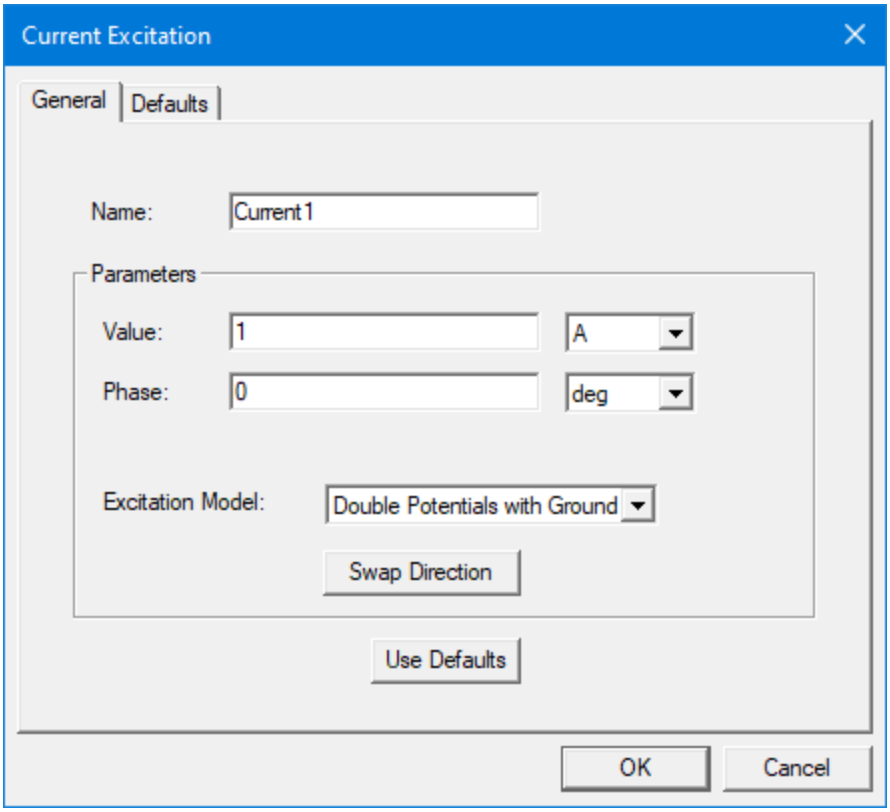
Assigning a Current Excitation

To define a current across a surface for the AC Magnetic A-Phi Solver:

1. Select the 2D section of the geometry on which you want to apply the excitation.
2. Click **Maxwell 3D** > **Excitations** > **Assign** > **Current**.

   The Current Excitation window appears.
3. Enter a name for the excitation in the **Name** field or accept the default.
4. In the Parameters section:
   a. Enter the current strength in the **Value** field, and select the unit from the drop-down menu.
   b. Enter the phase angle in the **Phase** field, and select the unit from the drop-down menu.
   c. Choose the **Excitation Model**. Refer to the next section for details.
   d. Optionally, to change the direction of the current flow, click **Swap Direction**.
5. Optionally, click **Use Defaults** to revert to the default values in the window.
6. Click **OK** to assign the excitation to the selected object.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Excitation Models

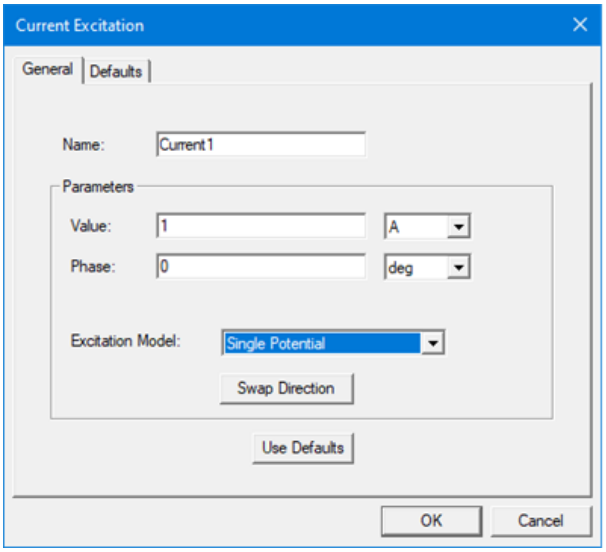
Current excitation supports three different excitation models for a Solid conductor type based on how the potential DOF is assigned on the face: **Single Potential**, **Double Potentials**, and **Double Potentials with Ground**.

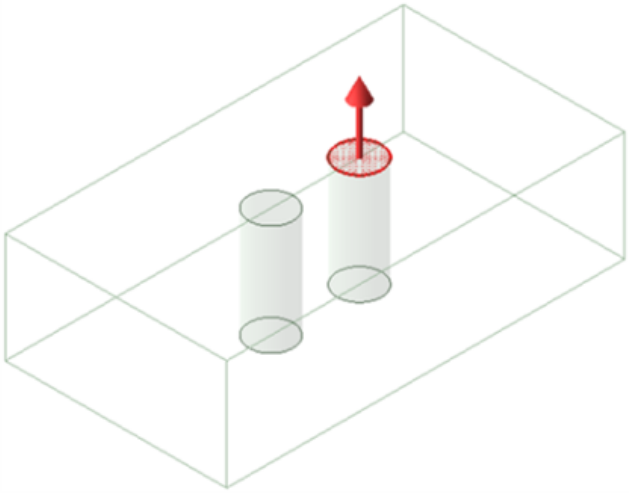
Single Potential:

Only one scalar electric potential DOF is assigned to the surface, and it is unknown.

> **Important:** This model must be used only at the external boundaries of the problem domain.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

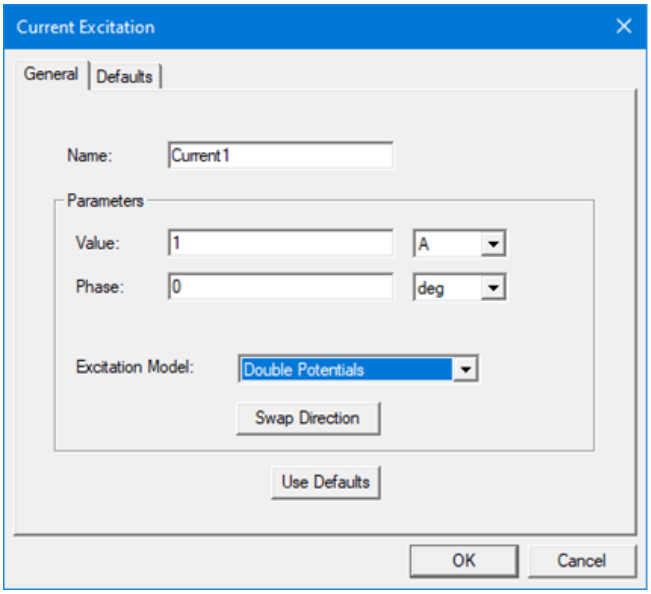


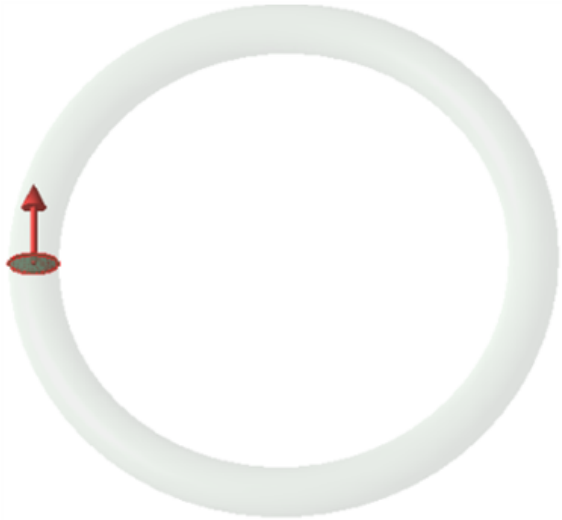


Double Potentials:

At the terminal face, two electric scalar potential DOFs are defined and the DOFs are unknown. There is a potential drop at the terminal. The current will be unidirectional, and it will flow in the direction that user specifies if the current value entered is a positive quantity.

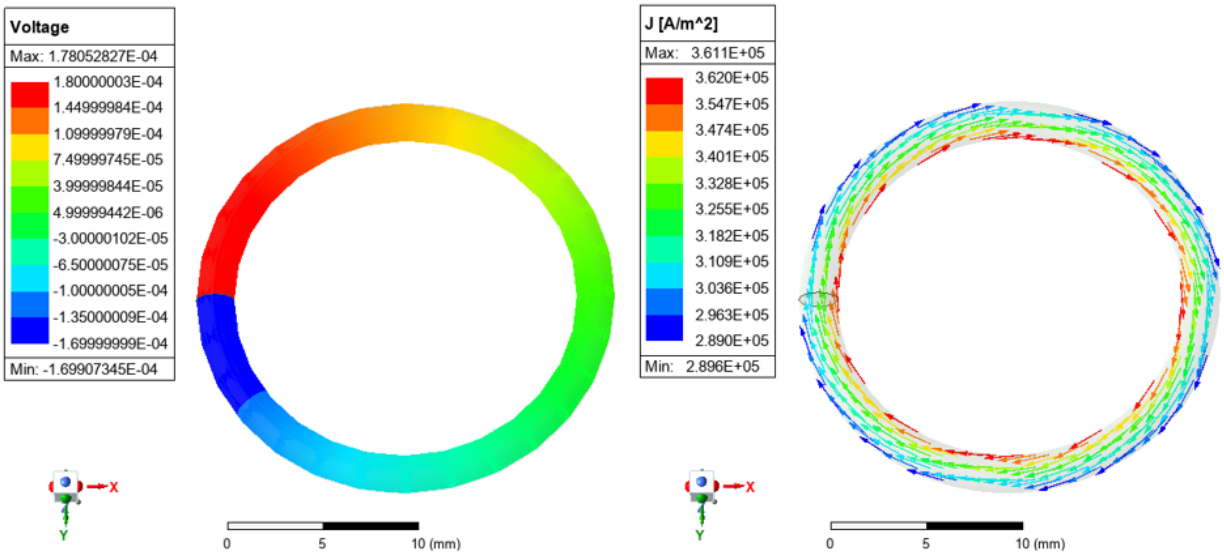
> **Important:** This model must be used on faces inside the problem domain.





The results are shown below:

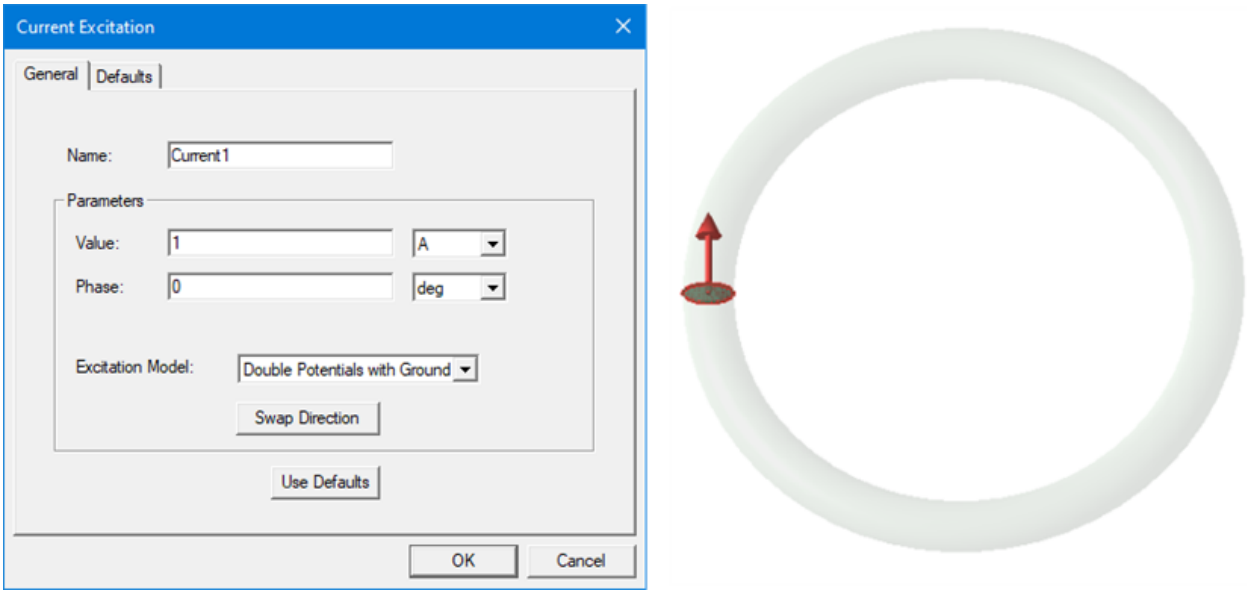
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Double Potentials with Ground:

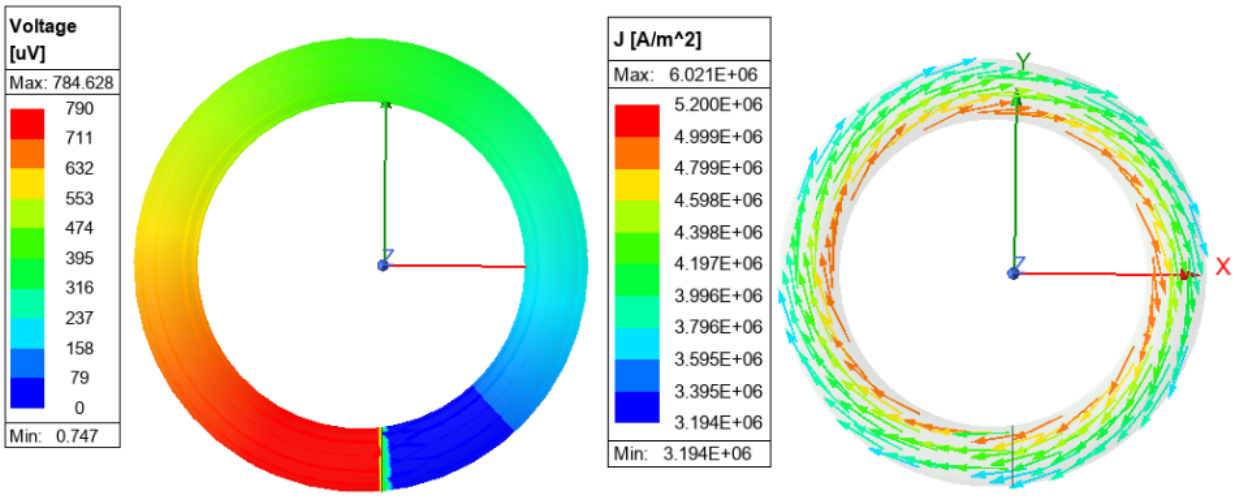
At the terminal face, two scalar electric potential DOFs are defined. The potential on the opposite side of the direction that the user specified is set to 0. The current will be unidirectional, and it will flow in the specified direction if the current value entered is a positive quantity.

> **Important:** This model must be used on faces inside the problem domain.



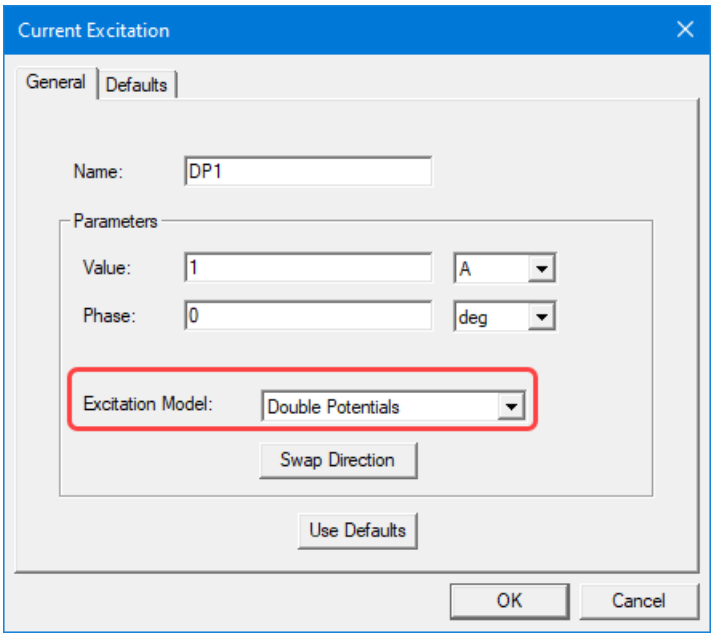
The results are shown below:

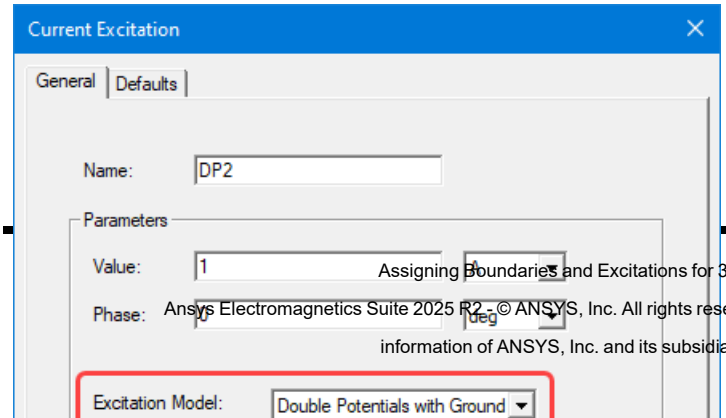
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Closed Loop Excitations

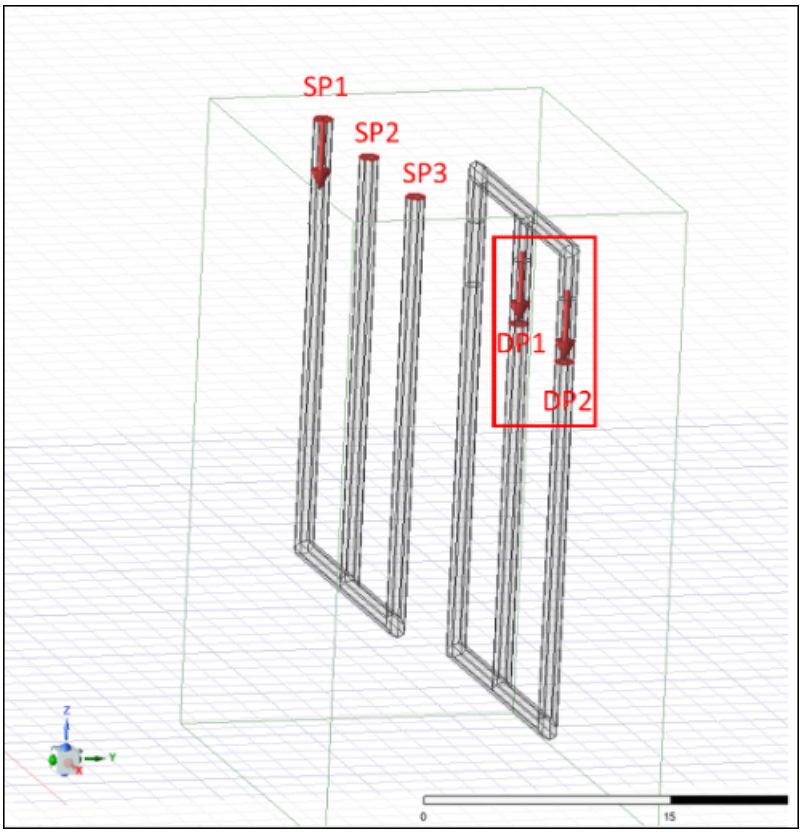
To simulate closed loop excitations, the Excitation Model should be Double Potentials or Double Potentials with Ground:





Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The figure that follows shows an example of closed loop excitation (DP1 and DP2) assignments. Multiple terminals assigned on one conduction path are also supported; see single potential excitation (SP1, SP2, SP3) assignments in the figure.



Note: Closed loop excitations are also applicable to single-frequency model export.

Assigning a Voltage Excitation

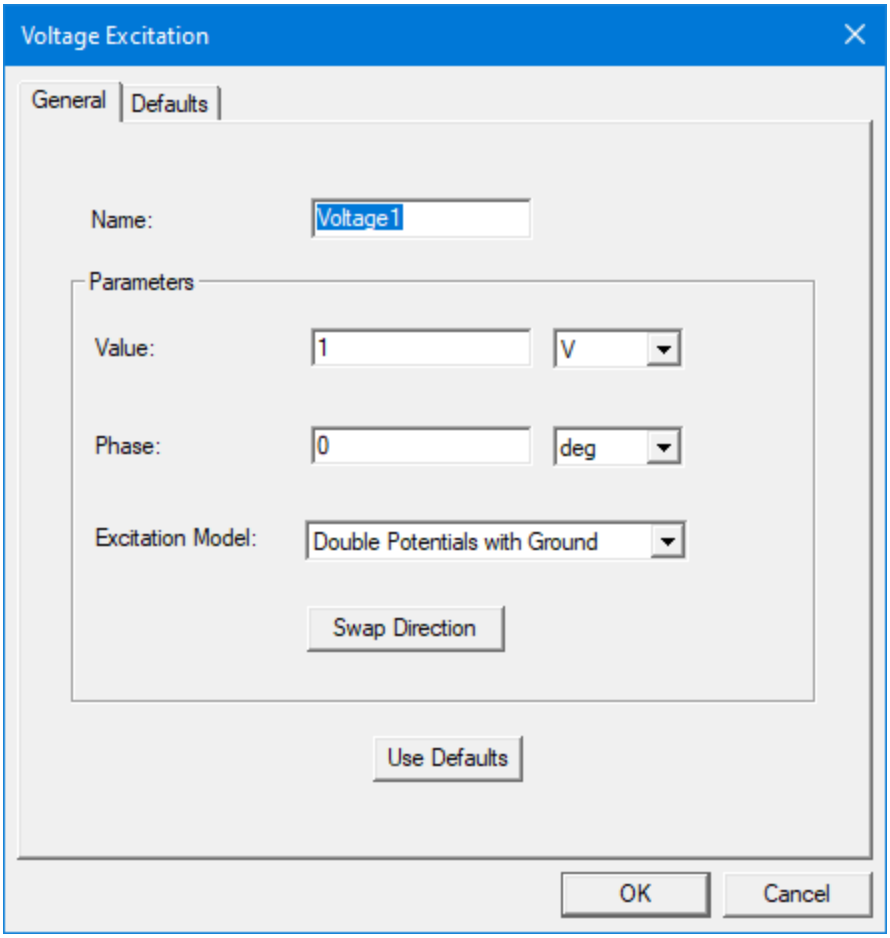
To set a voltage excitation for the 3D AC Magnetic A-Phi Solver:

1. Select the section of the geometry on which you want to apply the excitation.
2. Click **Maxwell 3D** > **Excitations** > **Assign** > **Voltage**. The Voltage Excitation window appears.
3. Enter a name for the excitation in the **Name** field, or accept the default.
4. In the Parameters section:
    a. Enter the electric potential in the **Value** field, and select the unit from the drop-down menu. You can enter a numerical value.
    b. Enter a value in the **Phase** field, and select a unit from the drop-down menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Choose the Excitation model. Refer to the next section for details on the options.
d. Optionally, to change the direction of the voltage flow, click **Swap Direction**.

5. Optionally, click **Use Defaults** to revert to the default values in the window.
6. Click **OK** to assign the excitation to the selected object.



Excitation Models

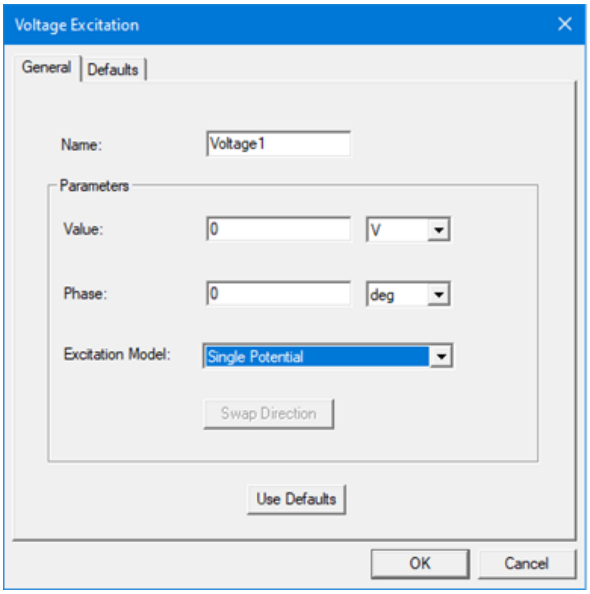
Voltage excitation supports three different excitation models: **Single Potential**, **Double Potentials**, and **Double Potentials with Ground**.

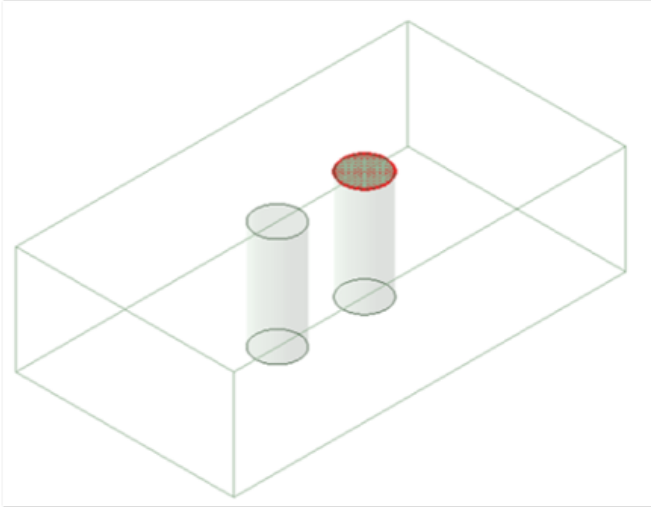
Single Potential

The solver treats the terminal as an equipotential terminal. Only one scalar electric potential DOF (Degree of Freedom) is assigned to the surface, and the value of the DOF is set to the value that the user inputs.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important: This model must only be used at the external boundaries of the problem domain.



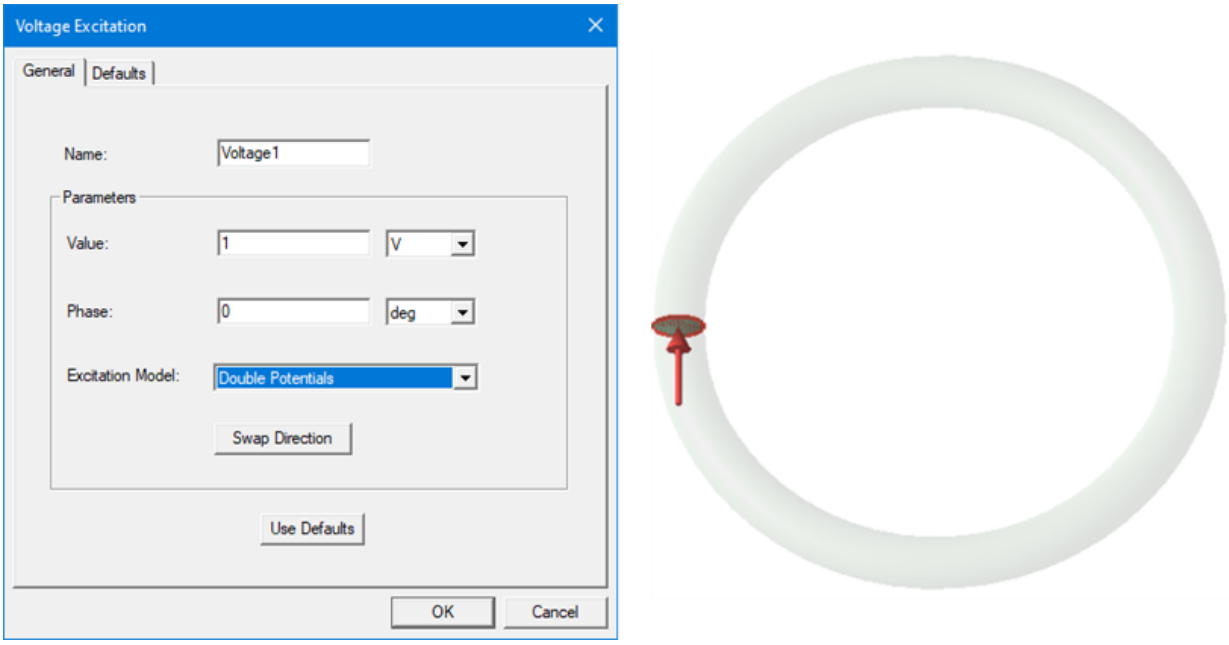


Double Potentials

The solver treats the terminal as a voltage (potential drop) terminal. At the terminal face, two scalar electric potential DOFs are defined, and the DOFs are unknown. The difference between those potentials equals the voltage value that the user specified. If the voltage value is a positive quantity, the higher potential is in the specified direction.

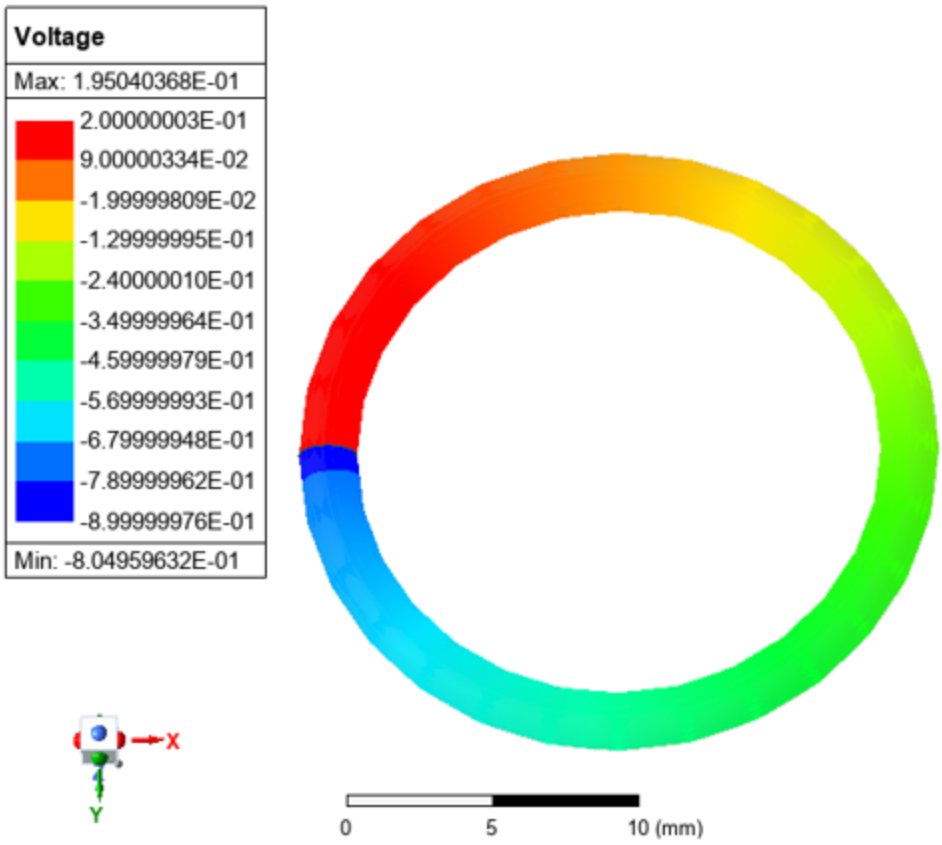
Important: This model must be used on faces inside the problem domain.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The voltage profile for **Double Potentials** is shown below:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

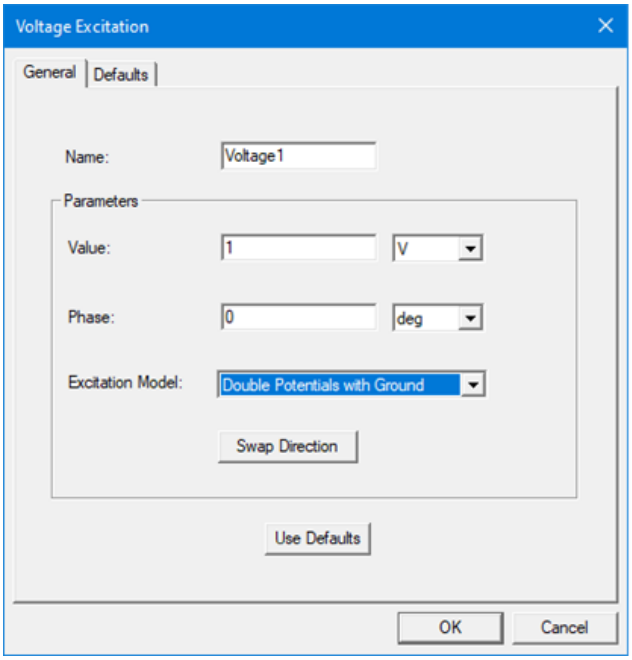


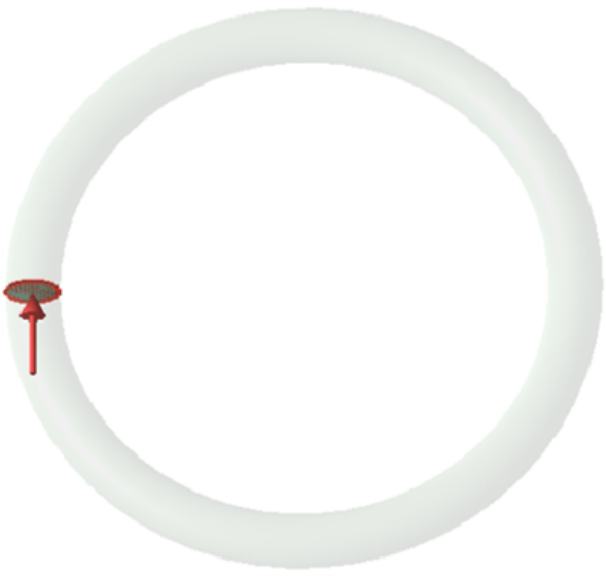
Double Potentials with Ground

The solver treats the terminal as a voltage (potential drop) terminal. At the terminal face, two scalar electric potential DOFs are defined, and both DOFs are known. The difference between those potentials equals the voltage value that the user specified. If the voltage value is a positive quantity, the DOF in the specified direction is set to the specified voltage value, and the DOF in the opposite direction is set to 0.

> **Important:** This model must be used on faces inside the problem domain.

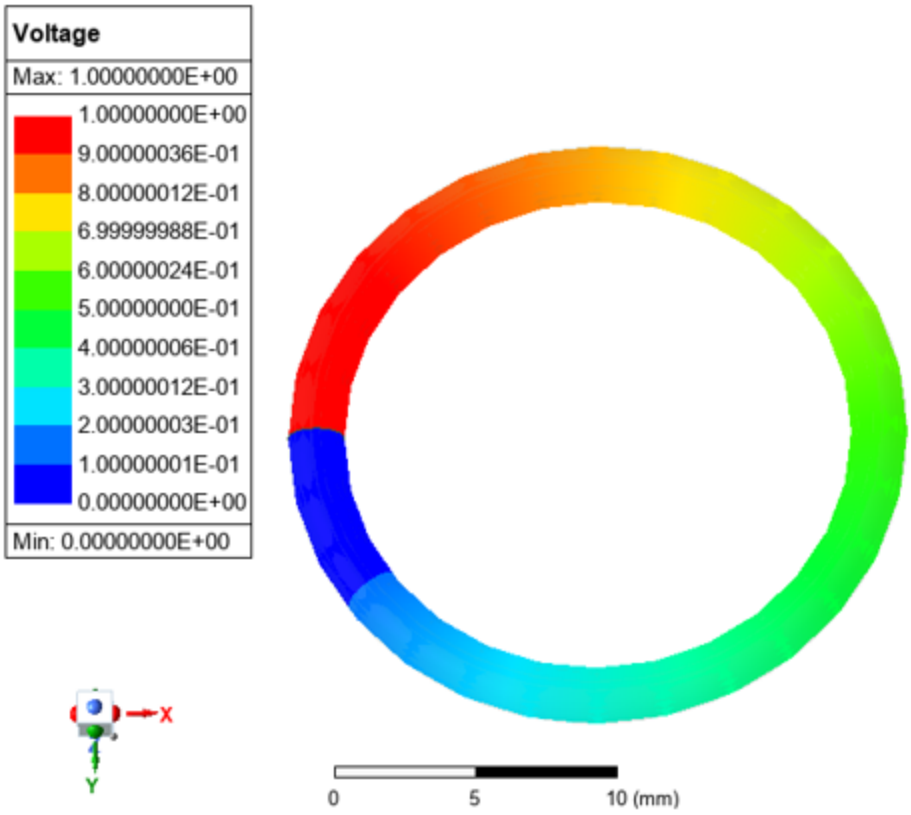
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





The voltage profile for **Double Potentials with Ground** is shown below:

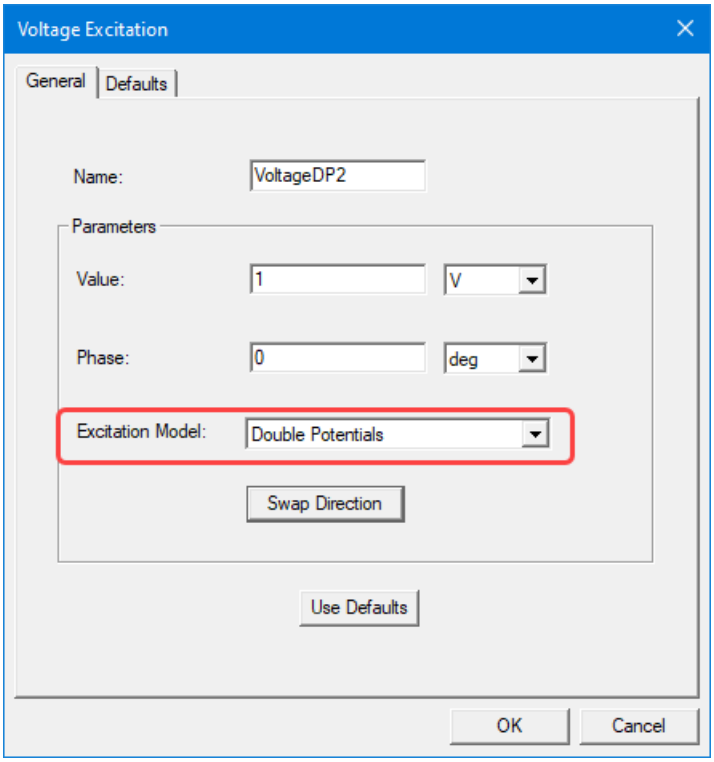
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

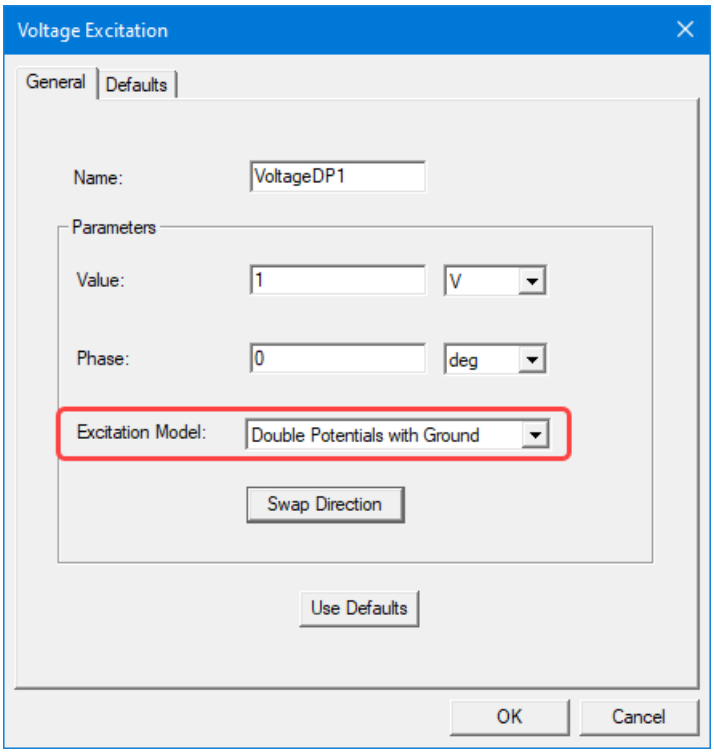


Closed Loop Excitations

To simulate closed loop excitations, the Excitation Model should be Double Potentials or Double Potentials with Ground:

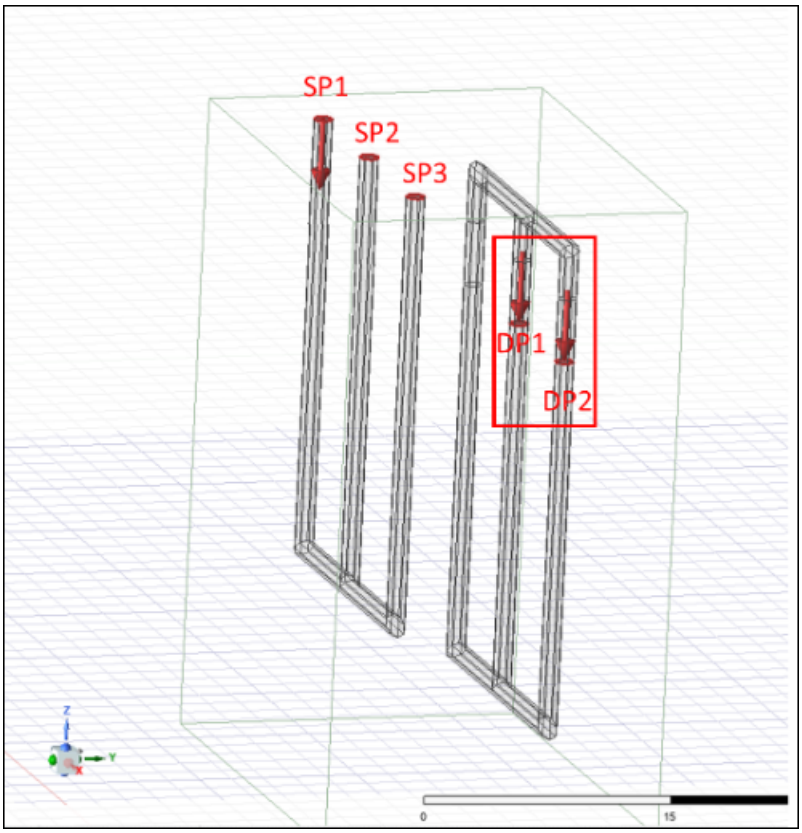
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The figure that follows shows an example of closed loop excitation (DP1 and DP2) assignments. Multiple terminals assigned on one conduction path are also supported; see single potential excitation (SP1, SP2, SP3) assignments in the figure that follows.



Note: Closed loop excitations are also applicable to single-frequency model export.

Transient T-Omega Boundaries and Excitations

Define at least one of the following as a source of magnetic fields:

- The stranded or solid windings, with a voltage or current supply or connected to an external circuit winding.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** For stranded conductors, it is assumed that the current density is uniform on the cross-section of the respective conductors. Thus, the stranded option for current excitations should mostly be used only in cases where the cross-section of conductors is constant, consistent with the assumption that the respective object is a coil built with strands of wire. For this reason, conductors (coils) could be created by either sweeping around an axis using zero segments or sweeping along a path having only true surface segments in order to create a smooth surface coil having a completely uniform cross-sectional area over its entire length.

- A permanent magnet.

You may need to set at least one outer boundary to the following:

- The default boundary conditions.
- An odd symmetry (flux tangential) boundary.
- An even symmetry (flux normal) boundary.

Transient T-Omega Boundaries

The transient field solver allows you to define the following types of boundary conditions.

Boundary Type	H-Field Behavior	Used to model...
Default Boundary Conditions (Natural and Neumann)	Field behaves as follows: • Natural boundaries – **H** is continuous across the boundary: The normal component of B and the tangential component of H (surfaces without superficial current distribution) are continuous. • Neumann boundaries – **H** is tangential to the boundary and flux cannot cross it. Typically, Neumann boundaries are placed at a sufficient distance from the sources of the field to avoid an over-constraint of the solution. • By default, the Boundary/Source Manager automatically applies the Neumann boundary to all faces of the problem region — that is, surfaces exposed to the non-meshed space.	Ordinary field behavior. Initially, object interfaces are natural boundaries; outer boundaries, and excluded objects are Neumann boundaries.
Zero Tangential H Field	The tangential components of **H** are set to zero. Flux is perpendicular.	External magnetic fields.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Integrated Zero Tangential H Field	For applications such as a motor with the shaft excluded, there remains a hole in the middle. In such cases, on the hole's boundary, neither the Tangential H Field boundary (where integration of the tangential H field is non-zero), nor the Zero Tangential H Field boundary (where the tangential H field is zero everywhere) can be applied because – for the hole boundary case – integration of the tangential H field is zero, but the tangential H field is not zero everywhere. Integrated Zero Tangential H Field can be used for such applications.	Cases such as the boundary around the hole that remains when a motor's shaft is excluded.
Insulating	Same as Natural boundary, except that current cannot cross the boundary.	Thin, perfectly insulating sheets between touching conductors.
Symmetry	Field behaves as follows: • Odd Symmetry (Flux Tangential) — **H** is tangential to the boundary; its normal components are zero. • Even Symmetry (Flux Normal) — **H** is normal to the boundary; its tangential components are zero.	Planes of geometric and magnetic symmetry.
Matching (Independent and Dependent)	The H-field on the dependent boundary is forced to match the magnitude and direction (or the negative of the direction) of the H-field on the independent boundary.	Planes of symmetry in periodic structures where **H** is oblique to the boundary.
Resistive Sheet	Same as Natural boundary	Voltage drop and loss on the contact surface between two different objects within a conduction path.
Impedance	Includes the effect of induced currents beyond the boundary surface.	Conductors with very small skin depths.
Touching	Same as Natural boundary	Interface between a moving part and a stationary part
Thin Layer (Air Gap)	When two parts of the domain are separated by a thin volume representing an air gap, it can be modeled by a surface with an appropriate boundary condition.	Narrow air gaps

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	This boundary creates a field discontinuity between the two sides of the selected surface based on its thickness. The relative magnetic permeability of the air gap is assumed to be equal to one.

Default Boundary Conditions

These boundary conditions are automatically defined for a transient model:

- Natural boundaries are assigned to the surfaces between objects.
- Neumann boundaries are assigned to the outside edges of the problem region.

To leave a surface set to its default boundary condition, you do not need to set any boundary conditions. Deleted boundary conditions and excitations automatically reset to the default boundary conditions.

Assigning a Zero Tangential H Field Boundary

For the transient solver, this type of boundary defines external magnetic fields in the model.

> **Warning:** When using zero tangential magnetic field boundary conditions, always double-check that Ampere's law is not violated!

To define a zero tangential H field boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).
2. Click **Maxwell** > **Boundaries** > **Assign** > **Zero Tangential H Field**.

   The **Zero Tangential H Field** window appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Click **OK** to assign the boundary to the selected object.

Assigning an Integrated Zero Tangential H Field Boundary

This type of boundary is applied when the integral of the tangential H field is zero.

> **Warning:** When using integrated zero tangential magnetic field boundary conditions, always double-check that Ampere's law is not violated!

To define an integrated zero tangential H field boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **Maxwell 3D** > **Boundaries** > **Assign** > **Integrated Zero Tangential H Field**.

   The **Integrated Zero Tangential H Field** window appears.

3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Click **OK** to assign the boundary to the selected object.

The new boundary is added to the boundary list in the project tree.

Assigning an Insulating Boundary

For the Transient solver, this boundary condition is used to model very thin sheets of perfectly insulated material between touching conductors. Current cannot cross an insulating boundary.

To set an insulating boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).
2. Click **Maxwell 3D** > **Boundaries** > **Assign** > **Insulating**.

   The **Insulating Boundary** dialog box appears.

3. On the **Insulating Boundary** tab, enter a name for the boundary in the **Name** box, or accept the default.
4. Click **OK** to assign the boundary to the selected object.

> **Note:** An insulating boundary condition can operate on the source current. It can also be used to control the flow of induced eddy currents.

Assigning a Symmetry Boundary

F or a Transient solver, this boundary condition defines a plane of geometric or magnetic symmetry in a structure. Assign it only to the outer surfaces of the problem region.

To set a symmetry boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).
2. Click **Maxwell** > **Boundaries** > **Assign** > **Symmetry**.
3. In the **Symmetry Boundary** window that opens, enter a name for the boundary in the **Name** box, or accept the default.
4. Select one of the following as the type of symmetry:

Odd (Flux Tangential)	H is tangential to the boundary; its normal components are zero.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Even (Flux Normal)	H is normal to the boundary; its tangential components are zero.

5. Optionally, click **Use Defaults** to revert to the default values in the window.
6. Click **OK** to assign the boundary to the selected object.

> **Warning:** When using even symmetry boundaries, always double-check that Ampere's law is not violated!

Assigning a Independent Boundary

Independent and dependent boundaries can be used to model planes of periodicity where the H-field at every point on the dependent boundary surface is forced to match the H-field of every corresponding point on the independent boundary surface. The transformation used to map the H-field from the independent to the dependent is determined by specifying a coordinate system on both the independent and dependent boundaries.

To set an independent boundary:

1. Select the face to which you want to assign the independent boundary.
2. Click **Maxwell** > **Boundaries** > **Assign** > **Matching** > **Independent**.
3. In the **Independent Boundary** window that opens, enter a name for the boundary in the **Name** box, or accept the default.
4. In the **Coordinate System** section, do the following to define the coordinate system:

   > **Note:** You must specify the coordinate system in the plane on which the boundary exists. First draw the U vector of the coordinate system. Maxwell uses the U vector you draw and the normal vector of the boundary face to calculate the v-axis. Then specify the direction of the V vector.

   a. Select **New Vector** from the **U Vector** drop-down menu.
   The **Create Line** message appears, asking you to draw the U vector of the coordinate system in the plane of the selected face, and the **Independent Boundary** dialog box disappears while you draw the U vector.
   b. Select the U vector's origin, which must be on the boundary's surface, in one of the following ways:
      - Click the point.
      - Type the point's coordinates in the in the **X**, **Y**, and **Z** boxes.
   c. Select a point on the u-axis.
   The **Independent Boundary** dialog box reappears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. To reverse the direction of the vector, select the **Reverse Direction** check box.
6. Click **OK**.

The Maxwell solver computes the H-field on this boundary and maps it to the dependent boundary using the transformation defined by the independent and dependent coordinate systems.

Assigning a Dependent Boundary

For the transient solver, assigning a dependent boundary is the second step in creating matching boundaries. The field on the independent boundary is mapped to the dependent boundary.

> **Note:** You must define an independent boundary before creating the dependent boundaries that are associated with it. Assign dependent boundaries only to the outer surfaces of the problem region.

To set a dependent boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).
2. Click **Maxwell** > **Boundaries** > **Assign** > **Matching** > **Dependent**.

   The **Dependent Boundary** window appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Select the independent boundary to which the dependent is to be assigned from the **Independent** drop-down menu.
5. To specify the coordinate system, do the following in the Coordinate System section:

   > **Note:** You must specify the coordinate system in the plane on which the boundary exists. First draw the U vector of the coordinate system. Maxwell uses the U vector you draw and the normal vector of the boundary face to calculate the v-axis. Then specify the direction of the V vector.

   a. Select **New Vector** from the **U Vector** drop-down menu.
   The **Create Line** message appears asking you to draw the U vector of the coordinate system in the plane of the selected face, and the **Dependent Boundary** dialog box disappears while you draw the U vector.
   b. Select the U vector's origin in one of the following ways:

      > **Note:** The U vector's origin must be on the boundary's surface.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click the point.
- Type the point's coordinates in the in the **X**, **Y**, and **Z** boxes.

c. Select a point on the u-axis in the same way.
When the second point is identified, the vector is set, and the **Dependent Boundary** window reappears, with **Defined** listed in the **U Vector** drop-down menu.

d. To specify the direction of the V vector, select or clear the **Reverse Direction** check box.

6. In the **Relation** section, select one of the following to specify the field behavior on the boundary:

Hdep = Hind	Click this radio button if the dependent and independent boundaries have the same magnitude and direction.
Hdep = -Hind	Click this radio button if the dependent boundary field has the same magnitude as but the opposite direction from the independent boundary field.

7. Optionally, click **Use Defaults** to revert to the default values in the window.
8. Click **OK** to assign the boundary to the selected object.

> **Note:** The origin must be a vertex point of one of the objects.

Assigning a Resistive Sheet Boundary

> **Note:**
>
> This topic explains how to assign a resistive sheet boundary for the transient solver.
>
> For AC Magnetic designs, see Assigning a Resistive Sheet Boundary for the AC Magnetic Solver.
>
> For 3D Magnetostatic designs, see Assigning a Resistive Sheet Boundary for the 3D Magnetostatic Solver.

When two conductors are in contact within a conduction path, a voltage drop normally exists across the contact surface due to the imperfection of the contact between the two conductors. This voltage drop should be taken into account when the winding is a voltage excitation. This boundary condition assigns a resistive sheet to consider the impact of this kind of voltage drop. The resistive sheet is defined by a lumped resistance in ohms.

To assign a resistive sheet boundary:

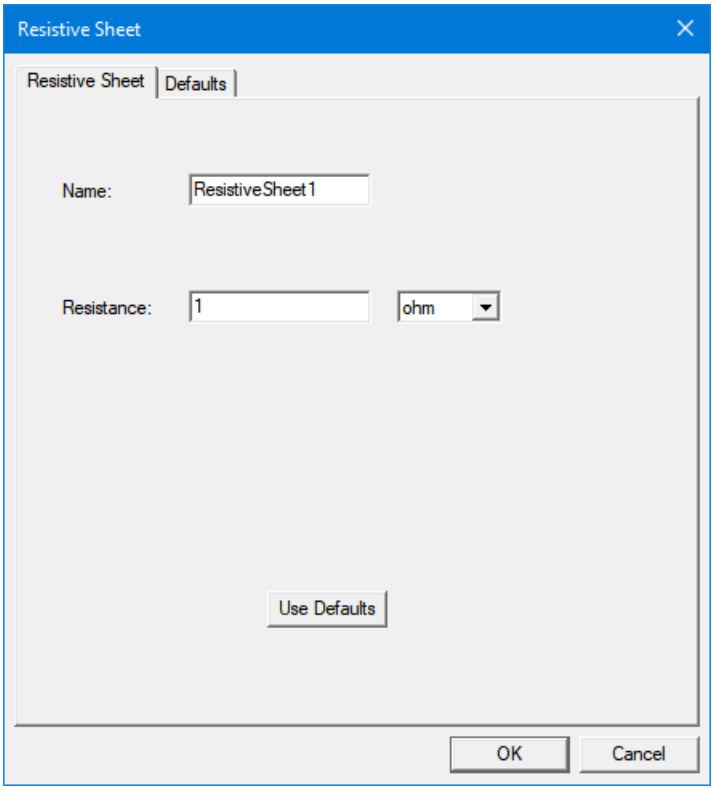
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the section of the geometry on which you want to apply the boundary condition (typically a sheet object or face of a 3D conducting body touching another conducting body).

   Note:

   - The sheet must have conductors touching both of its faces.
   - The sheet can not be smaller than the contacting surface.
   - The sheet cannot span multiple faces.

2. Click **Maxwell** > **Boundaries** > **Assign** > **Resistive Sheet**.
3. In the **Resistive Sheet** window that opens, enter a name for the boundary in the **Name** box, or accept the default.



4. Enter a resistance value in the **Resistance** box and select the unit of measure, or accept the default (1 ohm).
5. Optionally, the **Defaults** tab allows you to save the current resistance value for future use, and to revert to the standard defaults.
6. Click **OK** to assign the boundary to the selected object.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** When switching the solution type between transient and AC Magnetic, resistive sheet boundaries will be kept.

Assigning an Impedance Boundary

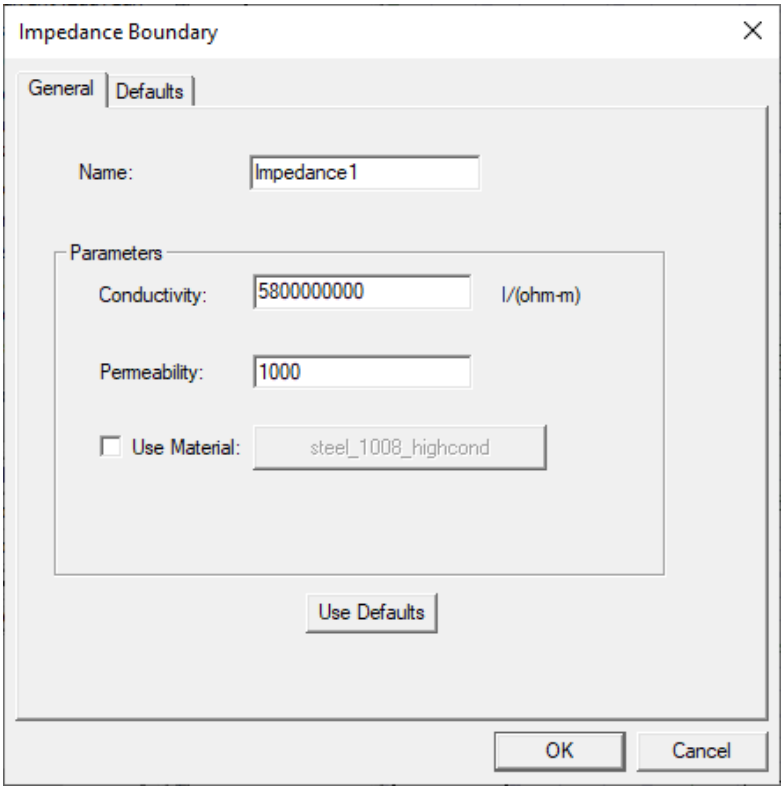
> **Note:** This topic explains how to assign an impedance boundary for the transient solver. To define an impedance boundary for the AC Magnetic solver, refer to Assigning an Impedance Boundary for the AC Magnetic Solver.

This boundary condition is used to simulate the effect of induced currents in a conductor without explicitly computing them. The conductor must be excluded from the model (saving time needed to mesh and solve for currents), so assign the impedance boundary condition to an outside edge of the problem region or to an excluded object.

To define an impedance boundary:

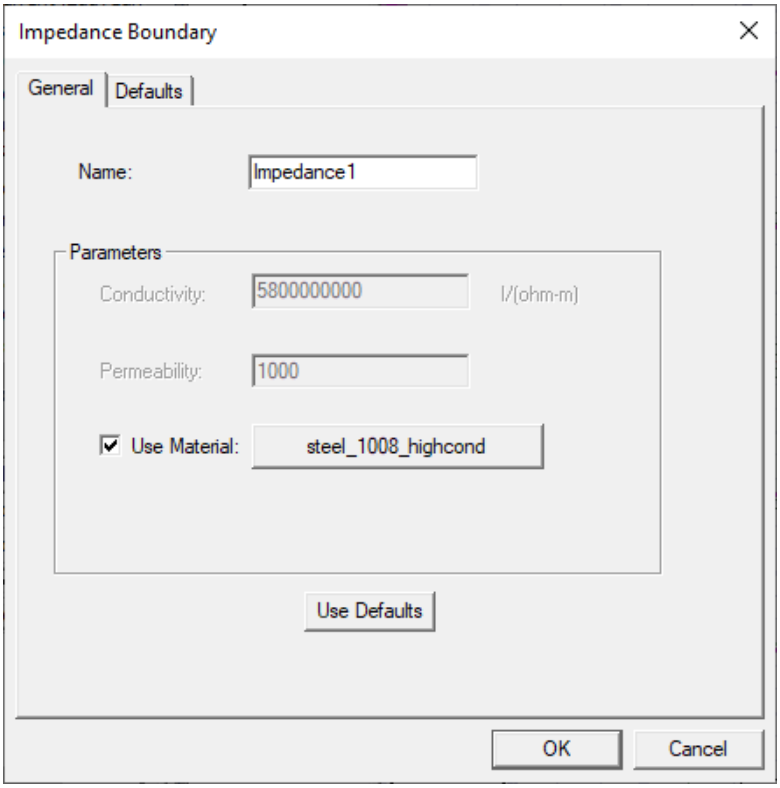
1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).
2. Click **Maxwell 3D** > **Boundaries** > **Assign** > **Impedance** to open the **Impedance Boundary** dialog box.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Do one of the following:
    - If the **Use Material** check box is unchecked, enter the conductivity (in inverse ohm-meters) in the **Conductivity** field, and the conductor's relative permeability in the **Permeability** field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- If you wish to select a material from the library, check the **Use Material** check box, then use the button to select the desired material.

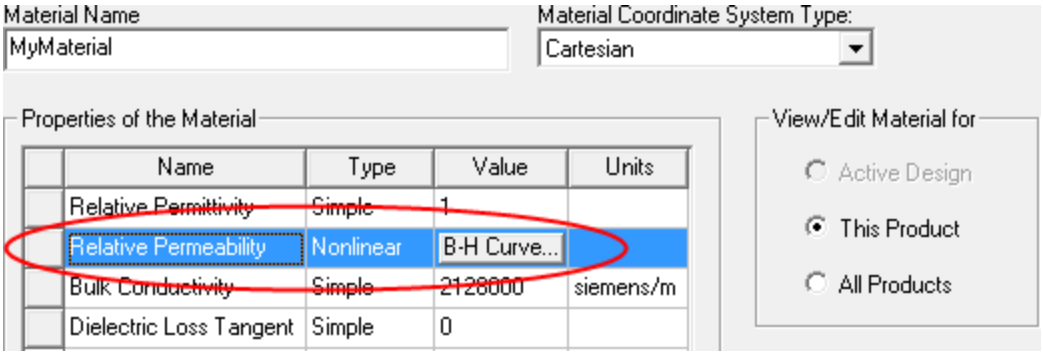
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Nonlinear Permeability**

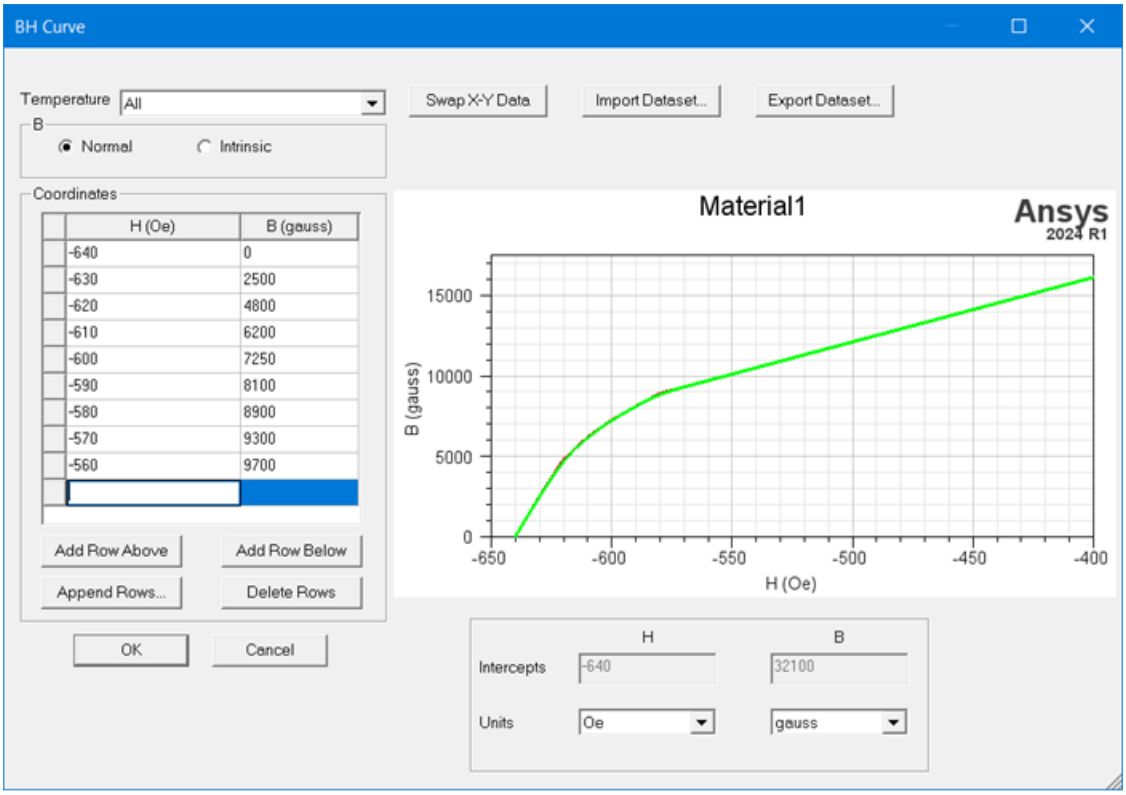
  If you select a material that has nonlinear permeability, the nonlinear coefficient is computed automatically.

  The nonlinear relative permeability of the material is determined by the B-H curve. If the desired B-H curve of the permeability belongs to an existing material, you can directly use that existing material.



  Otherwise, you can edit a new material with the desired B-H curve.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Refer to Relative Permeability for a Maxwell or RMxprt Material for details on using nonlinear relative permeability.

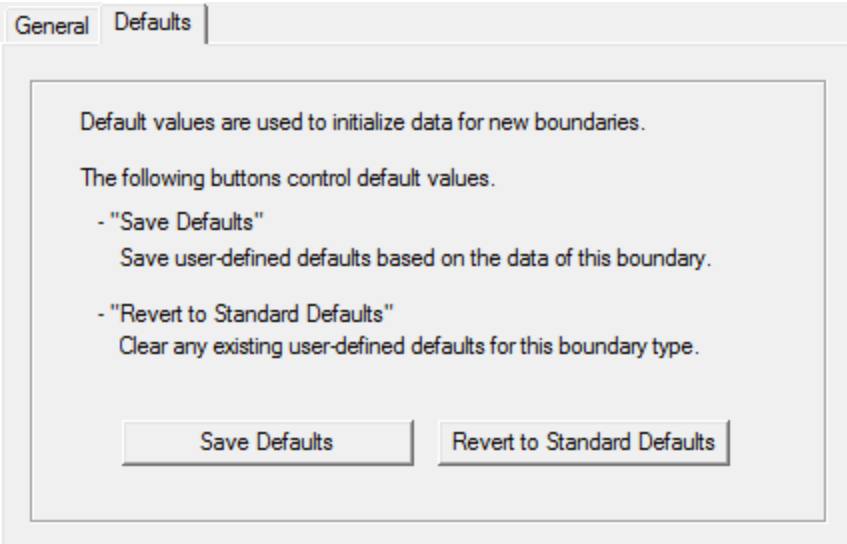
- **Temperature-Dependent Impedance Boundary**

  The electrical conductivity and permeability of a material used for an impedance boundary condition can be temperature dependent. The Maxwell 3D transient solver provides an option to allow you to input functions as thermal modifiers to consider the effect of temperature on those properties. To enable this feature, check the Use Material check box as above. You can then click Material to open the material properties panel in which you can add a thermal modifier. (Refer to *Setting the Temperature of Objects* for additional information on using temperature-dependent impedance boundaries).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Optionally, click **Use Defaults** to revert to the default values in the window.

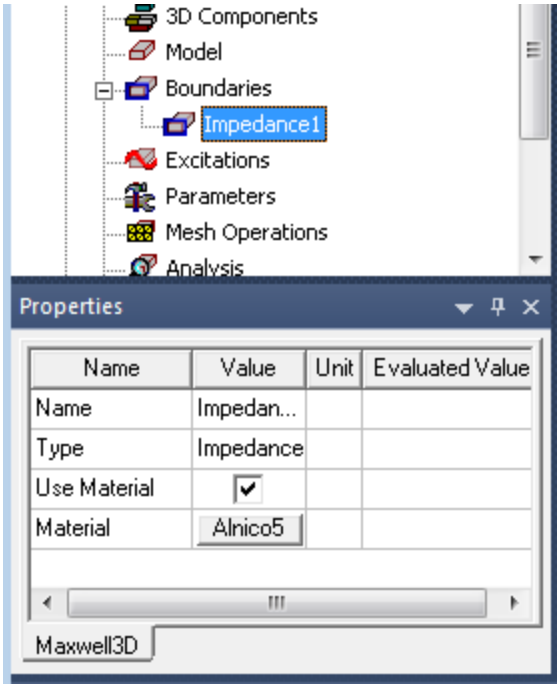
The **Defaults** tab allows you to control default values. The **Save Defaults** button saves the values currently defined on the **General** tab as the defaults to be assigned to new impedance boundaries. **Revert to Standard Defaults** clears existing user-defined values and replaces them with the standard default values.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Click **OK** to assign the boundary to the selected object.

   The Project Manager tree lists the newly assigned impedance boundary in the tree. You can select the boundary in the tree to view and edit its properties in the Properties window. You can also double-click the boundary entry in the tree to open it for editing in the Impedance Boundary dialog box.

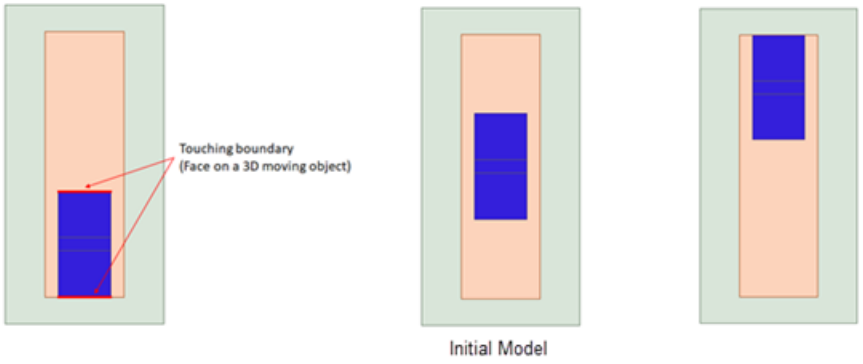


Assigning a Touching Boundary

For non-periodic translational motion designs in which a moving part contacts a stationary part this boundary condition assigns a touching boundary to the face of the moving part that contacts the stationary part. When the model is created, there must be an air gap between the moving objects and the stationary objects although these can touch during the simulation (either at beginning or end).

For example, with this feature, the linear actuator can be modeled from one end directly touching the stationary part to the other end also directly touching the stationary part as shown; however the initial model must be drawn without touching the ends as shown in the center image below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To assign a touching boundary to a non-periodic translational motion part:

1. Select the face of the geometry on which you want to apply the boundary condition (the face of the 3D moving body that will touch a stationary part).

   Note:

   - The touching boundary should be defined on the surface of the moving part.
   - The touching boundary can only be applied to the moving part with non-periodic translation motion.
   - At any timestep, each touching boundary must either fully touch the stationary part or be completely separated from the stationary part. The touching boundary can not partially touch the stationary part at any time.
   - The moving and stationary parts are not allowed to be on one conduction path.
   - Moving object(s) cannot touch the stationary objects in the initial model.

2. Click **Maxwell** > **Boundaries** > **Assign** > **Touching**.

   The **Touching Boundary** window appears.

3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Click **OK** to assign the boundary to the selected object.

Assigning a Thin Layer (Air Gap) Boundary for a 3D Transient Solver

When two parts of the domain are separated by a thin volume representing an air gap, it can be modeled using a **Thin Layer** boundary condition. This boundary creates a scalar potential discontinuity across the two sides of the surface based on its thickness. The magnetic permeability of the air gap is assumed to be equal to that of free space.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This boundary condition can be used to model narrow air gaps, such as those in power transformers, inductors, or any other electric device forming a magnetic circuit with an air gap without needing to explicitly model (and mesh) this gap.

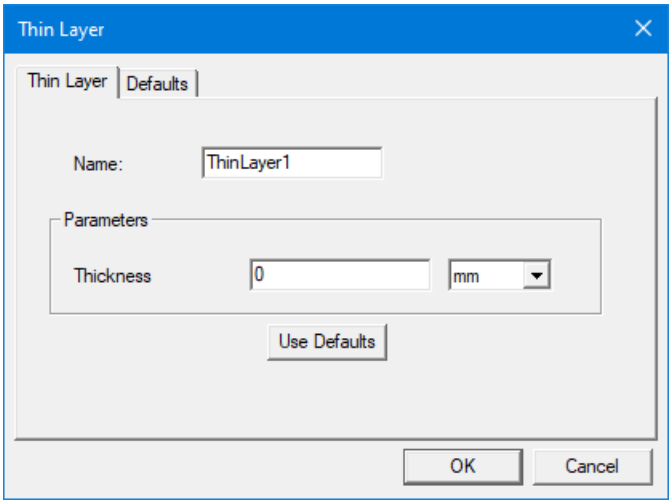
Note:

- The thin layer cannot be a closed shell encompassing a conductor.
- The thin layer does not support a mechanical movement, *i.e.*, the boundary condition cannot be applied to the interface of moving and stationary bodies.
- The thin layer cannot be defined on a conductor boundary or as a sheet object that touches or crosses any conducting body. In such cases, the Maxwell solver displays an error message indicating the name of the conducting body that the thin layer is touching.

For the transient solver, the thin layer for an air gap is defined purely in terms of its thickness. It is assumed to be made of air (non-conducting material with relative magnetic permeability equal to one).

To define a thin layer boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a sheet object or face of a 3D body).
2. Click **Maxwell 3D** > **Boundaries** > **Assign** > **Thin Layer** to open the **Thin Layer** window.



3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Enter the thickness of the thin layer and the unit of measure. It must be a value greater than zero. The use of a variable is supported, but it cannot be an intrinsic variable.
5. Optionally, click **Use Defaults** to revert to the default values in the window.

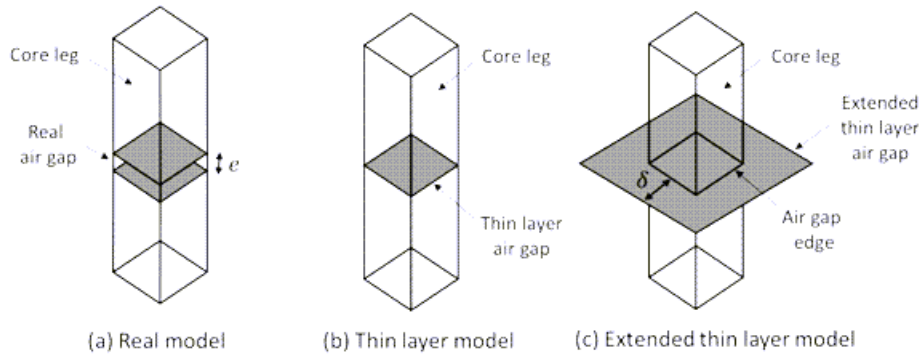
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. The **Defaults** tab allows you to control default values. The **Save Defaults** button saves the values currently defined on the **Thin Layer** tab as the defaults to be assigned to new impedance boundaries. **Revert to Standard Defaults** clears existing user-defined values and replaces them with the standard default values.
7. Click **OK** to assign the boundary to the selected object.

The Project Manager tree lists the newly assigned thin layer boundary in the tree. You can select the boundary in the tree to view and edit its thickness in the Properties window. You can also double-click the boundary entry in the tree to open it for editing in the Thin Layer dialog box.

Getting Better Convergence with the Thin Layer Model

For a transient simulation, the thin layer model for air gaps requires the duplication of the scalar potential to create a field discontinuity beyond the dimension of the sheet object and correctly represent the fringing flux effects. The Maxwell solver automatically duplicates the potential in one layer of elements from the gap edges. However, during the mesh adaptive process, the lateral elements may become too small and thus too close to the gap sides. This results in a low convergence of the mesh adaptive steps as the magnetic energy on such elements becomes more important. To improve convergence, the user can extend the sheet object manually (see figure below). Better convergence is observed by raising its lateral sides from one to three times the air gap thickness (δ≅3e) depending on the model geometry.



Transient T-Omega Excitations

After defining the coil terminals for the model, you can use these terminals to define one or more windings using the **Maxwell** > **Excitations** > **Add Winding** option. Transient problems use the following sources of magnetic fields:

Excitation	Type of Excitation
Coil Terminal	Used to define one or more model windings.
Winding With Current	Current for both a stranded and solid conductor.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Winding With Voltage	Voltage for both a stranded and solid conductor.
Winding With External Circuit Connection	External circuit connection for both a stranded and solid conductor.

> **Note:** A winding is a flexible excitation. Six different combinations can be used: Current, Voltage, and External Circuit — each with a solid or stranded conductor. The Coil Terminal type of excitation is needed to set up a winding.

> **Note:** When a source conductor touches an outer boundary, Maxwell assumes current will flow perpendicular to the boundary. If you do not want current to cross the outer boundary, an insulating boundary should be used.

In addition, permanent magnets serve as sources of magnetic fields.

Assigning a Coil Terminal

To assign a coil terminal as a transient excitation:

1. Select the section of the geometry on which you want to apply the excitation (typically a 2D planar object).
2. Click **Maxwell** > **Excitations** > **Assign** > **Coil Terminal**. (You can also right-click an existing winding in the project tree, and select **Assign Coil Terminal**.)
   The **Coil Terminal Excitation** dialog box appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the number of conductors for the coil terminal in the **Number of Conductors** box.

   > **Note:** This value represents only the number of conductors inside of the selected geometry. If the Coil Terminal is cut due to symmetry, then only enter the Number of Conductors in the portion modeled.

5. If you want to switch the direction for the coil terminal, click **Swap Direction**.

   > **Note:** The direction is shown in the model by a bold red arrow.

6. Optionally, click **Use Defaults** to revert to the default values in the window.
7. Click **OK** to assign the coil terminal to the selected object.

To add this coil terminal to an existing winding:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the Project Manager tree, right-click the coil terminal, and select **Add to Winding**.
2. In the **Add to Winding** dialog box that appears, select the winding to which you want to add the coil terminal, and click **OK**.

To remove a coil terminal from an existing winding:

- In the project tree, right-click the coil terminal, and select **Remove from Winding**. The terminal is removed from the winding and moved up one level in the project tree (directly beneath **Excitations**).

> **Note:** For 3D transient solutions using the A-Phi Formulation, refer to the Winding Excitations for A-Phi Formulation section for information on Ordering the Coil Terminals.

Assigning a Winding Setup

> **Note:**
>
> This topic explains how to assign a winding setup for a transient solver.
>
> - If you wish to define a winding for an AC Magnetic solution type, refer to Assigning a Winding Setup for an AC Magnetic Solver.
> - If you wish to define a winding for a Magnetostatic solution type, refer to Assigning a Winding Setup for a Magnetostatic Solver.

You can use the predefined coil terminal(s) to define one or more current or voltage windings.

To define a winding for your model:

1. Click **Maxwell 3D** > **Excitations** > **Add Winding** to open the **Winding** dialog box.
2. Enter a name for the winding in the **Name** box, or accept the default.
3. In the **Parameters** section:
    a. Select **Current, Voltage**, or **External** from the **Type** drop-down menu.

    > **Note:** Maxwell transient designs can be dynamically coupled to Twin Builder components through the Twin Builder user interface. You can enable this feature on the Advanced Product Coupling tab of the **Design Settings** dialog box. Source windings set to **External** are available as conservative pins in Twin Builder. For more information, refer to the Twin Builder Help.

    b. Select the **Solid** or **Stranded** radio button to specify the type of conductor.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Enter values in the following fields (enabled according to winding type), and select the desired units.

For a current winding	For a voltage winding[1]	For an external winding
Current	Resistance	Initial Current
	Inductance	
	Voltage	
	Initial Current	

> **Note:** You can also type a function as an expression for any of these fields except Initial Current.

d. Enter a value in the **Number of parallel branches** text box.

> **Note:** For the **Number of parallel branches** setting, the basic assumption is that the windings are symmetric and balanced, so the current split is forced to be equal in parallel branches. For solid or stranded windings, the **Number of parallel branches** setting assumes there are no circulating currents in the parallel branches.
>
> For solid windings, the **Number of parallel branches** setting is intended for applications meeting these conditions: For example, a periodic electric machine wedge model having solid hairpin windings, where the parallel branches are in the symmetric parts of the periodic model. In order to include unbalanced circulating currents this **Number of parallel branches** setting is not applicable, and you must use separate windings for each parallel branch that is connected in an external circuit.

4. Optionally, click **Use Defaults** to revert to the default values in the window.
5. Click **OK** to assign the excitation to the selected object.

A winding is now defined for your model.

To add a coil terminal to this winding, do the following:

1. In the project tree, right-click the winding, and select **Add Terminals**.
   The **Add Terminals** dialog box appears, listing all coil terminals that do not already belong to that winding.
2. Select the coil terminal (s) you want to add. To select multiple terminals, press **Ctrl** and click each terminal.
3. Click **OK**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To assign a coil terminal excitation and add it to this winding:

1. Select the section of the geometry on which you want to apply the coil terminal excitation.
2. In the project tree, right-click the winding, and select **Assign Coil Terminal**.
   The **Coil Terminal Excitation** dialog box appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the number of conductors for the coil in the **Number of Conductor**s box.

   > **Note:** This value represents only the number of conductors inside of the selected geometry. If the Coil Terminal is cut due to symmetry, then only enter the Number of Conductors in the portion modeled

5. Click **Swap Direction** to change the reference direction for the coil terminal. You can also change this direction in the coil terminal's Properties window by selecting either **Point into terminal**, or **Point out of terminal** for the **Direction** parameter.
6. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
7. Click **OK** to assign the coil terminal excitation.
   The coil terminal excitation is assigned and is added to the winding.

To delete all coil terminal excitations that belong to this winding:

- In the Project Manager tree, right-click the winding, and select **Delete All Terminals**.
  All coil terminal excitations are removed from the winding and deleted from the excitations.

> **Note:** For 3D transient solutions using the A-Phi Formulation, refer to Winding Excitations for A-Phi Formulation for information on ordering the coil terminals.

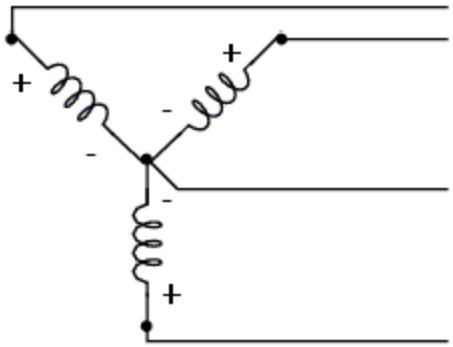
---

1. For a solid winding, the resistance term can represent: the resistance of a portion of the winding which is not modeled (for instance, end-effects), the leads connecting the winding to the source, or the source resistance. (The main winding resistance is calculated directly by the solver.) For a stranded winding, the resistance term is the complete DC resistance of the winding (since the solver does not determine resistance of a stranded winding) as well as the resistance of the end-effects, leads, source, etc. For both solid and stranded windings, the inductance term can represent: the extra inductance for a portion of the winding of the winding which is not modeled (for instance, end-effects), the leads connecting the winding to the source, or the source inductance. (The main winding inductance itself is calculated directly by the solver.)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Up a Y Connection

The Y Connection function available in 2D and 3D Transient solution types allows multiple windings to be connected in a classical Y (sometimes referred to as wye) configuration with the negative terminals connected to a common node as illustrated below.



Setup Y Connection requires voltage winding definitions be set and the negative voltage terminal for each winding will be connected to form the common node of the circuit.

Each voltage source specified in the winding description drives the positive side of the coil. Y Connections are commonly used in motor and transformer applications driven by three-phase voltage; however, any number of windings may be connected to a common node in Maxwell.

Note:

- Only one Y Connection is allowed in a given Maxwell design.
- For 3D designs, only stranded voltage windings in a Y-connection are supported.
- For 2D transient designs, both stranded and solid windings are supported, but a mix of stranded winding and solid winding in one Y-connection group is not allowed.
- A mix of PWM voltage winding and (normal) voltage winding in one Y-connection group is not allowed.
- Duplicate windings are not allowed.

To define a Y-connection:

1. Click **Maxwell 2D** (or **Maxwell 3D**) > **Excitations** > **Setup Y Connection**.

   The **Setup Y Connection** dialog box appears.
2. In the **Winding** list on the left, select the windings to be grouped into a Y connection.
3. Click the **Group** button to create a Y Connection group with the selected windings.
4. Click **OK**.

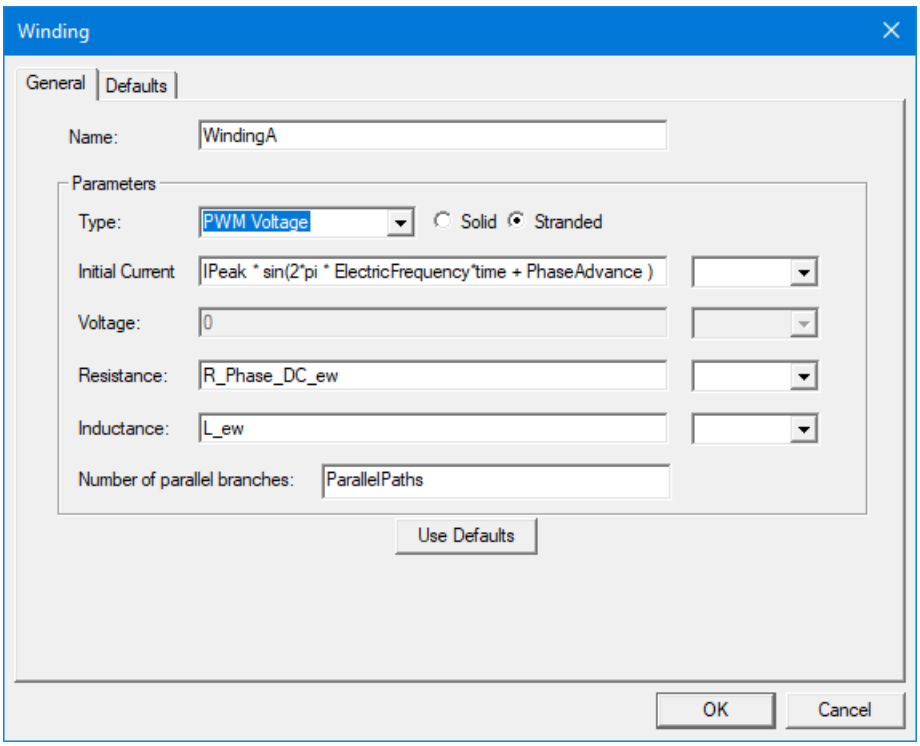
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Up an SVPWM Excitation

Pulse Width Modulation (PWM) is a term used for any method that produces a square wave signal with varying duty cycle and frequency to control the average power delivered to a load. PWM is widely used in applications such as light dimming, switching power supplies, and electric motor drive. In these types of designs, the AC motor needs to be powered by an alternating current excitation, which is supplied by the DC supply voltage through an inverter circuit using power device switches. Controlling how the power switches are turned on and off is accomplished by PWM.

For 3D and 2D magnetic transient T-Omega solutions, Space Vector Pulse Width Modulation (SVPWM) can be directly created in Maxwell for three-phase voltage-excited windings. For all three phases, Maxwell will calculate the PWM voltage waveforms based on user inputs and automatically set up the winding excitations in the transient simulations. The generated waveform represents the Phase-Neutral voltage, and it is applicable to Y(star)- or delta-connected winding.

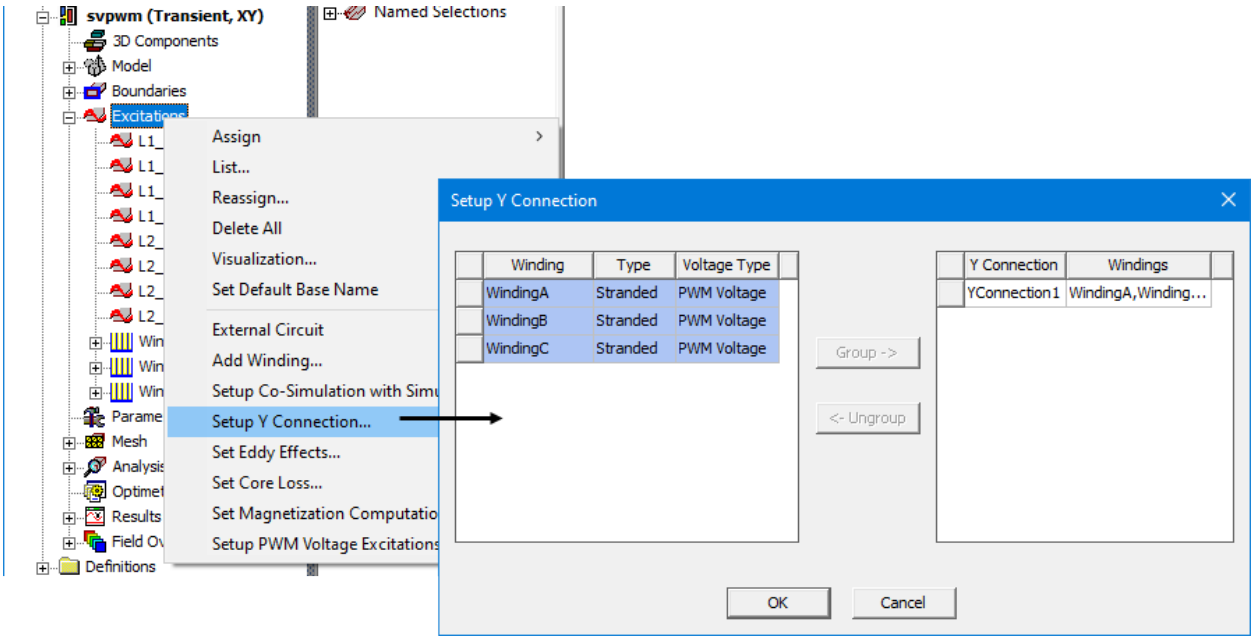
1. Before setting up the SVPWM excitation:
    a. Define three PWM voltage windings. For the **Type** option in the Winding window, make sure to select **PWM Voltage**:



Note that when the **PWM Voltage** type is selected, the Voltage field is grayed out because it will use the data generated by PWM voltage excitation setup.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. If you will be using a Y connection for the winding, right click on the **Excitations** entry in the Project Manager, and select **Setup Y Connection**. Select the three windings, and click **Group**.



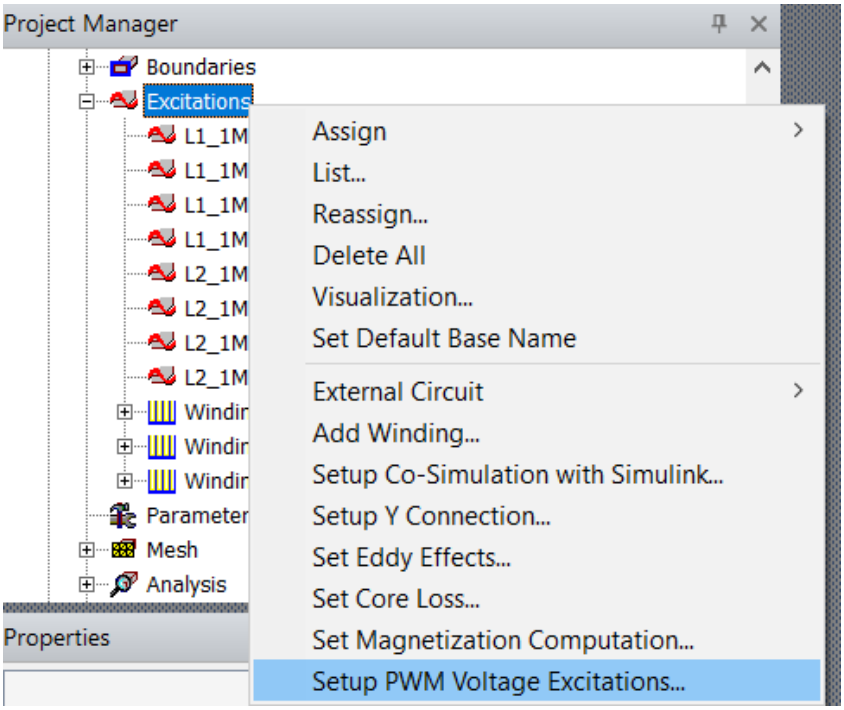
Note:

- For 3D transient designs, only stranded windings are allowed for the Y-connection.
- For 2D transient designs, both stranded and solid windings are supported, but a mix of stranded winding and solid winding in one Y-connection group is not allowed.
- A mix of PWM voltage winding and (normal) voltage winding in one Y-connection group is not allowed.
- Only one Y Connection is allowed in a given Maxwell design.

2. In the Project Manager tree, right click **Excitations**, and select **Setup PWM Voltage Excitations**.

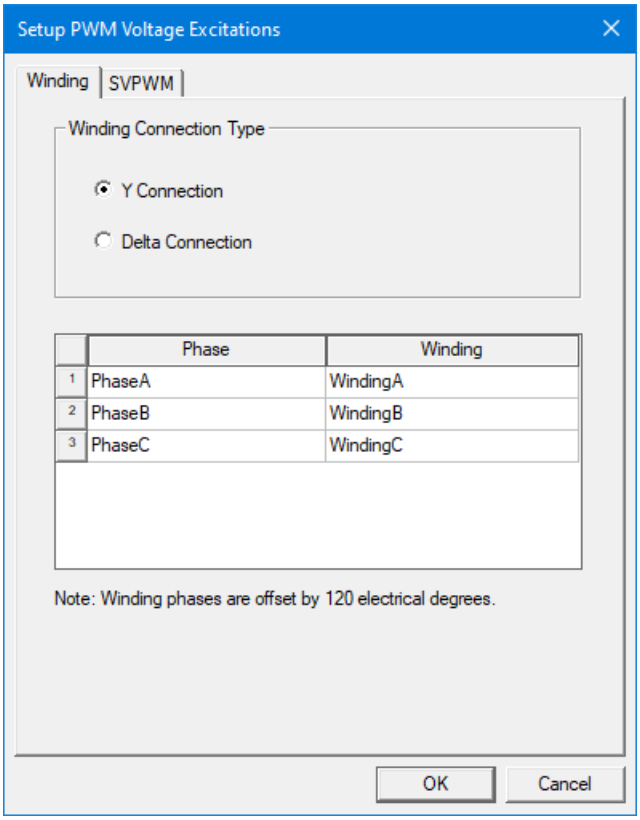
   Alternatively, from the menu bar, select **Maxwell 2D/3D** > **Excitations** > **Setup PWM Voltage Excitations**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. In the Setup PWM Voltage Excitations window that opens, select either the **Y Connection** or the **Delta Connection**, and arrange the three windings to represent the three phase windings (PhaseA, PhaseB and PhaseC) accordingly. Note that the **Winding** drop-down menu will only include windings that were defined as the **PWM Voltage** type.

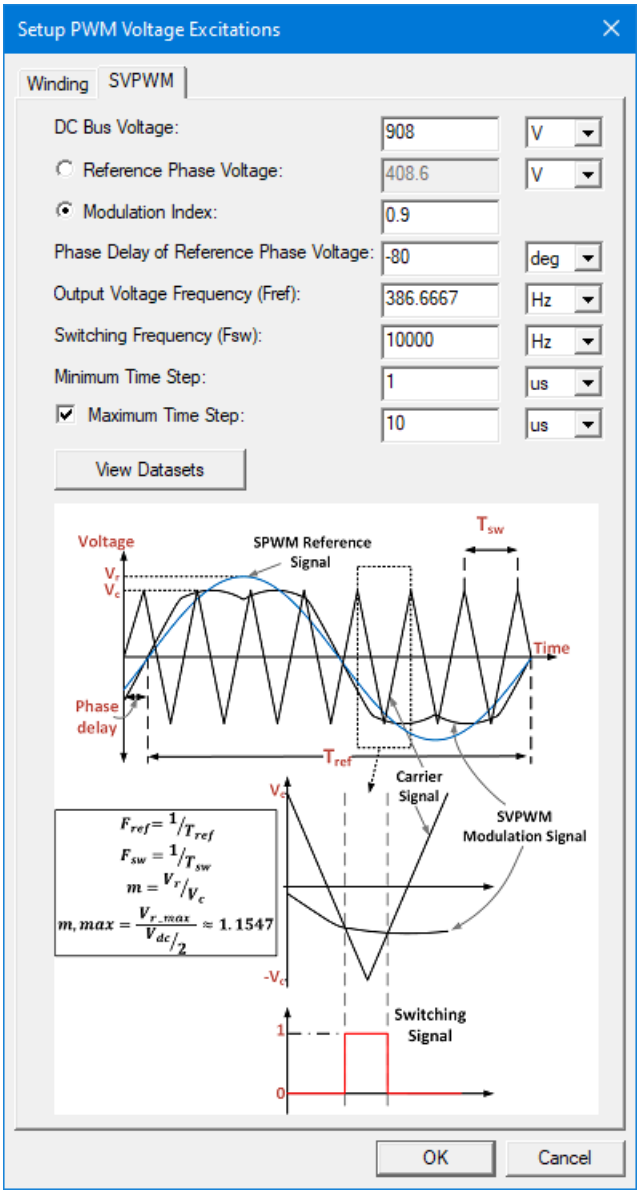
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:** For Delta Connection winding, the PhaseA, PhaseB, and PhaseC voltages are different than for the Y connection winding: if voltage values for Y connection are Va, Vb, Vc, then the voltage values for delta connection are Va-Vb, Vb-Vc, Vc-Va.

4. Click on the **SVPWM** tab, and input the corresponding parameters for SVPWM. Example parameters are shown below:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note: All the parameters of the Setup PWM Voltage Excitations window can be defined with a variable **except** Modulation Index.

- **DC Bus Voltage** – Specifies the voltage available at the main DC bus as input to the inverter.
    - **Reference Phase Voltage** – Defines the reference voltage signal, seen as the blue curve (labeled SVPWM Reference Signal) in the diagram in the Setup PWM Voltage Excitations window. If this option is selected, the **Modulation Index** setting is grayed out and automatically calculated

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

according to ModulationIndex = ReferencePhaseVoltage/(DCBusVoltage/2).

- **Modulation Index** (m) – Specifies the ratio between the reference signal to the carrier signal; it determines the amplitude of the fundamental output voltage with respect to the available DC bus voltage as described by the equation below.

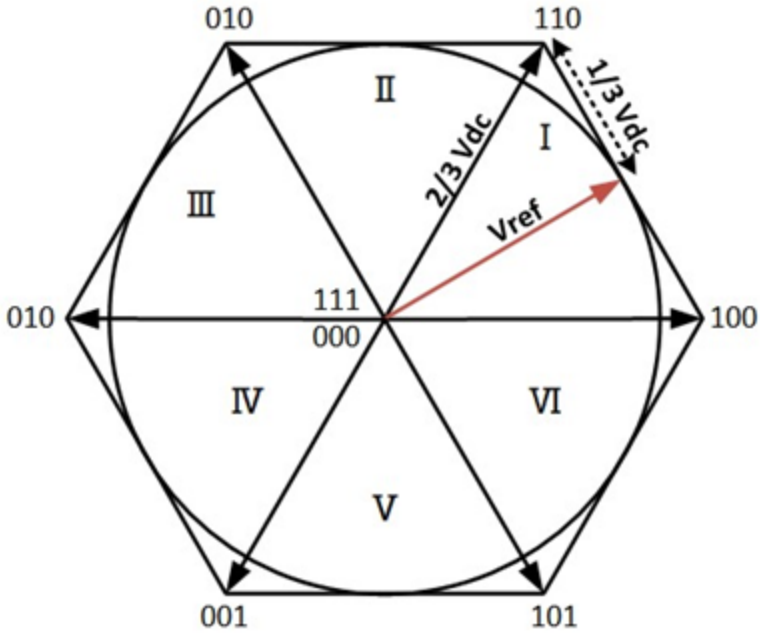
$$V_{peak,\ Phase-Neutral} = m * \frac{V_{DC}}{2}$$

The range of the modulation index is 0 < modulation index ≤ 2/sqrt(3) (*i.e.*, approx. 1.1547) for SVPWM.

> **Important:** For a small modulation index, care must be taken when setting the minimum time step to control numerical error.

If this option is selected, the **Reference Phase Voltage** setting is grayed out and automatically calculated according to ReferencePhaseVoltage = ModulationIndex*(DCBusVoltage/2).

To understand how the maximum modulation index is calculated, start with the figure below, which shows the space vector hexagon (represented by six sectors and eight vectors). Each vector represents a different switching state of the inverter.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The vector V_{ref} is the reference revolving voltage vector formed from the three phase sinusoidal voltages; at each sector, this vector is realized by alternating between the two nearest space vectors along with the zero vectors.

The circle inside the hexagon represents the maximum value possible for V_{ref} (and hence the output voltage) that can be produced with the SVPWM technique.

Knowing that the magnitude of each vector is 2/3 V_{DC}, the maximum value of the output voltage (V_{ref}) can be computed as

$$V_{RefMax} = \sqrt{\left(\frac{2}{3}V_{dc}\right)^2 - \left(\frac{1}{3}V_{dc}\right)^2}$$

$$V_{RefMax} = \frac{\sqrt{3}}{3}V_{DC}$$

From these definitions, the maximum modulation for SVPWM can be derived as

$$m_{Max} = \frac{V_{RefMax}}{\frac{V_{DC}}{2}} = \frac{2*\sqrt{3}}{3} \approx 1.1547$$

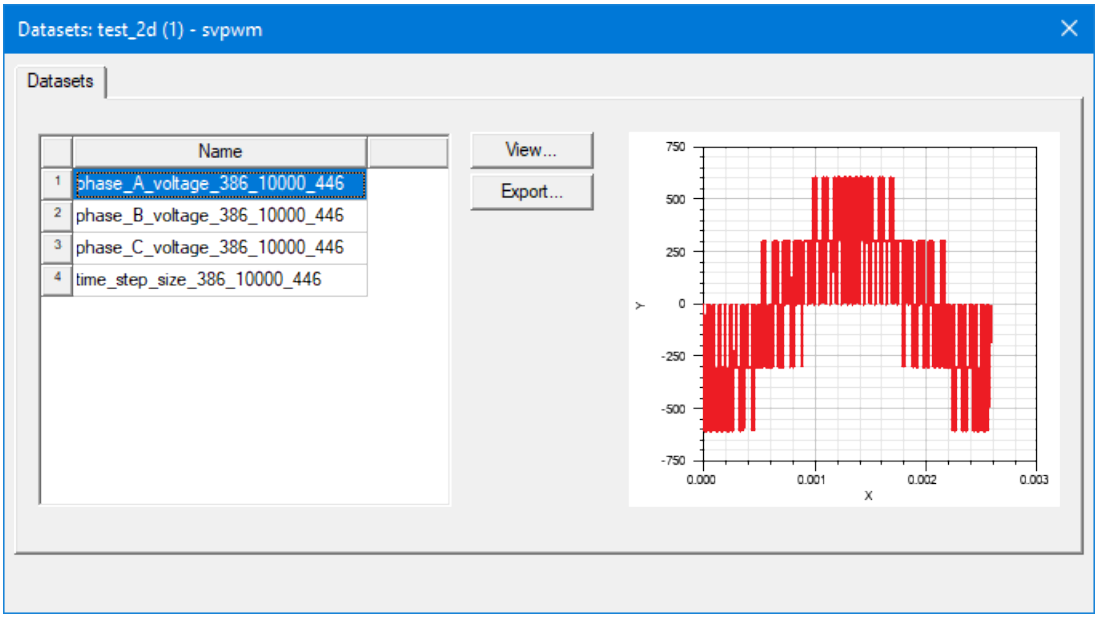
- **Output Voltage Frequency (Fref)** – Defines the fundamental frequency of the reference signal, and therefore, the PWM output voltage.
- **Switching Frequency (Fsw)** – Defines the carrier signal frequency that is used to generate the PWM switching signal. A typical value for traction inverters is between 10 – 20 kHz.
- **Phase Delay** – Defines the phase delay angle of the reference signal (and hence the output voltage). The phase delay depends on the operating point, and it can be provided from system-level simulation or by running a transient simulation with an ideal sinusoidal current and checking the phase delay on the output of the phase voltage.
- **Minimum Time Step** – Defines the rate at which the carrier signal and the reference signal are compared to generate the switching signals. This value is important to correctly capture the switching events. The recommended value is 1/(Fsw*100) or smaller.
- **Maximum Time Step –** Optional setting that specifies the largest time step that can be used to generate the switching signal; it is used to modify the time step dataset; see the explanation that follows. If left unchecked, there will be no maximum time step considered when the time step dataset is created. If checked, the default value is 10*Minimum Time Step.
- **View Datasets –** Click to view the new datasets generated by the PMW Voltage Excitation setup.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Three datasets of the voltage (Phase_X_voltage_<Fref>_<Fsw>) vs. time are generated based on the settings. The datasets are generated by comparing the reference signal to the triangular carrier at a sampling frequency defined by the minimum time step. Subsequently, the datasets are reduced by removing the timesteps where no change of the voltage values occurs.

The datasets represent the three normalized PWM phase voltages for one electrical cycle. The phase sequence of the generated dataset is A-B-C with Phase B and C lagging Phase A with 120 deg and 240 deg, respectively. If a variable was used to define a setting, the View Dataset function will only display nominal values.

A time step dataset is also created. Note that when creating the time step dataset, the solver considers the Maximum Time Step (*dt*), and it will add *N* more time steps (equal spaced) between the two time stamps Y(n-1) and Y(n) (say *deltaT* = Y(n) - Y(n-1)) when *deltaT* > *dt*, and *N* = floor(*deltaT*/*dt*). For example, with timeStep = [1.7, 2, 3.3] and *dt* = 0.4, then newTimeStep = [1.7, 2, 2.325, 2.65, 2.975, 3.3]. Three new time steps are added: 2.325, 2.65, and 2.975, and they are equally spaced between 2 and 3.3.



Note: The Export button can be used to export the datasets as design datasets.

5. Click **OK**.

The technique used by the Maxwell application to implement SVPWM excitation computes the pattern of voltage pulses for each of the three datasets by comparing a triangular carrier wave with frequency Fsw and amplitude 1 to three reference functions.

Unlike for sinusoidal PWM, where the reference functions are sinusoidal, for this SVPWM implementation, the three reference functions (V_{a_Ref}, V_{b_Ref}, V_{c_Ref}) need to be constructed as

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a first step. This is achieved by superposing the common mode voltage (V_{CM}) to the three sinusoidal signals (V_{a_0}, V_{b_0}, V_{c_0}) of a three-phase system:

$$V_{a_Ref} = V_{a_0} + V_{CM}$$

$$V_{b_Ref} = V_{b_0} + V_{CM}$$

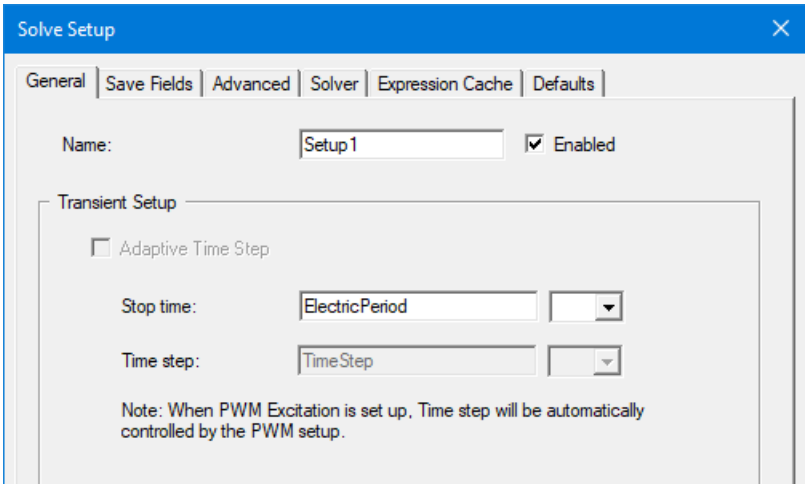
$$V_{c_Ref} = V_{c_0} + V_{CM}$$

V_{a_0} is defined as ModulationIndex*sin(2*pi*Fref*time-Phase Delay). V_{b_0} and V_{c_0} lag V_{a_0} by 120° and 240°, respectively.

The amplitude V_{max} can be calculated as (DC Bus Voltage)/2 * Modulation Index.

At every timestep, the common mode voltage is calculated as half of the mid-valued function among V_{a_0}, V_{b_0}, V_{c_0}.

The dataset time_step_size_<Fref>_<Fsw> tracks the exact instance the switching happens and will be used to define the variable time step in the transient setup to ensure that all switching events are captured with the minimum number of time steps possible. This dataset will be used in the Solve Setup Time step value field, so the **Time step** value field is grayed out. For a 2D analysis, the **Adaptive Time Step** check box is grayed out as well.



Setting Up an External Circuit Connection for an External Winding Type

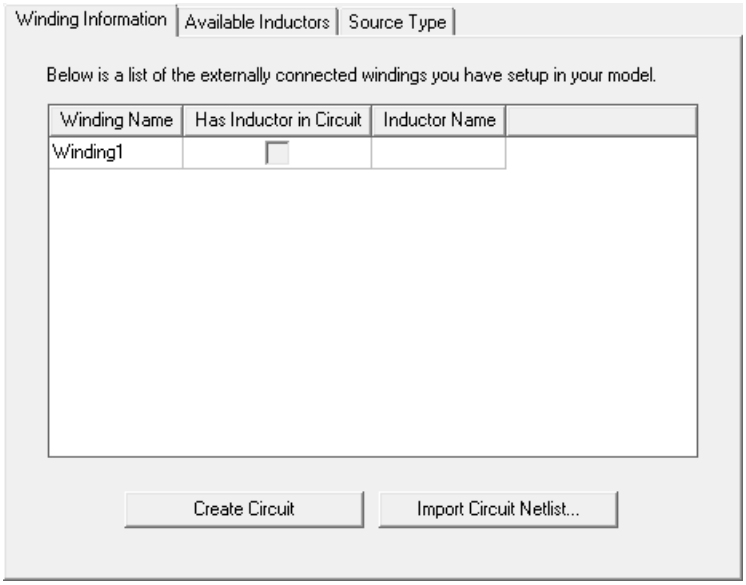
The Maxwell Circuit design editor is used to set up the external circuit excitation for windings in Maxwell 2D and 3D transient solutions.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: External circuits intended for use with the Time Decomposition Method (TDM) can only support basic circuit elements that are solution-independent. This is because, for TDM, all timesteps (or all timesteps in a subdivision) are solved simultaneously - and thus the circuit topology and its parameters must be known before solving. Therefore some circuit elements, such as solution-dependent switches, diodes, capacitors, and inductors which are not directly connected to windings in series, are not supported.

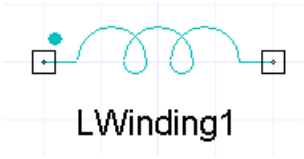
To set up an external circuit connection for an External winding type:

1. Click **Maxwell 3D** > **Excitations** > **External Circuit** > **Edit External Circuit** to open the **Edit External Circuit** dialog box. You can also right-click on **Excitations** in the Project tree and select **External Circuit** > **Edit External Circuit**.
    a. If no external winding connections have been set up for your model, the **Winding Information** tab lists the external winding(s) in the design. Continue with step 2.

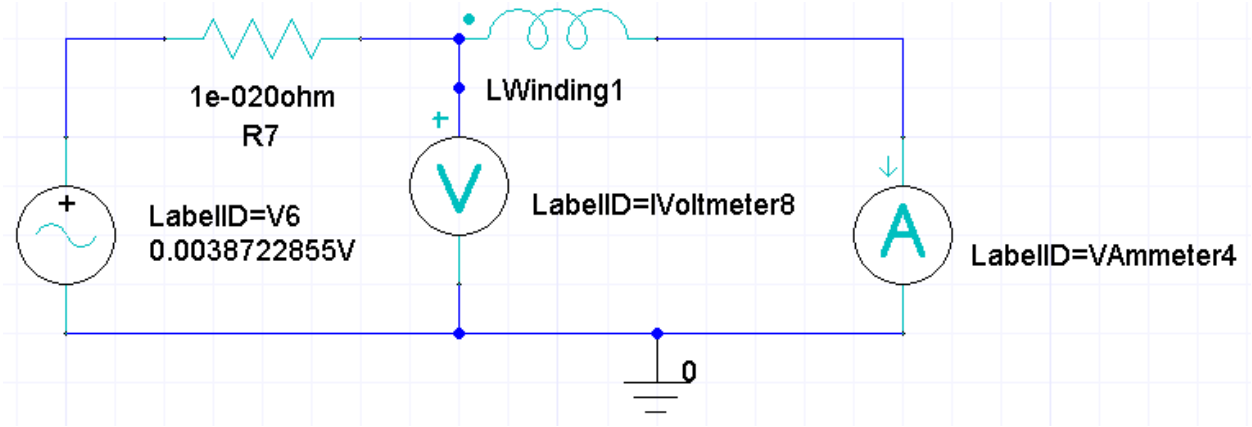


    b. If external windings have already been set up, continue with step 7.
2. On the **Winding Information** tab, click **Create Circuit** to add a circuit design to the project. The new circuit design automatically opens for editing in the Maxwell Circuit Editor. The circuit includes one inductor for each external winding in the original Maxwell transient design. The following example is for a design having one external winding.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Add the components needed to complete the external circuit design such as shown the following example.



4. When the circuit design is finished, click **Maxwell Circuit** > **Export Netlist** and export the netlist **.sph** file to the desired location.
5. Return to the Maxwell design and click **Maxwell 3D** > **Excitations** > **External Circuit** > **Edit External Circuit** to open the **Edit External Circuit** dialog box. You can also right-click on **Excitations** in the Project Manager tree and select **External Circuit** > **Edit External Circuit**.
6. On the **Winding Information** tab, click **Import Circuit** to import the circuit.
7. To view a list of inductors in the imported circuit, click the **Available Inductors** tab.
8. To view the sources used in the externally connected windings, click the **Source Type** tab.
9. If variables are used in the imported circuit, they and their design values are listed on the **Parameter Values** tab. You can link to variables within the imported circuit as follows:
    a. On the **Parameter Values** tab, click in the **Value** column of the parameter to be mapped.
    b. Enter a new local variable name and press the `Enter` key to open the **Add Variable** dialog.
    c. Select a **Unit Type** and **Unit**, and enter a **Value** for the variable and click **OK**. The new Maxwell variable is now mapped to the variable in the imported circuit, and can be varied directly by Maxwell or used for optimetrics analyses.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10. If the imported file is a .**sph** file, you can click the **Circuit Path** tab to view the original project and design names (which are removed after you import a non-sph file).

    > **Note:** After you import a circuit netlist, the **Create Circuit** button changes to **Update Netlist from Circuit**. Use **Update Netlist from Circuit** to reimport the netlist to keep it current.

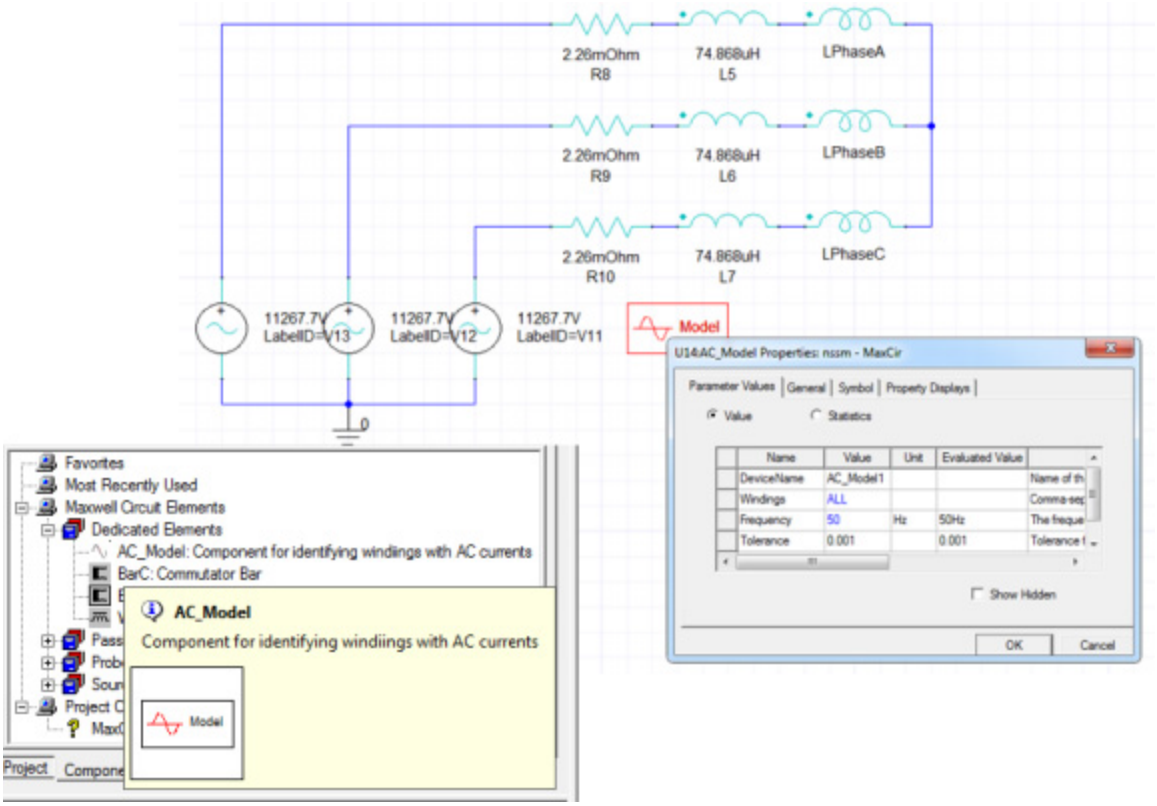
11. Click **OK** to close the **Edit External Circuit** dialog box.

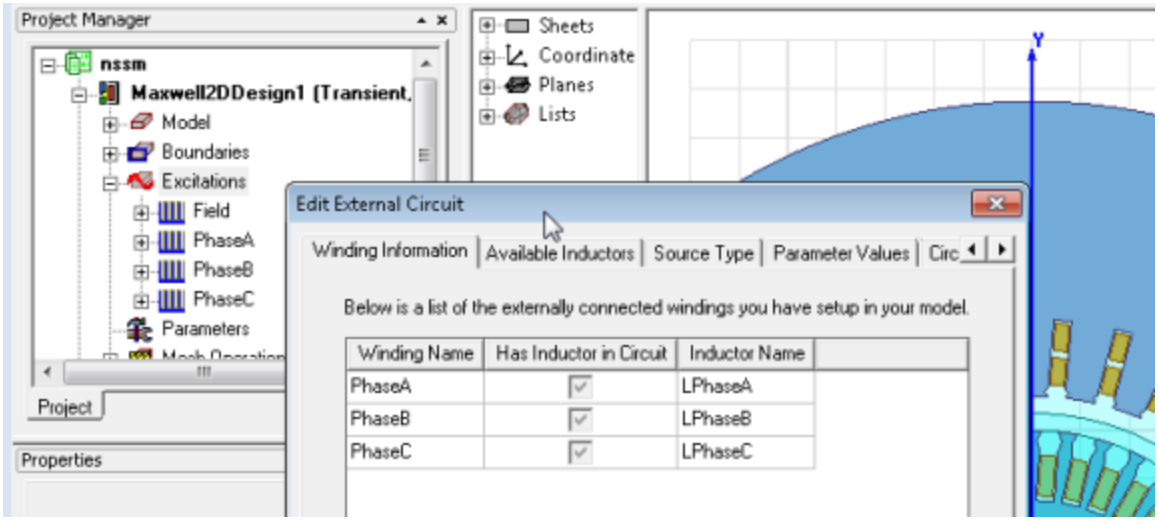
> **Note:**
>
> For transient designs, the user-set timesteps can be modified when external circuits are used to drive the windings of a transient finite element model. Following is a list of situations that will lead to a timestep change:
>
> - All power electronic switching instances that do not coincide with user-specified solve times request a new solution time from the transient solver. The respective switching generating a new solve time request for the transient solver can be time, position, or speed dependent.
> - When current and/or voltage sources with a piecewise linear variation are used, a new solution time is requested from the transient solver at each (time) definition point used in the corresponding source definition table.
> - When the change in any winding inductance value is excessive, a new timestep (smaller) is calculated and a new solution generated (recalculated) accordingly.
>
> If a user-specified save field time is missed because of the above reasons, the next solved timestep fields are saved instead.

Using the AC_Model Component to Speed Up Analysis

Use of the AC_Model component when setting up an external circuit design can improve transient analysis time to steady state significantly.

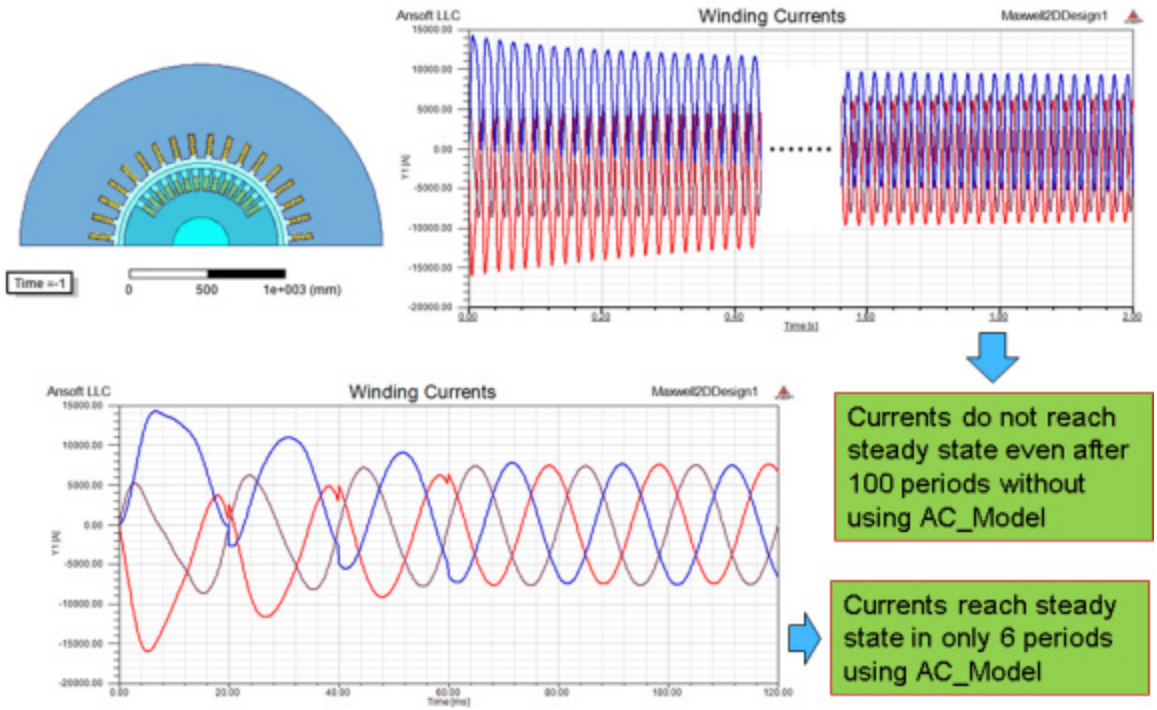
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





In the above example, the AC_Model is associated with ALL three of the phase windings. The AC_Model modifies currents for the AC windings at the beginning of each period without interrupting the transient process - thus improving the analysis time.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Setting Up an External Circuit Using Simulink

In addition to using the Maxwell Circuit Editor to set up an external circuit, you also can use MATLAB® Simulink® to set up an external circuit for both Maxwell 2D and 3D transient designs.

Supported Simulink versions are

- R2019b
- R2020a
- R2020b
- R2021a
- R2021b
- R2022a
- R2022b
- R2023a

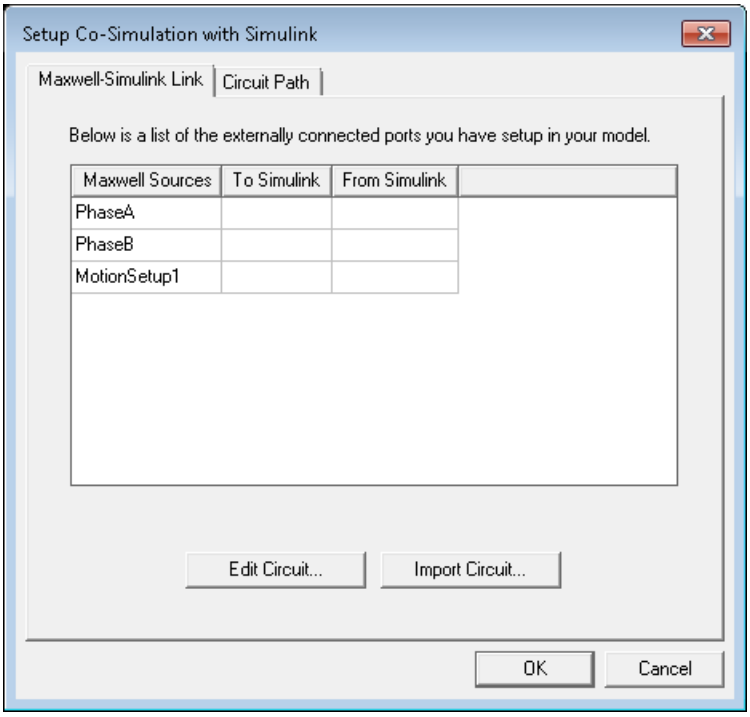
In addition to winding sources, motion sources can also be linked in the external circuit.

- Winding sources ("External" type only): voltage, current, and flux linkage.
- Motion sources: speed, position, and force/torque

To set up an external circuit using Simulink:

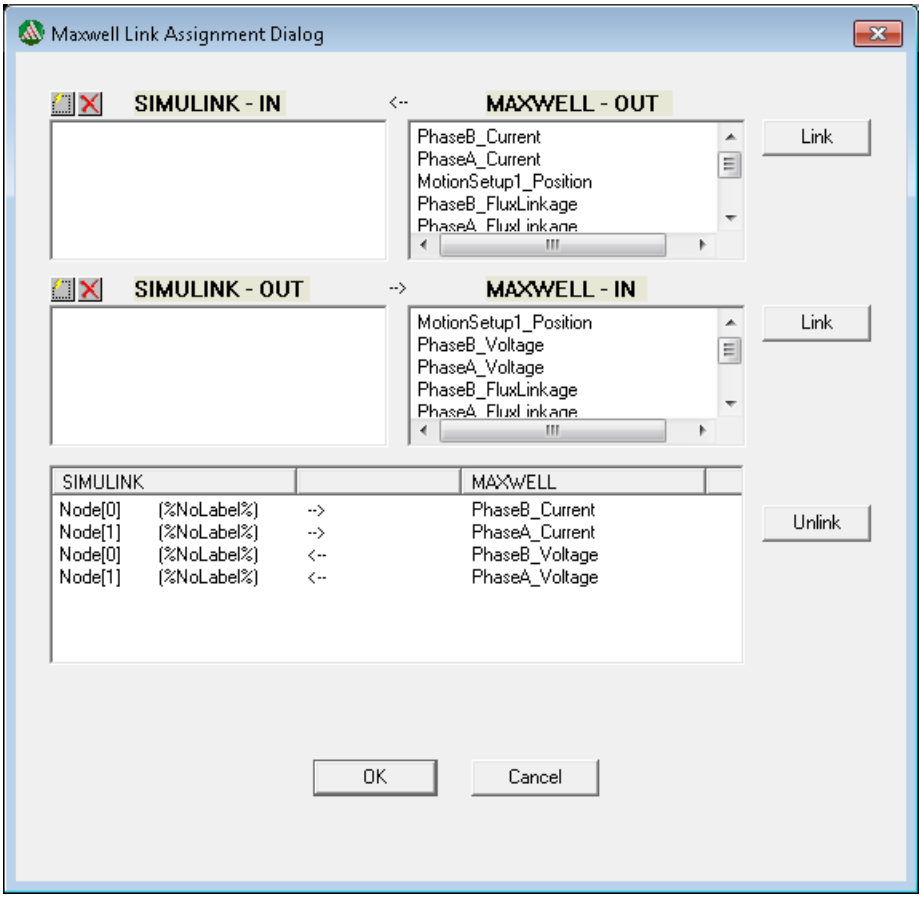
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the Maxwell Project Manager, right-click **Excitations** and select **Setup Cosimulation with Simulink** to open the **Setup Cosimulation with Simulink** dialog box. You can also open the dialog box from the main **Maxwell 2D** or **3D** > **Excitations** menu. If an external Simulink circuit has already been created and linked, **To Simulink** and **From Simulink** connections information is shown. Otherwise, these table cells are empty.



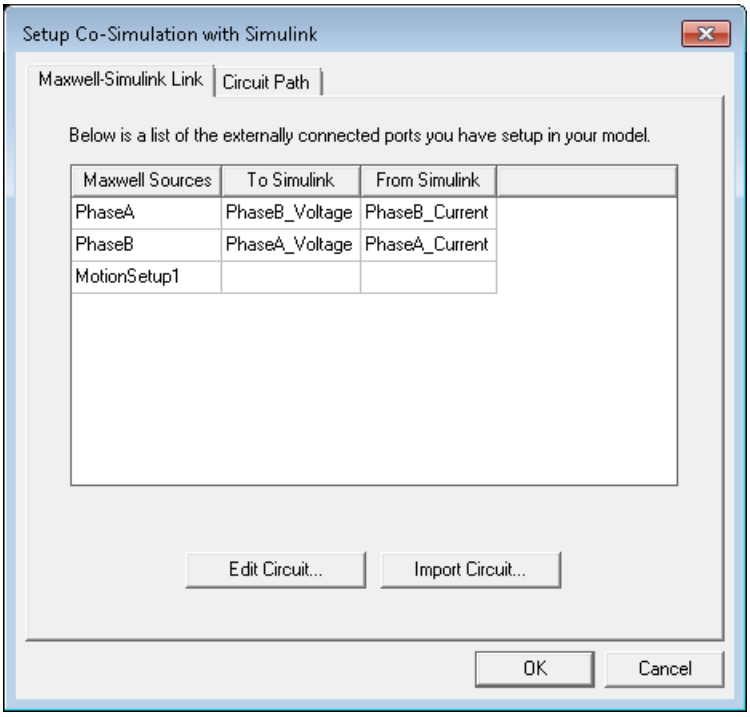
2. On the Maxwell-Simulink Link tab, click the **Edit Circuit** button to launch Simulink. The Maxwell sources are listed in the in the **Maxwell Link Assignment Dialog** box. If a circuit has already been created, the circuit also displays in the Simulink circuit editor window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Add **Link** SIMULINK - IN and SIMULINK- OUT connections to the desired Maxwell parameters to set up the Maxwell component ports. You can also **Unlink** connections as needed. Click **OK** once the ports are set up. The ports will show in the Simulink circuit editor window.

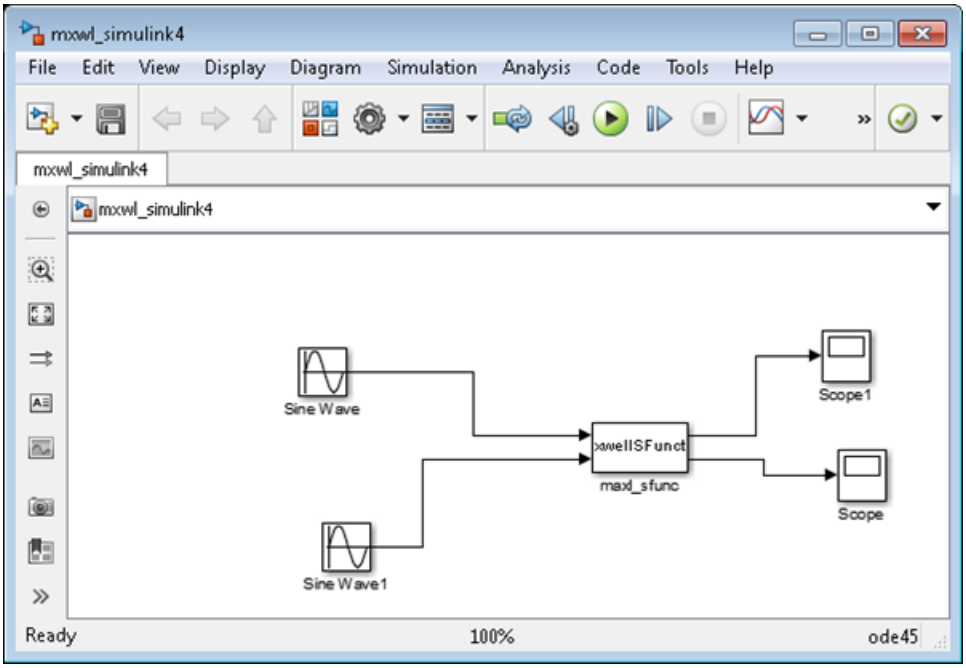
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If changes are needed, click **Edit Circuit** to reopen the **Maxwell Link Assignment** dialog box.

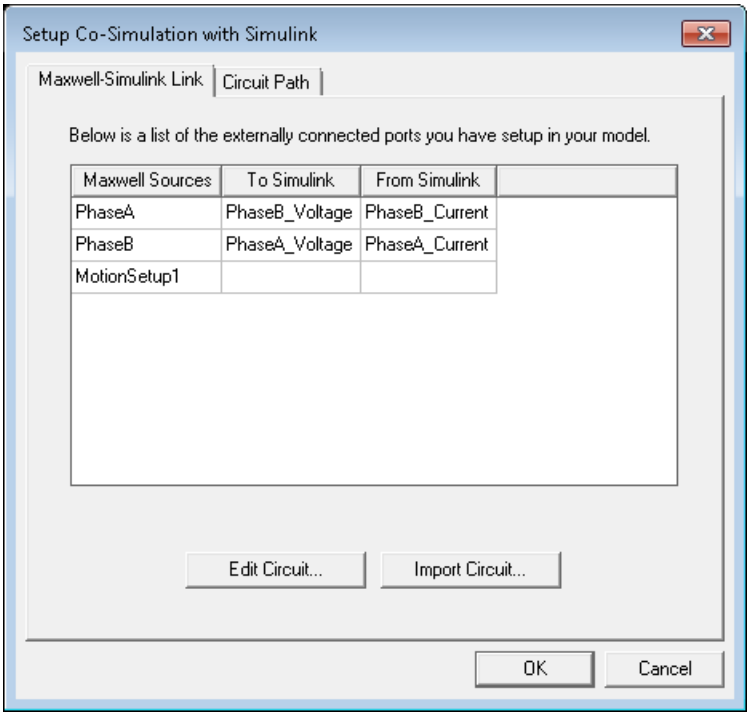
4. Modify (or create) the circuit in the Simulink circuit editor window as needed, and save the Simulink circuit project.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. In the Maxwell **Setup Cosimulation with Simulink** dialog box, click the **Import Circuit** button to import the Simulink circuit project into the Maxwell project. The linked connections appear in the **Edit External Circuit** dialog box. The **Circuit Path** tab shows the path to the location from which the Simulink circuit file was imported.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:** You can also modify an existing Simulink circuit by opening the circuit file directly in Simulink, and then importing the modified circuit file back into the Maxwell design.

6. When you start the Maxwell design analysis, the cosimulations are started between Maxwell and Simulink.

> **Note:**
>
> - Making changes to the Maxwell winding and motion setups invalidate the external circuit.
> - Making changes to the external Simulink circuit setup invalidates the solutions.

Setting Up Motion for Transient Projects

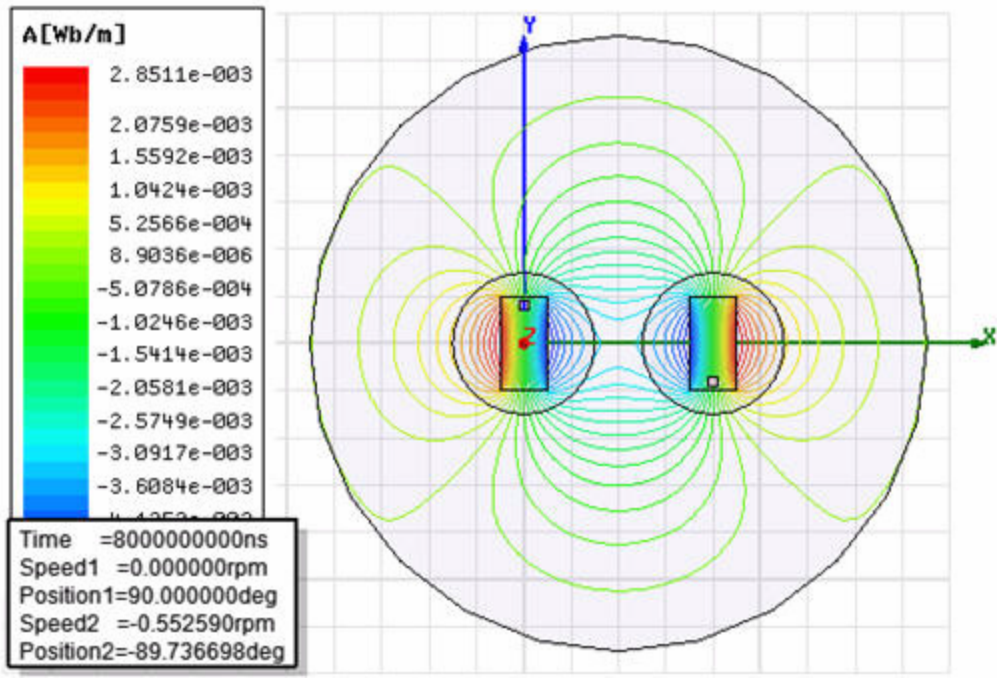
For projects where you selected transient as the solution type, you can define how the model components move in relation to one another. This is accomplished by assigning a **MotionSetup** to a **Band** object. Two or more motion bands may be assigned, each with its own independent motion characteristics. Multiple motion bands are used primarily for certain types of motors

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where multiple rotors may rotate about a common central axis; however, other applications are possible as shown in the following examples.

Example - Two Bands of Motion

The project shown below comprises two rectangular magnets, each surrounded by a band object with a MotionSetup assigned to it. When one magnet moves, the other magnet realigns itself. Also note that the solution context display shows the position and speed for both motion setups.

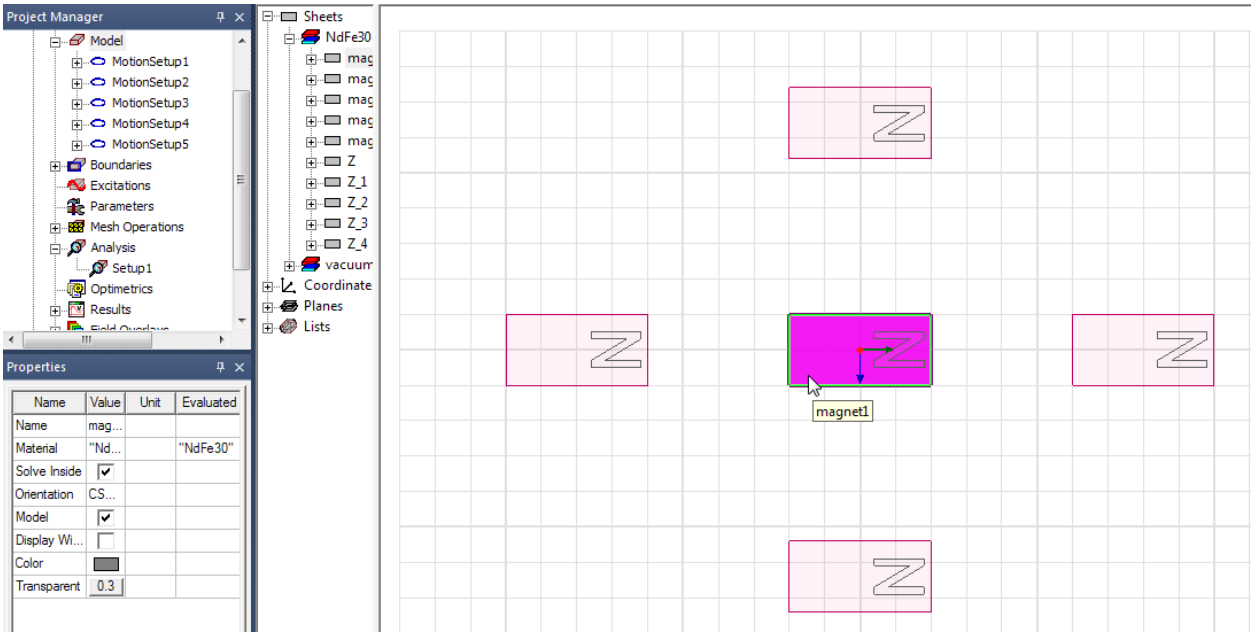


Note: The two moving bands must not touch each other. There must be some stationary region between the two bands, even if it is filled only with vacuum.

Example - Multiple Bands of Motion

The magnet project shown below comprises five bar magnets. When the center magnet (MotionSetup1) moves, the other magnets also move.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:** The moving bands may not touch each other. There must be some stationary region between the bands, even if it is filled only with vacuum.

Assigning a Band of Motion

You can assign a band of motion in both Maxwell 2D and Maxwell 3D projects. For both Maxwell 2D and 3D projects, two or more motion bands may be assigned.

> **Note:** Overlapping bands are not supported so they cannot be located inside of each other.

For both rotational and translational problems, the band object must always enclose all of the moving objects. For rotational problems, this means the band object should be a solid cylinder or a solid wedge – not a hollow shell or segment of a shell. If the band object encloses all moving aspects, it is easier to identify the moving objects and reduces errors related to misidentifying them.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: An inner region (or wrapper object) enclosed by the band can normally be used as a “fake” moving object for two reasons:

- The first is to avoid creating holes inside the band which would result in the band having multiple connected domains.
- The second is due to the torque/force computation being carried out by one layer of elements inside the band region that directly touches moving objects. If an inner region is not included, the torque/force computation will be very sensitive to mesh noise - thus leading to an inaccurate solution.

Note: For 2D and 3D translational motion, here are some additional recommendations:

- An all-encompassing inner region (non-magnetic, non-conductive wrapper such as air or vacuum) must be used around all moving objects in a band.
- The inner region wrapper object, like the Band itself, should be a simplified shape, such as a rectangle or polygon in 2D, or a polyhedral or segmented sweep section in 3D. Any simplifications to the inner region and band object will result in better re-meshing for each time-step and faster overall simulation times.
- In order to give more accurate forces, if there is sufficient clearance between moving and stationary objects, you can provide some padding between the inner region and the moving objects and the band. At a minimum, the inner region should be the same size as the outside of the enclosed objects (like shrink wrap) with no gap between the wrapper and objects inside.
- Ansys highly recommends that the band not touch any outer stationary objects, with the only exception being for symmetry boundaries on the outer region. It is suggested to locate the band halfway in the air-gap between fixed and moving objects or equally spaced with the inner region object if there is sufficient clearance at the gap.
- In order to get stable force results, length-based mesh operations (approximately equal to the distance between the band and the inner region) should be applied using the Length-Based-Inside mesh refinement assigned to the Band object so that the mesh length is refined in the band region that is re-meshed for each time-step.

To assign the band of motion:

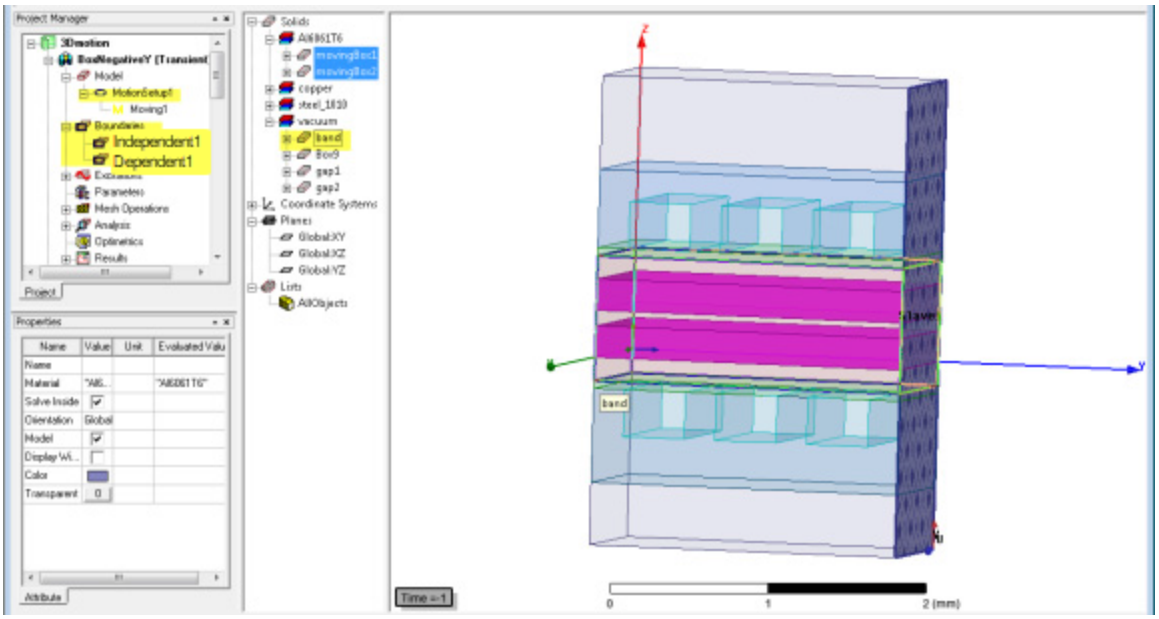
1. Select the band object.
2. Click **Maxwell** > **Model** > **Motion Setup** > **Assign Band**.

   The **Motion Setup** window appears.
3. On the **Type** tab: select either **Translation** or **Rotation** as the **Motion Type**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For **Rotation** about a fixed point you may select the **Non-Cylindrical** check box.
- For **Translation**, you may select the **Periodic** check box.

For **Periodic** motion, you must define independent and dependent boundary conditions. Both band and the object(s) inside it move, and both should touch the independent and dependent boundaries as shown in the following example.



4. Set the motion direction:
   a. For **Translation** (including **Periodic**), select the desired X, Y, or Z (for 3D) axis of motion from the **Moving Vector** drop-down menu; then select either **Positive** or **Negative** direction.
   b. For **Rotation**, from the **Moving Axis** drop-down menu, select an axis of rotation. Use the radio buttons to select either **Positive** or **Negative** rotation.

   > **Note:**
   > - For 2D XY rotation, only **Positive** rotation is allowed.
   > - If the expected moving vector does not exist, click **Modeler** > **Coordinate System** > **Create** > **Relative CS** to create a new coordinate system.

5. On the **Data** tab: type a value in the **Initial Position** box, and select the units from the drop-down menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. To set the motion limits, do the following in the **Translate Limit** or **Rotate Limit** section:

   Note: For Periodic motion, you do not need to define motion limits because the solver does not consider the limits for periodic motion; thus the **Translate Limit** section is not present.

   a. For rotational motion, click the **Rotate Limit** check box.
   b. For translational (non-**Periodic**) or rotational motion, type a value in the **Negative** box, and select the units from the drop-down menu.
   c. For translational (non-**Periodic**) or rotational motion, type a value in the **Positive** box, and select the units from the drop-down menu.

7. On the **Mechanical** tab, do one of the following to specify the object velocity (dynamic or constant):

   Note: When you select **Consider Mechanical Transient**, you are telling the solver to use the calculated force and user-specified mass, damping, and other parameters to dynamically determine how the object moves, rather than giving it a constant velocity.

   Note:
   - For Maxwell 2D/3D designs for motor applications, the computed output mechanical power at given constant speed may not reach the desired rated power. Therefore, it is useful to apply constant power as the mechanical load. In Maxwell 2D/3D designs created by RMxprt, the mechanical transient is set up according to the rated mechanical output power, and equivalent damping is added to accelerate the process approaching to the rated output power. This setup is visible when **Consider Mechanical Transient** is checked.
   - The **Consider Mechanical Transient** setup is only for the sake of getting the operating point with given output power. it is not related to any mechanical load type you may set in RMxprt.
   - 2D/3D designs generated from RMxprt include the setup data, but **Consider Mechanical Transient** is unchecked initially. To ensure that the setup data is preserved, make sure that you check **Consider Mechanical Transient** before saving the project.

   - To dynamically change how the object moves, click the **Consider Mechanical Transient** check box, complete the following fields, and select any corresponding units:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Initial Velocity** (translational) or **Initial Angular Velocity** (rotational)
- **Mass** (translational) or **Moment of Inertia** (rotational)

Use the **Calculate** button to have Maxwell calculate the Mass or Moment of Inertia with the following restrictions:

- For 2D problems, the moving bodies are assumed to be of model-unit thinness.
- Rotating bodies must be symmetric about the axis of rotation. Partial models (e.g., half a motor with implied symmetry) are not supported.
- **Damping** (rotational motion) – A value for damping for rotational motion (for electric machines) can be calculated by the windage and friction loss in Watts divided by the speed in rad/sec (at which those losses are defined); the resulting units are in W / $(rad/sec)^2$ or N*m * sec/rad.
- **Damping** (linear motion) – A value for damping for linear motion can be calculated by the windage and friction loss in Watts divided by the speed in m/sec (at which those losses are defined); the resulting units are in W / $(m/sec)^2$ or N*m * sec/m.
- **Load Force** (translational) or **Load Torque** (rotational)

  Note:
  - For translational motion, the load force is positive if it is applied in the same direction as the moving vector and negative if applied in the opposite direction.
  - For rotational motion, the load torque sign is determined based on the moving vector, using the right-hand rule. Point your thumb in the direction of the moving vector. If the load torque is applied in the same direction as your fingers, then the load torque is positive. If it is applied in the opposite direction, then the load torque is negative.

- To set a constant velocity (and disable the mechanical transient), clear the **Consider Mechanical Transient** check box, type a value in the **Velocity** or **Angular Velocity** box, and select the units from the drop-down menu.

8. For translational motion, a **Post-Processing** tab is available to set the coordinate system for post processing calculation.
9. Click **OK**.

- A **MotionSetup*x*** is created and added to the **Model** folder in the **Project Manager** window.
- For Maxwell 3D projects only, if cylindrical **Rotation** was specified and a **CylindricalGap** mesh operation does not already exist, a **CylindricalGap** mesh operation is automatically added to the **Mesh** folder. Only one **CylindricalGap** mesh operation is permitted.

Modifying Motion Setup Properties

To modify existing motion setup properties:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the Project Manager **Model** folder, right-click on **MotionSetup*x*** and select **Properties** from the context menu to open the **Motion Setup** dialog box.
   - Alternatively, you can double-click the **MotionSetup*x*** icon.
2. Edit the desired properties in the **Motion Setup** dialog box. Refer to *Assigning a Band of Motion* for detailed information on the various **Motion Setup** dialog box settings.

Note:

For Maxwell 3D projects only:

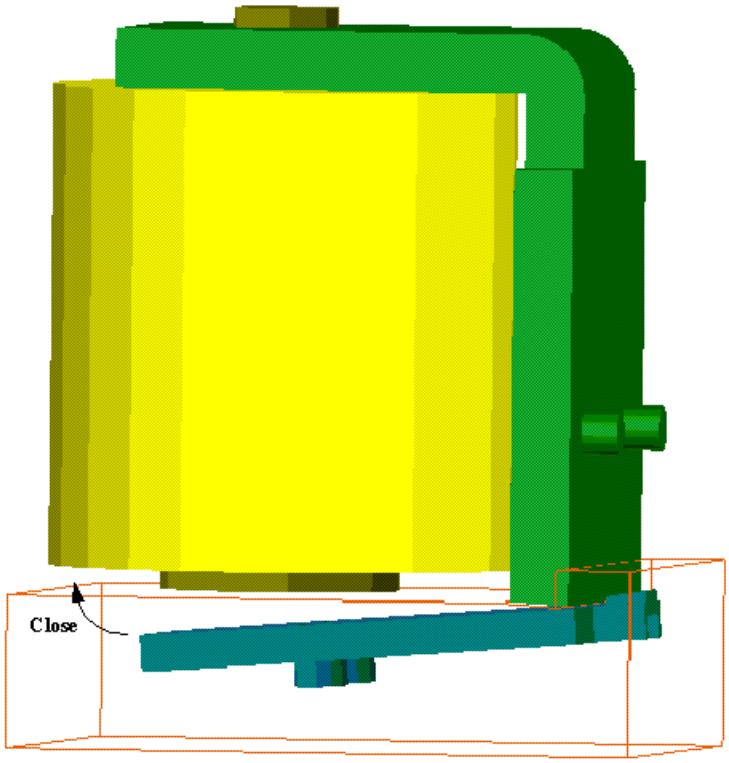
- If you change the motion type to cylindrical **Rotation** and a **CylindricalGap** mesh operation does not already exist, a **CylindricalGap** mesh operation is automatically added to the **Mesh** folder. Only one **CylindricalGap** mesh operation is permitted.
- If you change a cylindrical **Rotation** motion type to another motion type, the **CylindricalGap** mesh operation remains in the **Mesh Operation** folder.
- Alternatively, you can edit many of the motion setup properties in the **Properties** window.

Non-Cylindrical Rotational Motion

With cylindrical rotation, the moving objects are free to rotate completely, and the mesh remains fixed as the moving objects rotate. With non-cylindrical motion, the geometry shape of the band region changes as the moving objects rotate, and the mesh is recreated.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- A motor is an example of cylindrical motion.
- A relay is an example of non-cylindrical motion.



Reassigning a Band of Motion

When a **MotionSetup** has been created, the object that the setup is assigned to is changed using the **Reassign** command, allowing you to identify the motion setup parameters with a new object without losing the motion setup parameters themselves. To reassign a **MotionSetup**:

1. Select the object that the **MotionSetup** will be reassigned to in the modeler window or in the history tree.
2. Right-click on **MotionSetup*x*** in the Project Manager window and select **Reassign**.

> **Note:** An existing **CylindricalGap** mesh operation's assignment will be synced up to a rotational cylindrical motion setup's assignment when you reassign the band object of the motion setup.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adding and Deleting Objects from a Band of Motion

To add objects to, or delete objects from a motion setup without changing the parameters of the **MotionSetup** do the following:

1. Select the object to be added or deleted from the **MotionSetup** in the modeler window or in the history tree.
2. Right-click **Model** > **MotionSetup*x*** > **Moving*x*** in the Project Manager window and choose **Add Selected Objects** or **Remove Selected Objects** from the shortcut menu.
3. To verify the addition or deletion of the object from the MotionSetup definition, use **Select Assignment** from the shortcut menu to view the objects assigned to the band.

Showing and Hiding Motion Objects

1. Click **Maxwell** > **Model** > **Motion Setup** > **Visualization**.
2. From the **Motion Setup Visualization Options** window that appears, select **All**, **Band**, or **Moving** from the **Select** drop-down menu.
3. Click **Deselect All** to hide all motion objects.
4. Click **Close** to close the window.

Setting the Visibility for Motion Objects

To show or hide a parameter:

1. Click **View** > **Active View Visibility**.
2. From the **Active View Visibility** dialog box that appears, click the **Motion** tab.
3. Select or clear the **Visibility** check box for each motion object listed.
4. Click **Done**.

Modeling Maxwell Motion Eccentricity

You can model motion eccentricity for Maxwell 2D and 3D transient designs that have cylindrical rotational motion set up. Ansys Electronics Desktop provides a **Maxwell Eccentricity Wizard** with settings for

- rotating part eccentricity
- rotation axis eccentricity

For rotating part eccentricity modeling, the eccentricity is only applied to rotating objects, not to a motion axis. More specifically, these rotating objects are all objects enclosed by the band, but not the band itself.

For rotation axis modeling, the eccentricity is only applied to the motion axis, which is equivalent to moving the band itself, but not any rotating objects inside the band.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using these two basic settings, you can model the following physics.

- **Static eccentricity**: To model static eccentricity, you need to set the rotating part eccentricity and rotation axis eccentricity to the same values.
- **Dynamic eccentricity**: To model dynamic eccentricity, set the rotating part eccentricity values only.
- **Mixed eccentricity**: This is the combination of static and dynamic eccentricities. To model mixed eccentricity, set both rotating part eccentricity and rotation axis eccentricity, but with different values.

Using the Maxwell Eccentricity Wizard

> **Note:**
>
> The **ACT Extensions** window and the design wizards it contains (*5G Wizard, HFSS Antenna Design Toolkit, HFSS-EMA3D Link,* and *Maxwell Eccentricities*) are only available for the Windows version of the Ansys Electronics Desktop software. These items are not available when the software is installed on a Linux platform.

You can use the Maxwell Eccentricity wizard to model both 2D and 3D transient designs that have cylindrical rotational motion setup.

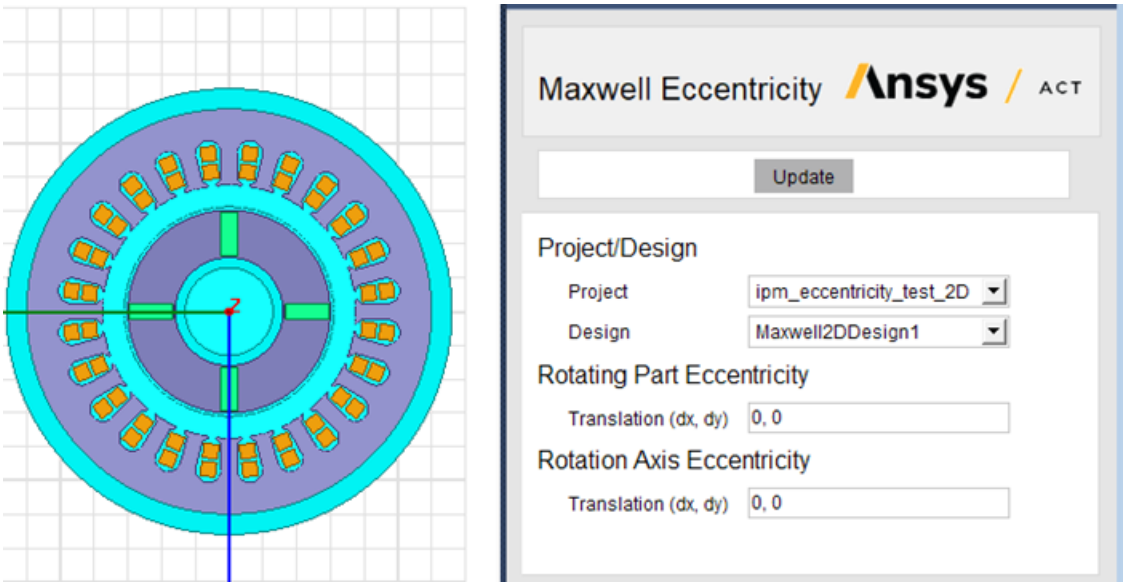
> **Note:**
>
> If you want to generate a 3D rotational eccentricity motor design from an RMxprt-generated Maxwell 3D design, you must enter **Eccentricity 1** in the RMxprt **Design Setting** window's **User Defined Data tab** for the RMxprt design *before* generating the Maxwell 3D design.

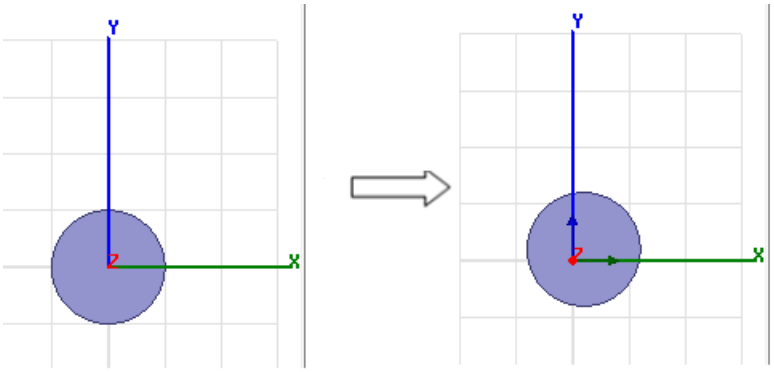
1. Select **View** > **ACT Extensions** to open the ACT Extensions window.
2. Click the **Wizards** button to open the Wizards page, then click **Maxwell Eccentricity** to load the Maxwell Eccentricity setup page.
3. Select the project and design you want to use.

   > **Note:**
   >
   > - The **Update** button synchronizes the project and design selection lists with those currently loaded in the Desktop.
   > - You can click **Exit Wizard** to close the eccentricity wizard and return to the wizard start page without executing any changes.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

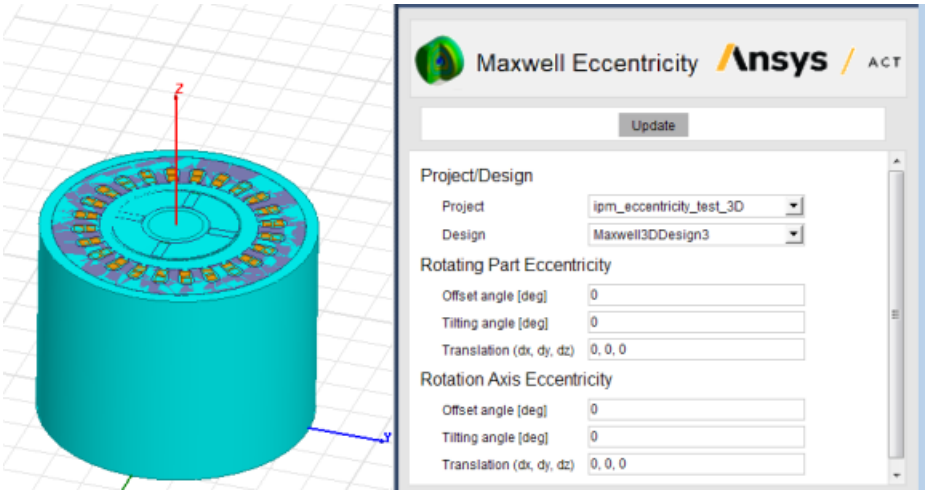
4. Depending on whether your design is 2D or 3D, do one of the following:
    - For a 2D Maxwell project, enter the desired dx and dy translation values to set up the Rotating Part Eccentricity and Rotation Axis Eccentricity. The unit of measure for translation values is the model's unit, which is set using **Modeler** > **Units**.



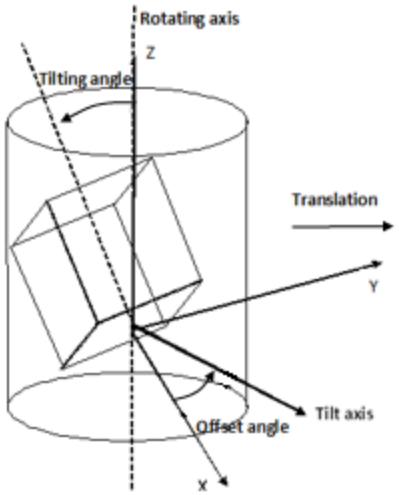


    - For a 3D Maxwell project, enter the desired dx, dy , and dz translation values, the offset angle, and the tilting angle for the rotating part eccentricity axis and rotation axis. The unit of measure for translation values is the model's unit, which is set using **Modeler** > **Units**.

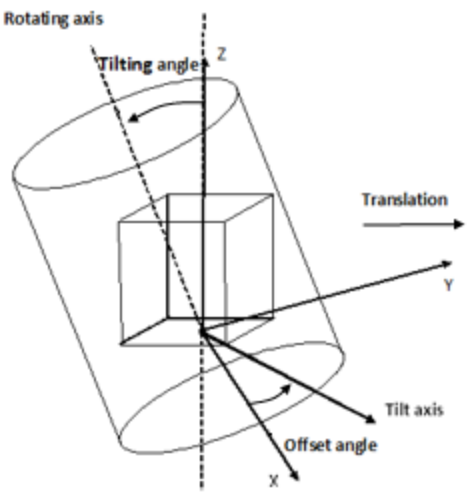
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Rotation Part Eccentricity



Rotation Axis Eccentricity



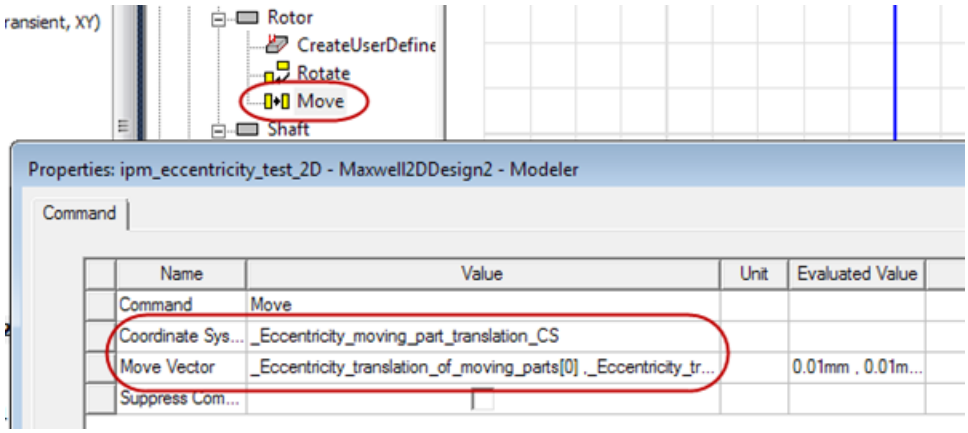
5. Click the **Finish** button at the bottom of the page. The page closes and the wizard adds a new design in the same project, in which the Maxwell geometry has been updated in accordance with the eccentricity setup values you entered, as explained in the sections that follow.

For 2D Designs

- Design variables are created for each of the data entered in the wizard.
  - For rotating part eccentricity: _Eccentricity_translation_of_moving_parts
  - For rotation axis eccentricity:_Eccentricity_translation_of_rotating_axis

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The moving parts in the design motion setup are shifted according to the data defined for **Rotating Part Eccentricity** in the Wizard. New coordinate systems are generated as the object coordinate systems for moving parts, according to the same operations.
- The band is translated according to the data defined for **Rotation Axis Eccentricity** in the wizard.
- The **Moving Vector** entry in the design's motion setup is replaced with the new direction (represented by an axis of a new coordinate system) generated according to the data defined for **Rotation Axis Eccentricity** in the Wizard.



- New design coordinate systems are created.

  _Eccentricity_rotating_axis_translation_CS

  _Eccentricity_moving_part_translation_CS

  _Eccentricity_moving_part_CS_<*part name*> – one for each moving part

Note:

- All the above operations of translation, as well as the creation of new coordinate systems, are dependent on the eccentricity design variables.
- You can modify these design variables to adjust the eccentricity.
- The newly added coordinate systems should not be changed manually by the user.
- Using the wizard to modify a design that has already been set up for eccentricity is not recommended.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For 3D Designs

- Design variables are created for each of the data entered in the wizard.

  For rotating part eccentricity:

  - _Eccentricity_offset_angle_of_moving_parts
  - _Eccentricity_translation_of_moving_parts
  - _Eccentricity_tilting_angle_of_moving_parts

  For rotation axis eccentricity:

  - _Eccentricity_offset_angle_of_motion_axis
  - _Eccentricity_translation_of_motion_axis
  - _Eccentricity_tilting_angle_of_motion_axis
- The moving parts in the design motion setup are rotated and shifted according to the data defined for **Rotating Part Eccentricity** in the Wizard. New coordinate systems are generated as the object coordinate systems for moving parts, according to the same operations.
- The band is rotated and translated according to the data defined for **Rotation Axis Eccentricity** in the wizard.
- The **Moving Vector** entry in the design's motion setup is replaced with the new direction (represented by an axis of a new coordinate system) generated according to the data defined for **Rotation Axis Eccentricity** in the Wizard.
- New design coordinate systems are created.

  For the motion moving vector:

  - _Eccentricity_motion_axis_CS
  - _Eccentricity_motion_axis_tilting_axis_CS

  For the moving parts:

  - _Eccentricity_moving_part_tilting_axis_CS
  - _Eccentricity_moving_part_translation_CS
  - _Eccentricity_moving_part_CS_*<part name>* – one for each moving part

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- All the above operations of rotation and translation, as well as the creation of new coordinate systems, are dependent on the eccentricity design variables.
- You can modify these design variables to adjust the eccentricity.
- The newly added coordinate systems should not be changed manually by the user.
- Using the wizard to modify a design that has already been set up for eccentricity is not recommended.

Note:

When modeling motion eccentricity, the position and direction of the moving objects and/or band are adjusted based on the eccentricity setup. In such cases, all the adjustments are referenced to the original position and direction of the geometry of each object, As a result, after you finish the eccentricity setup and click the **Finish** button, all the geometries in the design created by the wizard are updated to new positions based on the eccentricity setup. If you want to edit geometry, you should do so in the original design. After editing the geometry, you can then redo the eccentricity setup to create a new design.

Transient A-Phi Formulation Boundaries and Excitations

The Transient A-Phi solver is an alternative to the default TAU transient solver. The A-Phi solver allows multi-terminal conductors and it supports sources of various types on a single conduction path; the J and E fields are calculated directly because it is a first-order approximation. *A* is the Magnetic Vector Potential and Φ (Phi) is the Electric Scalar Potential.

To use this solver, define at least one of the following as a source of magnetic fields:

- A voltage or current.
- Stranded or solid windings, with a voltage or current supply or connected to an external circuit winding.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** For stranded conductors, it is assumed that the current density is uniform on the cross-section of the respective conductors. Thus, the stranded option for current excitations should mostly be used only in cases where the cross-section of conductors is constant, consistently with the assumption that the respective object is a coil built with strands of wire. For this reason, conductors (coils) could be created by either sweeping around an axis using zero segments or sweeping along a path having only true surface segments in order to create a smooth surface coil having a completely uniform cross-sectional area over its entire length.

- A permanent magnet.

You may need to set at least one outer boundary to the following:

- The default boundary conditions.
- An odd symmetry (flux tangential) boundary.
- An even symmetry (flux normal) boundary.

Transient A-Phi Formulation Boundaries

The transient A-Phi field solver allows you to define the following types of boundary conditions.

> **Note:** The setup for Insulating, Symmetry, Independent, Dependent, and Resistive Sheet boundaries is the same for both Transient and Transient A-Phi Formulation solvers.

Boundary Type	H-Field Behavior	Used to model...
Default Boundary Conditions (Natural and Neumann)	Field behaves as follows: • Natural boundaries –The tangential component of **H** and the normal component of **J** are continuous across the boundary. • Neumann boundaries – **H** is normal to the boundary; its tangential components are zero. **J** field is tangential to the boundary. • By default, the Boundary/Source Manager automatically applies the Neumann boundary to all faces of the	Ordinary field behavior. Initially, object interfaces are natural boundaries; outer boundaries, and excluded objects are Neumann boundaries.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	problem region – that is, surfaces exposed to the non-meshed space.	
Flux Tangential	**H** is tangential to the boundary and flux cannot cross it.	Problems with flux tangential at outer boundary.
Insulating	Same as Natural boundary, except that current cannot cross the boundary.	Thin, perfectly insulating sheets between touching conductors.
Symmetry	Field behaves as follows: • Odd Symmetry (Flux Tangential) – **H** is tangential to the boundary; its normal components are zero. The potential value at the boundary is zero. • Even Symmetry (Flux Normal) – **H** is normal to the boundary; its tangential components are zero.	Planes of geometric and magnetic symmetry.
Matching (Independent and Dependent)	The **H**-field on the dependent boundary is forced to match the magnitude and direction (or the negative of the direction) of the **H**-field on the independent boundary.	Planes of symmetry in periodic structures where **H** is oblique to the boundary.
Resistive Sheet	Same as Natural boundary.	Voltage drop and loss on the contact surface between two different objects within a conduction path.

Default Boundary Conditions for a Transient A-Phi Solver

These boundary conditions are automatically defined for a transient model:

- Natural boundaries are assigned to the surfaces between objects.
- Neumann boundaries are assigned to the outer surface of the problem region. **J** field is tangential to the boundary, and **H** field is normal to the boundary.

To leave a surface set to its default boundary condition, you do not need to set any boundary conditions. Deleted boundary conditions and excitations automatically reset to the default boundary conditions.

Assigning a Flux Tangential Boundary for a Transient A-Phi Solver

> **Note:** This setting is also applicable to the 3D AC Magnetic A-Phi Solver.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This type of boundary is applied when the **H** field is tangential to the boundary.

To define a zero tangential **H** field boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically a face).
2. Click **Maxwell** > **Boundaries** > **Assign** > **Flux Tangential**.
3. The **Flux Tangential** window appears.
4. Enter a name for the boundary in the **Name** box, or accept the default.
5. Click **OK** to assign the boundary to the selected object.

Transient A-Phi Formulation Excitations

Transient A-Phi formulation supports the following excitations:

Excitation	Type of Excitation
Coil Terminal	Used to define windings.
Winding With Current	Current for a stranded or solid conductor.
Winding With Voltage	Voltage for a stranded or solid conductor.
Winding With External Circuit Connection	External circuit connection for a stranded or solid conductor.
Voltage	Voltage for a solid conductor.
Current	Current for a stranded or solid conductor.
Permanent Magnet Field	Permanent magnets serve as sources of magnetic fields.

Note: A winding is a flexible excitation. Six different combinations can be used: Current, Voltage, and External Circuit — each with a solid or stranded conductor. The Coil Terminal type of excitation is needed to set up a winding.

Note: Winding excitation is assigned as in the transient solver. However, coil terminals must be ordered according to the physical potential distribution as explained in Boundaries and Excitations > Winding Excitations for A-Phi Formulation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Electric Transient Solver Boundaries and Excitations

The following boundaries can be specified for this solver:

- Symmetry
- (Matching) Independent/Dependent
- Insulating

Specify at least one of the following excitations as a source of electric fields:

- The total charge on object.
- The volume charge density inside an object.
- The current.
- Define the electric potential on each surface using a voltage excitation.
- Floating models conductors at unknown potentials and specifies the total charge on the conductor.
- Sink can be applied only to 2D surfaces which are grounded with reference potential value. It is the same as applying zero voltage excitation on those surfaces.

Electric Transient Excitations

The following excitations are available for electric transient problems:

- **Charge** – This excitation can be applied to 2D surfaces or 3D volume bodies, if a body has net charge q. This type of excitation defines the total charge on a surface or object.
- **Current** – This excitation can be applied only to 2D surfaces, where the total current flowing in/out of that surface is specified.
- **Floating** – This excitation can be applied to 2D surfaces or 3D volume bodies; then these 2D or 3D objects are with equipotential. This type of excitation models conductors at unknown potentials and specifies the total charge on the conductor.
- **Sink** – This excitation can be applied only to 2D surfaces that are grounded with reference potential value. It is the same as applying zero voltage excitation on those surfaces.
- **Voltage** – This excitation specifies the value of the electric scalar potential value on 2D surfaces or 3D volume bodies.
- **Volume Charge Density** – This excitation can be applied only to 3D volume bodies. If the body has net charge q, the charge density is q/VOL, where VOL stand for the volume of that body.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning a Voltage Excitation for an Electric Transient Solver

To define a voltage excitation:

1. Select the section of the geometry on which you want to apply the excitation.
2. Click **Maxwell** > **Excitations** > **Assign** > **Voltage**.

   The **Voltage Excitation** dialog box appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the voltage in the **Value** box, and select the units. Then, select a **Coordinate System**.
5. Optionally, click **Use Defaults** to revert to the default values.
6. Click **OK** to assign the excitation to the selected object.

Assigning a Charge Excitation for an Electric Transient Solver

To define a charge excitation:

1. Select the section of the geometry on which you want to apply the excitation.
2. Click **Maxwell** > **Excitations** > **Assign** > **Charge**.

   The **Charge Excitation** dialog box appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the charge in the **Value** box.
5. Optionally, click **Use Defaults** to revert to the default values.
6. Click **OK** to assign the excitation to the selected object.

Assigning a Floating Excitation for an Electric Transient Solver

To define a floating excitation:

1. Select the section of the geometry on which you want to apply the excitation.
2. Click **Maxwell** > **Excitations** > **Assign** > **Floating**.

   The **Floating Excitation** dialog box appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the floating excitation in the **Value** box.
5. Optionally, click **Use Defaults** to revert to the default values.
6. Click **OK** to assign the excitation to the selected object.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning a Volume Charge Density Excitation for an Electric Transient Solver

To define a volume charge density excitation:

1. Select the section of the geometry on which you want to apply the excitation.
2. Click **Maxwell** > **Excitations** > **Assign** > **Volume Charge Density**.

   The **Volume Charge Density Excitation** dialog box appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the volume charge density in the **Value** box. Then, select a **Coordinate System**.
5. Optionally, click **Use Defaults** to revert to the default values.
6. Click **OK** to assign the excitation to the selected object.

Assigning a Current Excitation for an Electric Transient Solver

To define a current excitation:

1. Select the section of the geometry on which you want to apply the excitation.
2. Click **Maxwell 3D** > **Excitations** > **Assign** > **Current**.

   The **Current Excitation** dialog box appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the current excitation in the **Value** box, and select the units.
5. To swap the direction of the current, click **Swap Direction**.
6. Optionally, click **Use Defaults** to revert to the default values.
7. Click **OK** to assign the excitation to the selected object.

Assigning a Sink Excitation for an Electric Transient Solver

To define a current excitation:

1. Select the section of the geometry on which you want to apply the excitation.
2. Click **Maxwell** > **Excitations** > **Assign** > **Sink**.

   The **Sink Excitation** dialog box appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. Click **OK** to assign the excitation to the selected object.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11 - Assigning Boundaries and Excitations for 2D Designs

For every design, you need to assign boundaries and excitations. You can use the **Maxwell 2D** menu or the project tree to do the following:

- Define boundary conditions that control how the electric or magnetic field behaves planes of symmetry, periodicity, or edges of the problem region.
- Define solution type specific excitations of voltage, charge, coil, and current.
- For magnetic transient designs, if needed, set up a winding and an external circuit connection.

Each field solver requires you to specify excitations of electric or magnetic fields and references for computing these fields.

- Magnetostatic Boundaries and Excitations
- Electrostatic Boundaries and Excitations
- AC Conduction Boundaries and Excitation
- DC Conduction Boundaries and Excitations
- AC Magnetic(Eddy Current) Boundaries and Excitations
- Transient Boundaries and Excitations

You must specify at least one of the boundary conditions or excitations listed in these sections so that the simulator can compute accurate values for fields and parameters.

Selecting Objects and Faces in the 2D Geometry

Before creating a boundary or excitation, you must specify its location by selecting the object or surface on which to assign the boundary condition or source excitation.

Defining Boundary Conditions in 2D

To assign a boundary condition:

1. Select the section of the geometry on which you want to apply the boundary condition.
2. Click **Maxwell 2D** > **Boundaries** > **Assign**, or right-click **Boundaries** in the Project Manager tree and click **Assign**.
3. Select the desired boundary type from the sub-menu. The specific boundary types you can define depend on the solver you have chosen for your project.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Magnetostatic Boundary Conditions
- Electrostatic Boundary Conditions
- AC Conduction (Electric) Boundary Conditions
- DC Conduction (Electric) Boundary Conditions
- AC Magnetic Boundary Conditions
- 2D Transient Boundary Conditions

4. Enter the name for the boundary or accept the default.
5. Depending on the boundary type, you may need to define additional parameters.
6. Click **OK** to create the new boundary.

The new boundary is added to the boundary list in the project tree.

Defining 2D Excitations

The specific excitations you can define depend on the solver you have chosen for your design.

- Magnetostatic Excitations
- Electrostatic (Electric) Excitations
- AC Conduction (Electric) Excitations
- DC Conduction (Electric) Excitations
- AC Magnetic Excitations
- 2D Transient Excitations

To assign an excitation:

1. Select the section of the geometry on which you want to apply the excitation.
2. Click **Maxwell 2D** > **Excitations** > **Assign**, or right-click **Excitations** in the project tree and click **Assign**.

   A submenu appears with choices that depend upon the solver type.
3. Select one of the following excitation types from the submenu:
   - Voltage
   - Charge
   - Floating
   - Charge Density
   - Current Density
   - Current
   - Coil
   - End Connection
   - Permanent Magnet Field

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Enter the name for the excitation type or accept the default.
5. Enter the desired parameter values, units, and other settings for the excitation in the dialog box. You can also enter a mathematical function to determine values for voltage, current density, charge density, and other parameters. The Permanent Magnet Field excitation also requires you set up a link to a source project.

   > **Note:** When entering current, or coil terminals, the arrow associated with current in the selected object shows the direction as if a positive value is entered for the current. If a negative value is entered the actual current flow direction is opposite to what the arrow shows. Click **Swap Direction** if you want to reverse the direction of the arrow.

6. Click **OK** to create the new excitation.

The new excitation is added to the excitation list in the project tree.

Viewing 2D Boundaries and Excitations

To view a list of all boundaries or excitations set for the design:

1. In the project tree, right-click **Boundaries** or **Excitations**.
   A shortcut menu appears.
2. Select **List** from the shortcut menu.

   The **Design List** dialog box appears, with the tab page visible for the item you selected. This dialog box contains tabs for **Model**, **Boundaries**, **Excitations**, **Parameters**, **Mesh Operations**, and **Analysis Setup**.
3. To view the properties for an item in the list, select the item, and click **Properties**.
4. To delete an item from the list, select the item, and click **Delete**.
5. When you are finished in the **Design List** dialog box, click **Done** to close it.

Setting the Visibility for 2D Boundaries and Excitations

To show or hide a boundary:

1. Click **View** > **Active View Visibility**.

   The **Active View Visibility** dialog box appears.
2. Click the **Boundaries** tab.
3. Select or clear the **Visibility** check box for each boundary listed.
4. Click **Done**.

To show or hide a excitation:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **View** > **Active View Visibility**.

   The **Active View Visibility** dialog box appears.

2. Click the **Excitations** tab.
3. Select or clear the **Visibility** check box for each excitation listed.
4. Click **Done**.

Functional Boundaries and Excitations in 2D

Functional boundaries and excitations can be used to do the following:

- Define the value of a boundary or excitation quantity (such as the voltage, magnetic field, or current density) using a mathematical relationship — such as one relating its value to that of another quantity.
- Define the value of a boundary or excitation as a function of position.
- Current Density Excitation (Magnetostatic or AC Magnetic solver type)
- Voltage Excitation (Electric solver types)
- Volume Charge Density Excitation (Electrostatic or Electrostatic + DC solver types)
- If the parametric analysis capability is installed, identify which boundary or excitation quantities are to be varied during a parametric sweep. These variables are always set to constant values in the parametric analysis.

Setting Eddy Effects and Displacement Current in 2D

(2D AC Magnetic and/or Transient Solvers)

For both AC magnetic and transient solvers in 2D, you may need to specify the following additional settings:

- The behavior of eddy currents and the AC magnetic field in conductors. When you activate the **Eddy Effect** setting, the solver computes the induced eddy currents.
- For 2D Transient, source settings override the eddy effect settings. Objects assigned as “solid” sources will have eddy effects turned on, while objects assigned as “stranded” sources will have eddy effects turned off (regardless of the eddy effect settings). The eddy effect settings will be respected for non-sources.

Typically, background objects are excluded from eddy current settings.

To set eddy effects:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the section of the geometry on which you want to apply the excitation.
2. Click **Maxwell 2D** > **Excitations** > **Set Eddy Effects**.
   The **Set Eddy Effect** dialog box appears showing only objects which are valid for eddy effect calculation, specifically objects with conductivity greater than 1.
3. For each object in the list, select or clear the check boxes for **Eddy Effect**. When selecting multiple entries, the changes are synchronized for all selected rows.

   > **Note:** Multi-select is supported by holding the **Ctrl** or **Shift** key while clicking.

4. Click the **Select By Name** button to enter the name of a specific object for selection. Multiple objects may be selected using the * and ? wildcards.
5. Optionally, click the **Use suggested values** button to set the **Eddy Effects** for all elements recommended by Maxwell 2D.
6. Click **OK**.

The **Deselect All** button will deselect all selected objects in the list.

Modifying 2D Boundary Conditions and Excitations

Save your project before modifying boundary conditions and excitations. You can modify boundary conditions and excitations after a solution has been generated, but you may lose some solution data.

Duplicating 2D Boundaries and Excitations

To duplicate a boundary or excitation when its geometry is pasted or duplicated:

1. Click **Tools** > **Options** > **Maxwell 2D Options**.

   The **Maxwell 2D Options** dialog box appears.

2. Click the **General Options** tab.
3. Select **Duplicate boundaries with geometry**.
4. Click **OK**.

All boundaries and excitations are duplicated with their associated geometries until you clear this option.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tip:

Use this option to copy and paste boundaries. For example, do the following to use the same boundary on multiple objects:

1. Select the face to which you want to assign the boundary.
2. Click **Modeler** > **Surface** > **Create Object From Face** to create a new object from the existing face.
3. Assign the boundary to the new face object.
4. Copy and paste the new face object to copy and paste the boundary.

Showing and Hiding Boundaries and Excitations in 2D

You can choose to show or hide a boundary or excitation's geometry, name, or vectors in the active view window or in all view windows.

Showing and Hiding 2D Boundaries and Excitations in Every View Window

1. Click **Maxwell 2D** > **Boundaries** > **Visualization** or **Maxwell 2D** > **Excitations** > **Visualization**.
   The **Boundary Visualization Options** dialog box appears.
2. Select one of the following from the **Select** drop-down menu:
   - All
   - All Boundaries
   - All Excitations
   - By Name
3. If you select **By Name**, enter a name in the **Select By Name** dialog box, and click **OK**.
4. To clear the selection of all boundaries and excitations, click **Deselect All**.
5. Click **Close** to close the **Boundary Visualization Options** dialog box.

Setting Default Values for 2D Boundaries and Excitations

When assigning a boundary or excitation, many of the fields in the boundary and excitation dialog boxes have default values associated with them. These default values are initially set by **Maxwell 2D**, but can be overridden.

To modify the default values associated with a specific boundary or excitation type:

1. Assign a boundary or excitation.
2. Modify any default values.
3. Close the boundary or excitation's dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Re-open the new boundary or excitation's dialog box by right-clicking the boundary or excitation in the project manager and selecting **Properties**. The dialog now includes a **Defaults** tab.
5. Under the **Defaults** tab, click **Save Defaults**.

   The values assigned to this boundary are saved as the default values and are assigned when new boundaries of this type are created.
6. Optionally, click **Revert to Standard Defaults**.

The default values you set for this boundary type are cleared and will revert to the default values set by **Maxwell 2D**.

Reassigning 2D Boundaries

You can reassign a boundary to another surface. This is useful when you modify objects with assigned boundaries, invalidating the boundaries. For example, if you unite two objects with assigned boundaries, the second object's boundary becomes invalid because united objects maintain the characteristics of the first object selected. In this case, you would need to reassign the boundary or delete it.

To reassign a boundary:

1. Select the object or object face to which you want to assign an existing boundary.
2. Click **Maxwell 2D** > **Boundaries** > **Reassign**.

   The **Reassign Boundary** dialog box appears.

   > **Tip:** Alternatively, select the object or object face to which you want to assign an existing boundary. Right-click the existing boundary in the project tree, and then click **Reassign** on the shortcut menu.
3. Select the boundary you want to reassign, and click **OK**.

> **Note:** When reassigning a boundary that includes vectors in its definition, Maxwell attempts to preserve the vectors with the new assignment, but this is not always possible.

Reassigning 2D Excitations

To reassign an excitation:

1. Select the object or object face to which you want to assign an existing excitation.
2. Click **Maxwell 2D** > **Excitations** > **Reassign**.

   The **Reassign Excitation** dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Tip:** Alternatively, select the object or object face to which you want to assign an existing excitation. Right-click the existing excitation in the Project Manager tree, and then click **Reassign** on the shortcut menu.

3. Select the excitation you want to reassign, and click **OK**.

> **Note:** When reassigning an excitation that includes vectors in its definition, Maxwell attempts to preserve the vectors with the new assignment, but this is not always possible.

Deleting All 2D Boundaries and/or Excitations

To delete all boundaries or excitations:

- Click **Maxwell 2D** > **Boundaries** > **Delete All** or **Maxwell 2D** > **Excitations** > **Delete All**.

Viewing and Editing 2D Boundary or Excitation Properties

To open the **Properties** dialog box for a boundary or excitation that is already assigned:

1. In the Project Manager tree, under either **Boundaries** or **Excitations**, right-click a specific boundary or excitation, and select **Properties**.
2. In the **Properties** dialog box that appears, make the desired changes, and click **OK**.

2D Magnetostatic Boundaries and Excitations

Define at least one of the following as a source of static magnetic fields:

- The current in a conduction path, which can be either stranded or solid.
- The current density in a conductor.
- The magnetic field on an outside surface.

2D Magnetostatic Boundaries

The magnetostatic field solver allows you to define the following types of boundaries:

Boundary Type	H-Field Behavior	Used to model...
Default Boundary Conditions (Natural and	Field behaves as follows: • At the interface between objects, the H tangent and B normal are continuous.	Initially, object interfaces are natural boundaries; outer boundaries, and excluded objects are Neumann boundaries.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Neumann)	• On outer boundaries, B field is normal. As a rule, this behavior needs to be modified at least for some portions of the outer boundary to insure uniqueness of the solution.	
Vector Potential	Sets the magnetic vector potential A_Z, or rA_ϕ, on the boundary. The behavior of H depends on whether A_Z or rA_ϕ is constant or functional.	Outer boundaries at specific vector potentials; externally applied magnetic fields.
Balloon	This boundary is considered to be far away from the other sources in the problem.	Actual behavior is context dependent. In general, if a Dirichlet BC is not applied elsewhere to insure uniqueness, the "balloon" will be assigned a Dirichlet boundary condition.
Symmetry	Field behaves as follows: • Odd Symmetry (Flux Tangential) — **Magnetic Field** is tangential to the boundary; its normal components are zero. • Even Symmetry (Flux Normal) — **Magnetic Field** is normal to the boundary; its tangential components are zero.	Planes of geometric and magnetic symmetry.
Matching (Independent and Dependent)	The **Magnetic Field vector** on the dependent boundary is forced to match the magnitude and direction (or the negative of the direction) of the **Magnetic Field** vector on the independent boundary.	Planes of symmetry in periodic structures where **Magnetic Field** is oblique to the boundary.

Default Boundary Conditions

These boundary conditions are automatically defined for a magnetostatic model:

- Natural boundaries are assigned to the surfaces between objects. At the interface between objects, the H tangent and B normal are continuous.
- Neumann boundaries are assigned to the outside edges of the problem region. On outer boundaries, B field is normal. As a rule, this behavior needs to be modified at least for some portions of the outer boundary to insure uniqueness of the solution.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To leave a surface set to its default boundary condition, do nothing. Deleted boundary conditions and excitations automatically reset to the default boundary conditions.

Assigning a Vector Potential

To define a vector potential boundary for a 2D Magnetostatic solver:

1. Select the section of the geometry on which you want to apply the boundary condition (typically an edge). **Maxwell 2D** requires that the Vector Potential be assigned on a 1D object such as an edge or a line.
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Vector Potential**.

   The **Vector Potential Boundary** dialog box appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the value of the potential. The value may be specified as a single numeric value or a functional value.
5. If a functional value is specified, select a **Coordinate System** from the drop-down menu:
6. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
7. Click **OK** to assign the boundary to the selected object.

The new boundary is added to the boundary list in the project tree.

Assigning a Symmetry Boundary Condition

For the 2D Magnetostatic Solver, this boundary condition defines a plane of geometric or magnetic symmetry in a structure. Assign it only to the outer surfaces of the problem region.

To set a symmetry boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically an edge).
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Symmetry**.

   The **Symmetry Boundary** dialog box appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Select one of the following as the type of symmetry:

Odd (Flux Tangential)	**Magnetic Field** is tangential to the boundary; its normal components are zero.
Even (Flux Normal)	**Magnetic Field** is normal to the boundary; its tangential components are zero.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
6. Click **OK** to assign the boundary to the selected object.

Assigning a Balloon Boundary Condition

To set a balloon boundary for the 2D Magnetostatic solver:

1. Select the section of the geometry on which you want to apply the boundary condition (typically an edge).
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Balloon**.

   The **Balloon Boundary** dialog box appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Click **OK** to assign the boundary to the selected object.

Assigning an Independent Boundary

Independent and dependent boundaries enable you to model planes of periodicity where the **Magnetic Field** at every point on the dependent boundary surface is forced to match the **Magnetic Field** of every corresponding point on the independent boundary surface.

To set a independent boundary for the 2D Magnetostatic solver:

1. Select the edge to which you want to assign the independent boundary.
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Matching** > **Independent**.

   The **Independent Boundary** dialog box appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. To reverse the direction of the vector, select the **Reverse Direction** check box.
5. Click **OK**.

Assigning a Dependent Boundary

Assigning a dependent boundary is the second step in creating matching boundaries. The field on the independent boundary is mapped to the dependent boundary.

> **Note:** You must define a independent boundary before creating the dependent boundaries that are associated with it. Assign dependent boundaries only to the outer surfaces of the problem region.

To set a dependent boundary for the 2D Magnetostatic solver:

1. Select the section of the geometry on which you want to apply the boundary condition (typically an edge).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Matching** > **Dependent**.

   The **Dependent Boundary** dialog box appears.

3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Select the independent boundary to which the dependent is to be assigned from the **Independent** drop-down menu.
5. In the **Relation** section, select one of the following to specify the field behavior on the boundary:

Bdep = Bind	Click this radio button if the magnetic fields on the dependent and independent boundaries have the same magnitude and direction.
Bdep = -Bind	Click this radio button if the magnetic fields on the dependent boundary have the same magnitude as but the opposite direction from the independent boundary.

6. To reverse the direction of the vector, select the **Reverse Direction** check box.
7. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
8. Click **OK** to assign the boundary to the selected object.

2D Magnetostatic Excitations

The following excitations of magnetic fields are available for magnetostatic problems:

Excitation	Type of Excitation
Current Density	The known current density distribution in the cross-section of a conductor
Current	Total current in a conductor
Winding With Current	Current for both a stranded and solid conductor
Winding With Resistance Voltage Drop	Resistance Voltage Drop for both a stranded and solid conductor

In addition, permanent magnets materials serve as sources of magnetic fields.

Assigning a Current Density Excitation

For the 2D Magnetostatic solver, this command (depending upon solution type, either **X-Y** or **R-Z**) specifies the applicable components of the current density in the cross-section of an object. If

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the current density is a function of position, the value is entered in ampere/m^2, even if you change the units in the problem.

To define the current density:

1. Select the section of the geometry (i.e., the conductor) on which you want to apply the excitation (typically a 2D object).
2. Click **Maxwell 2D** > **Excitations** > **Assign** > **Current Density**.

   The **Current Density Excitation** dialog box appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, specify the following options:
   a. Enter a value for the current density in the **Value** box. The value may be a single numeric quantity or a functional quantity.
   b. If a functional quantity is entered in the **Value** box, which depends on the applicable coordinates, select a **Coordinate System** from the drop-down menu.
5. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
6. Click **OK** to assign the excitation to the selected object.

Assigning a Current Excitation

For the 2D Magnetostatic solver, the current excitation specifies the total current in the cross-section of an object.

To set a current excitation:

1. Select the section of the geometry (i.e., the conductor) on which you want to apply the excitation (typically a face or other 2D planar object).
2. Click **Maxwell 2D** > **Excitations** >**Assign** > **Current**.

   The **Current Excitation** dialog box appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, specify the following options:
   a. Enter a value for the current in the **Value** box, and select the units from the drop-down menu.
   b. Select **Positive** or **Negative** as the **Ref. Direction**.
5. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
6. Click **OK** to assign the excitation to the selected object.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning a Winding Setup

> **Note:** If you wish to define a winding for a 2D AC Magnetic solution type, refer to Assigning a Winding Setup for a 2D AC Magnetic Solver.
> If you wish to define a winding for a 2D Transient solution type, refer to Assigning a Winding Setup for a 2D Transient Solver.

You can use the predefined coil terminal(s) to define one or more current or voltage windings.

To define a winding for a 2D Magnetostatic solver:

1. Click **Maxwell 2D** > **Excitations** > **Add Winding** to open the **Winding** dialog box.
2. Enter a name for the winding in the **Name** box, or accept the default.
3. In the **Parameters** section, select **Current**, or **Resistance Voltage Drop** from the **Type** drop-down menu.
4. Select the **Solid** or **Stranded** radio button to specify the type of conductor.
5. Enter values in the following fields, and select the desired units:

For current winding	For resistance voltage drop winding[1]
Current	Resistance
	Voltage

> **Note:** You can also type a function as an expression for any of these items.

6. Enter a value in the **Number of parallel branches** text box.

> **Note:** Parallel branches for solid windings assume there are no circulating currents in the parallel branches

7. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
8. Click **OK** to assign the excitation to the selected object

The definition of a winding is final only after specifying the coils which belong to it. **To add a coil terminal to this winding**, do the following:

1. In the Project Manager tree, right-click the winding, and select **Add Coils**.

   The **Add Terminals** dialog box appears, listing all coil terminals that do not already belong to that winding.

2. Select the coil terminal (s) you want to add. To select multiple terminals, press **Ctrl** and

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

click each terminal.

3. Click **OK**.

To assign a coil terminal excitation and add it to this winding, do the following:

1. Select the section of the geometry on which you want to apply the coil terminal excitation.
2. In the Project Manager tree, right-click the winding, and select **Assign Coil**.

   The **Coil Excitation** dialog box appears.

3. Enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the number of conductors for the coil in the **Number of Conductors** box.

   > **Note:** This value represents only the number of conductors inside of the selected geometry. If the Coil Terminal is cut due to symmetry, then only enter the Number of Conductors in the portion modeled.

5. Select one of the following for the **Polarity**:
   - **Positive**
   - **Negative**
   - **Function**
6. If you selected **Function** as the **Polarity**, enter a function in the text box.
7. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
8. Click **OK** to assign the coil terminal excitation.
   The coil terminal excitation is assigned and is added to the winding.

To delete all coil terminal excitations that belong to this winding, do the following: In the Project Manager tree, right-click the winding, and select **Delete All Coils**.
All coil terminal excitations are removed from the winding and deleted from the excitations.

---

1. For a solid winding, the resistance term can represent: the resistance of a portion of the winding which is not modeled (for instance, end-effects), the leads connecting the winding to the source, or the source resistance. (The main winding resistance is calculated directly by the solver.) For a stranded winding, the resistance term is the complete DC resistance of the winding (since the solver does not determine resistance of a stranded winding) as well as the resistance of the end-effects, leads, source, etc. For both solid and stranded windings, the inductance term can represent: the extra inductance for a portion of the winding of the winding which is not modeled (for instance, end-effects), the leads connecting the winding to the source, or the source inductance. (The main winding inductance itself is calculated directly by the solver.)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning a Coil

To assign a coil as a 2D magnetostatic excitation:

1. Select the section of the geometry on which you want to apply the excitation (typically a 2D object).
2. Click **Maxwell 2D** > **Excitations** > **Assign** > **Coil**. (You can also right-click an existing winding in the project tree, and select **Assign Coil**.)
3. In the **Coil Excitation** dialog box that appears, enter a name for the excitation in the **Name** box or accept the default.
4. In the **Parameters** section, enter the number of conductors for the coil in the **Number of Conductors** box.
5. Select one of the following for the **Polarity**:
    - **Positive**
    - **Negative**
    - **Function**
6. If you selected **Function** as the **Polarity**, enter a function in the text box.
7. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
8. Click **OK** to assign the coil to the selected object.

To add this coil to an existing winding, do the following:

1. In the Project Manager tree, right-click the coil terminal, and select **Add to Winding**.
2. In the**Add to Winding** dialog box, select the winding to which you want to add the coil terminal, and click **OK**.

To remove a coil terminal from an existing winding, do one of the following:

- In the Project Manager tree, right-click the coil terminal, and select **Remove from Winding**.
  The terminal is removed from the winding and moved up one level in the project tree (directly beneath **Excitations**).
- Alternatively, all coils assigned to a winding may be deleted by right-clicking the winding in the Project Manager tree and selecting **Delete All Coils**.

2D Electrostatic Boundaries and Excitations

Specify at least one of the following excitations as a source of electric fields:

- The charge on a surface or object.
- The charge density inside an object.
- Define the electric potential on each surface and object using a voltage excitation.

Include at least one of the following as a reference for computing the electric potential:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- A voltage excitation.
- An odd symmetry (flux normal) boundary.

2D Electrostatic Boundaries

The following boundary conditions are available for electrostatic problems:

Boundary Type	E-Field Behavior	Used to model...
Default Boundary Conditions (Natural and Neumann)	Field behaves as follows: • Natural boundaries — The normal component of **D** changes by the amount of surface charge density, tangent component of **E** is continuous. No special conditions are imposed. • Neumann boundaries — **E** is tangential to the boundary. Flux cannot cross a Neumann boundary.	Ordinary E-field behavior on boundaries. Object interfaces are initially set to natural boundaries; outer boundaries are initially set to Neumann boundaries.
Symmetry	Field behaves as follows: • Even Symmetry (Flux Tangential) — **E** is tangential to the boundary; its normal components are zero. • Odd Symmetry (Flux Normal) — **E** is normal to the boundary; its tangential components are zero.	Planes of geometric and electrical symmetry.
Balloon	Two options are available: Charge — The charge at “infinity” balances the charge in the drawing region. The net charge is zero. Use for capacitance calculations. Voltage — The voltage at “infinity” is zero.	Electrically insulated structures (Charge option) or electrically grounded structures (Voltage option). For the voltage case, the balloon boundary will not be equipotential in general.
Matching (Independent and Dependent)	The E-field vector on the dependent boundary is forced to match the magnitude and direction (or the negative of the direction) of the E-field vector on the independent boundary.	Planes of symmetry in periodic structures where **E** is oblique to the boundary.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Default Boundary Conditions

These boundary conditions are automatically defined for a 2D electrostatic model:

- Natural boundaries are assigned to the interfaces between objects. The normal component of **D** changes by the amount of surface charge density, tangent component of **E** is continuous.
- Neumann boundaries are assigned to the outside edges of the problem region. **E** is tangential to the boundary. Flux cannot cross a Neumann boundary.

To leave a surface set to its default boundary condition, do nothing. Deleted boundary conditions and excitations automatically revert to the default boundary conditions.

Assigning a Symmetry Boundary Condition

For the 2D Electrostatic solver, this boundary condition defines a plane of geometric or electric symmetry in a structure. Assign it only to the outer surfaces of the problem region.

To set a symmetry boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically an edge).
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Symmetry**.

   The **Symmetry Boundary** dialog box appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Select one of the following as the type of symmetry:

Odd (Flux Normal)	**E** is normal to the boundary; its tangential components are zero.
Even (Flux Tangential)	**E** is tangential to the boundary; its normal components are zero.

5. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
6. Click **OK** to assign the boundary to the selected object.

Assigning a Balloon Boundary Condition

To set a balloon boundary for the 2D Electrostatic Solver :

1. Select the section of the geometry on which you want to apply the boundary condition (typically an edge).
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Balloon**.

   The **Balloon Boundary** dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Select **Voltage** or **Charge** as the **Balloon Type**.
5. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
6. Click **OK** to assign the boundary to the selected object.

Assigning a Independent Boundary

Independent and dependent boundaries enable you to model planes of periodicity where the E-field vector at every point on the dependent boundary surface is forced to match the E-field vector of every corresponding point on the independent boundary surface. The transformation used to map the E-field from the independent to the dependent is determined by specifying the orientation of a local, predefined coordinate system, indicated by an arrow, on both the independent and dependent boundaries.

To set an independent boundary for the 2D Electrostatic solver :

1. Select the edge to which you want to assign the independent boundary.
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Matching** > **Independent**.

   The **Independent Boundary** dialog box appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. To reverse the direction of the vector, select the **Reverse Direction** check box.
5. Click **OK**.

Assigning a Dependent Boundary

Assigning a dependent boundary is the second step in creating matching boundaries. The field on the Independent boundary is mapped to the dependent boundary.

> **Note:** You must define an Independent boundary before creating the dependent boundaries that are associated with it. Assign dependent boundaries only to the outer surfaces of the problem region.

To set a dependent boundary for the 2D Electrostatic solver:

1. Select the section of the geometry on which you want to apply the boundary condition (typically an edge).
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Matching** > **Dependent**.

   The **Dependent Boundary** dialog box appears.
3. Enter a name for the boundary in the **Name** box, or accept the default.
4. Select the independent boundary to which the dependent is to be assigned from the **Independent** drop-down menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. In the **Relation** section, select one of the following to specify the field behavior on the boundary:

Edep = Eind	Click this radio button if the dependent and independent boundaries have the same magnitude and direction.
Edep = -Eind	Click this radio button if the dependent boundary field has the same magnitude as but the opposite direction from the independent boundary field.

6. To reverse the direction of the vector, select the **Reverse Direction** check box.
7. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
8. Click **OK** to assign the boundary to the selected object.

2D Electrostatic Excitations

The following excitations are available for electrostatic problems:

Excitation	Type of Excitation
Voltage	DC voltage on a surface or object
Charge	Total charge on a surface or object.
Floating	Models conductors at unknown potentials
Charge Density	Charge density over an object (dielectric)

Assigning a Voltage Excitation

Voltage excitations are identical to voltage boundaries in previous versions of Maxwell. A voltage excitation can be assigned to any surfaces or 2D objects. These assignments could include spatial functions.

For the 2D Electrostatic solver, this type of excitation sets the electric potential to a specific value.

To set a voltage excitation:

1. Select the section of the geometry on which you want to apply the excitation.
2. Click **Maxwell 2D** > **Excitations** > **Assign** > **Voltage**.

   The **Voltage Excitation** dialog box appears.

3. Enter a name for the excitation in the **Name** box, or accept the default.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. In the **Parameters** section, enter the electric potential in the **Value** box, and select the units from the drop-down menu. You can enter a numerical value, parameter (design or project variable) or a spatial function.
5. For a spatial function, select a coordinate system from the **Coordinate System** drop-down menu.
6. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
7. Click **OK** to assign the excitation to the selected object.

Assigning a Charge Excitation

For the 2D Electrostatic solver, this type of excitation defines the total charge on a surface or object. The potential distribution on the object with charge excitation is computed during the solution.

To define a charge excitation:

1. Select the section of the geometry on which you want to apply the excitation.
2. Click **Maxwell 2D** > **Excitations** > **Assign** > **Charge**.
3. In the **Charge Excitation** dialog box that appears, enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the numerical value, parameter (design or project variable) or a spatial function in the **Value** box.
5. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
6. Click **OK** to assign the excitation to the selected object.

Assigning a Floating Excitation

For the 2D Electrostatic solver, this type of excitation models conductors at unknown potentials and specifies the total charge on the conductor.

To define a floating conductor:

1. Select the section of the geometry on which you want to apply the excitation. This can be a 2D or 1D geometry entity. If it is a 2D object, it doesn't have to be a conductor (but it will behave like one when the solution becomes available).
2. Click **Maxwell 2D** > **Excitations** > **Assign** > **Floating**.
3. In the **Floating Excitation** dialog box that appears, click the **General** tab.
4. Enter a name for the excitation in the **Name** box, or accept the default.
5. In the **Parameters** section, enter the charge value or parameter in the **Value** box.
6. Optionally, click **Use Defaults** to revert to the default values in the dialog box. You can also click the **Defaults** tab to save new default settings or to revert to the standard default values.
7. Click **OK** to assign the excitation to the selected object.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning a Charge Density Excitation

For the 2D Electrostatic solver, this type of excitation defines the volume charge density on an object.

To define the charge density on a dielectric object:

1. Select the object on which you want to apply the excitation.
2. Click **Maxwell 2D** > **Excitations** > **Assign** > **Charge Density**.
3. In the **Charge Density Excitation** dialog box that appears, enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the charge density in the **Value** box. You can enter a numerical value, parameter (design or project variable) or a spatial function.
5. For a spatial function, select the type of coordinate system from the **Coordinate System** drop-down menu.
6. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
7. Click **OK** to assign the excitation to the selected object.

2D AC Conduction Boundaries and Excitations

The following excitations can be specified as the source of electric fields:

- The voltage difference between surfaces and objects. Define the electric potential on each surface and object using a voltage excitation.

Include at least one of the following as a reference for computing the electric potential:

- A voltage excitation.
- An odd symmetry (flux normal) boundary.

2D AC Conduction Boundaries

The following boundary conditions are available for AC conduction electric problems:

Boundary Type	E-Field Behavior	Used to model...
Default Boundary Conditions (Natural and Neumann)	Field behaves as follows: • Natural boundaries - The normal component of **J** and the tangential component of **E** are continuous across discontinuity surfaces. No special conditions are imposed.	Object interfaces are initially set to natural boundaries; outer boundaries are initially set to Neumann boundaries.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	• Neumann boundaries - **E** and **J** vectors are tangential to the boundary. Flux cannot cross a Neumann boundary.	
Symmetry	Field behaves as follows: • Even Symmetry (Flux Tangential) - **E and J vectors are** tangential to the boundary; its normal components are zero. • Odd Symmetry (Flux Normal) - **E and J vectors are** normal to the boundary; its tangential components are zero.	Planes of geometric and electrical symmetry.
Balloon	Balloon boundaries model the region outside the drawing space as being nearly infinitely large — effectively isolating the model from other voltage sources.	Electrically insulated structures, equipotential lines can cross this type of boundary. In general, they are neither tangent nor perpendicular to the balloon boundary.
Matching (Independent and Dependent)	The E-field on the dependent boundary is forced to match the magnitude of the E-field on the independent boundary if the plus sign is used. If the minus sign is used, the field on the dependent boundary will oscillate with opposite phase angle (180 degrees phase shift).	Planes of symmetry in periodic structures where **E** is oblique to the boundary.

Default Boundary Conditions

These boundary conditions are automatically defined for a AC conduction model:

- Natural boundaries are assigned to the surfaces between objects. The normal component of **J** and the tangential component of **E** are continuous across discontinuity surfaces.
- Neumann boundaries are assigned to the outside edges of the problem region. **E** and **J** vectors are tangential to the boundary. Flux cannot cross a Neumann boundary.

To leave a surface set to its default boundary condition, do nothing. Deleted boundary conditions and excitations automatically revert to the default boundary conditions.

Assigning a Symmetry Boundary Condition

For the 2D AC Conduction solver, this boundary condition defines a plane of geometric or electric symmetry in a structure. Assign it only to the outer surfaces of the problem region.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To set a symmetry boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically an edge).
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Symmetry**.
3. In the **Symmetry Boundary** dialog box that appears, enter a name for the boundary in the **Name** box, or accept the default.
4. Select one of the following as the type of symmetry:

Odd (Flux Normal)	Voltages have opposite sign on the two sides of the symmetry plane.
Even (Flux Tangential)	Voltages have same sign on the two sides of the symmetry plane.

5. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
6. Click **OK** to assign the boundary to the selected object.

Assigning a Balloon Boundary Condition

To set a balloon boundary for the 2D AC Conduction solver :

1. Select the section of the geometry on which you want to apply the boundary condition (typically an edge).
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Balloon**.
3. In the **Balloon Boundary** dialog box that appears, enter a name for the boundary in the **Name** box, or accept the default.
4. Click **OK** to assign the boundary to the selected object.

Assigning an Independent Boundary

Independent and dependent boundaries enable you to model planes of periodicity where the **E**-field at every point on the dependent boundary surface is forced to match the **E**-field of every corresponding point on the independent boundary surface.

To set an independent boundary for the 2D AC Conduction solver:

1. Select the edge to which you want to assign the independent boundary.
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Matching** > **Independent**.
3. In the **Independent Boundary** dialog box that appears, enter a name for the boundary in the **Name** box, or accept the default.
4. To reverse the direction of the vector, select the **Reverse Direction** check box.
5. Click **OK**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Maxwell solver computes the E-field on this boundary and maps it to the dependent boundary.

Assigning a Dependent Boundary

Assigning a dependent boundary is the second step in creating matching boundaries. The field on the Independent boundary is mapped to the dependent boundary.

> **Note:** You must define an Independent boundary before creating the dependent boundaries that are associated with it. Assign dependent boundaries only to the outer surfaces of the problem region.

To set a dependent boundary for the 2D AC Conduction solver:

1. Select the section of the geometry on which you want to apply the boundary condition (typically an edge).
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Matching** > **Dependent**.
3. In The **Dependent Boundary** dialog box that appears, enter a name for the boundary in the **Name** box, or accept the default.
4. In the **Parameters** section, select the independent boundary to which the dependent is to be assigned from the **Independent** drop-down menu.
5. Under **Relation**, select one of the following to specify the field behavior on the boundary:

Edep = Eind	Click this radio button if the dependent and independent boundaries have the same magnitude and oscillate in phase.
Edep = -Eind	Click this radio button if the dependent boundary field has the same magnitude as the independent boundary field and oscillate 180 degrees out of phase.

6. To reverse the direction of the vector, select the **Reverse Direction** check box.
7. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
8. Click **OK** to assign the boundary to the selected object.

2D AC Conduction Excitations

The following excitations are available for electric AC conduction problems:

Excitation	Type of Excitation
Voltage	The AC voltage on a surface or object. This excitation type is available for all three of the 2D electric solvers.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** All field quantities oscillate with the same frequency, equal to the frequency specified in the solution setup. The phase of quantities computed in the solution space however will be different in general.

Assigning a Voltage Excitation

For the 2D AC Conduction solver, this type of excitation sets the electric potential (voltage) on a surface or object to a specific value.

To set a voltage excitation:

1. Select the section of the geometry on which you want to apply the excitation.
2. Click **Maxwell 2D** > **Excitations** > **Assign** > **Voltage**.
3. In the **Voltage Excitation** dialog box that appears, enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the electric potential in the **Value** box, and select the units from the drop-down menu. You can enter a numerical value or a spatial function.
5. For a spatial function, select a coordinate system from the **Coordinate System** drop-down menu.
6. Enter a value in the **Phase** text box, and select a unit from the drop-down menu.
7. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
8. Click **OK** to assign the excitation to the selected object.

2D DC Conduction Boundaries and Excitations

The following excitations can be specified as the source of electric fields:

- The voltage difference between surfaces and objects. Define the electric potential on each surface and object using a voltage excitation.

Include at least one of the following as a reference for computing the electric potential:

- A voltage excitation.
- An odd symmetry (flux normal) boundary.

2D DC Conduction Boundaries

The following boundary conditions are available for DC conduction electric problems:

Boundary Type	E-Field Behavior	Used to model...
Default	Field behaves as follows:	Object interfaces are initially

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Boundary Conditions (Natural and Neumann)	• Natural boundaries — Normal component of **J** and tangential component of **E** are continuous across discontinuity surfaces. No special conditions are imposed. • Neumann boundaries — **E** and **J** vectors are tangential to the boundary. Flux cannot cross a Neumann boundary.	set to natural boundaries; outer boundaries are initially set to Neumann boundaries.
Symmetry	Field behaves as follows: • Even Symmetry (Flux Tangential) — **E** and **J** vectors are tangential to the boundary; their normal components are zero. • Odd Symmetry (Flux Normal) — **E** and **J** vectors are normal to the boundary; their tangential components are zero.	Planes of geometric and electrical symmetry.
Balloon	Balloon boundaries model the region outside the drawing space as being nearly “infinitely” large — effectively isolating the model from other voltage sources.	Electrically insulated structures.
Resistance	A resistance boundary models a very thin layer of resistive material (such as that caused by deposits, coatings or oxidation on a metallic surface) on a conductor at a known potential.	Use this boundary condition when the resistive layer’s thickness is much smaller than the other dimensions of the model.
Matching (Independent and Dependent)	The **E**-field on the dependent boundary is forced to match the magnitude and direction (or the negative of the direction) of the **E**-field on the independent boundary.	Planes of symmetry in periodic structures where **E** is oblique to the boundary.

Default Boundary Conditions

These boundary conditions are automatically defined for a DC conduction model:

- Natural boundaries are assigned to the surfaces between objects. Normal component of **J** and tangential component of **E** are continuous across discontinuity surfaces. No special conditions are imposed.
- Neumann boundaries are assigned to the outside edges of the problem region. **E** and **J** vectors are tangential to the boundary. Flux cannot cross a Neumann boundary.

To leave a surface set to its default boundary condition, do nothing. Deleted boundary conditions and excitations automatically revert to the default boundary conditions.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning a Symmetry Boundary Condition

This boundary condition defines a plane of geometric or electric symmetry in a structure. Assign it only to the outer surfaces of the problem region.

To set a symmetry boundary for a 2D DC Conduction solver:

1. Select the section of the geometry on which you want to apply the boundary condition (typically an edge).
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Symmetry**.
3. In the **Symmetry Boundary** dialog box that appears, enter a name for the boundary in the **Name** box, or accept the default.
4. Select one of the following as the type of symmetry:

Odd (Flux Normal)	Voltages have opposite signs on the two sides of the symmetry plane.
Even (Flux Tangential)	Voltages have same signs on the two sides of the symmetry plane.

5. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
6. Click **OK** to assign the boundary to the selected object.

Assigning a Balloon Boundary Condition

To set a balloon boundary for a 2D DC Conduction solver:

1. Select the section of the geometry on which you want to apply the boundary condition (typically an edge).
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Balloon**.
3. In the **Balloon Boundary** dialog box that appears, enter a name for the boundary in the **Name** box, or accept the default.
4. Click **OK** to assign the boundary to the selected object.

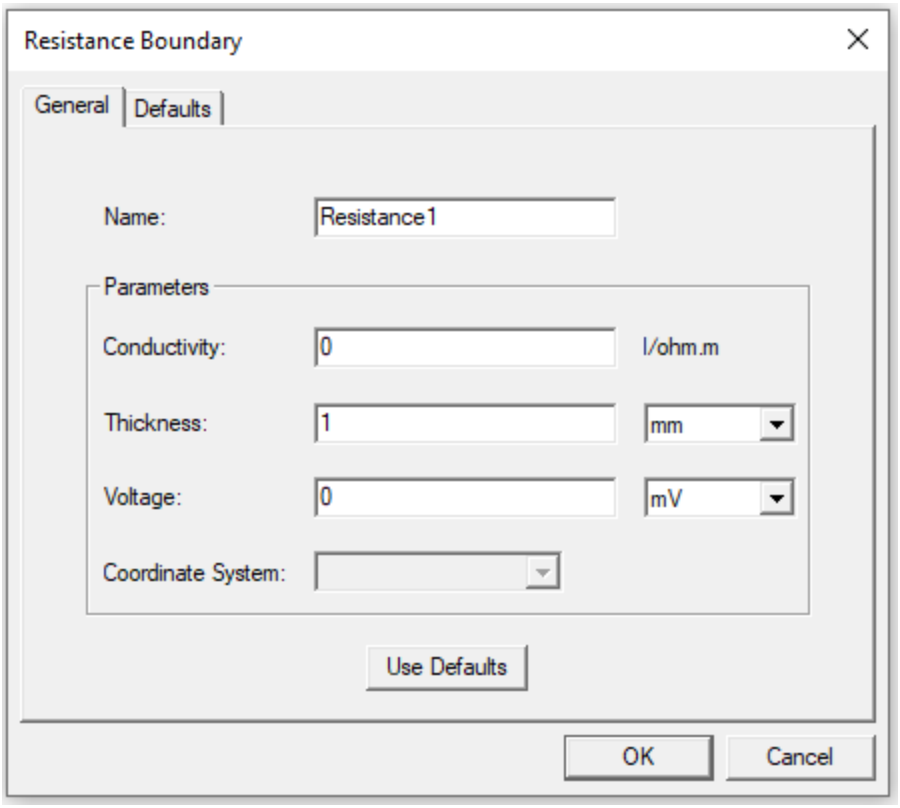
Assigning a Resistance Boundary Condition

To set a resistance boundary for a 2D DC Conduction solver:

1. Select the edge of the geometry on which you want to apply the boundary condition.
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Resistance**.
3. In the **Resistance Boundary** dialog box that appears, enter a name for the boundary in the **Name** box, or accept the default.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Enter the boundary parameters in the **Conductivity**, **Thickness**, and **Voltage** fields, and select the units of measure. Use of variables is supported.



5. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
6. Click **OK** to assign the boundary to the selected object.

> **Note:** Apply resistance boundaries only to the outside edge of the problem space. The external region touching to a resistance boundary is assumed to be a PEC (perfect conductor) with a known potential value that is specified in the resistance boundary setup. Refer to *Resistance* for technical details.

Assigning an Independent Boundary

Independent and dependent boundaries enable you to model planes of periodicity where the **E**-field at every point on the dependent boundary surface is forced to match the **E**-field of every corresponding point on the independent boundary surface.

To set an independent boundary for a 2D DC Conduction solver:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the edge to which you want to assign the independent boundary.
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Matching** > **Independent**.
3. In the **Independent Boundary** dialog box that appears, enter a name for the boundary in the **Name** box, or accept the default.
4. To reverse the direction of the vector, select the **Reverse Direction** check box.
5. Click **OK**.

Assigning a Dependent Boundary

Assigning a dependent boundary is the second step in creating matching boundaries. The field on the independent boundary is mapped to the dependent boundary.

> **Note:** You must define an Independent boundary before creating the dependent boundaries that are associated with it. Assign dependent boundaries only to the outer surfaces of the problem region.

To set a dependent boundary for a 2D DC Conduction solver:

1. Select the section of the geometry on which you want to apply the boundary condition (typically an edge).
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Matching** > **Dependent**.
3. In the **Dependent Boundary** dialog box that appears, enter a name for the boundary in the **Name** box, or accept the default.
4. Select the independent boundary to which the dependent is to be assigned from the **Independent** drop-down menu.
5. In the **Relation** section, select one of the following to specify the field behavior on the boundary:

Edep = Eind	Click this radio button if the **E** field on the dependent and independent boundaries have the same magnitude and direction.
Edep = -Eind	Click this radio button if the **E** field on the dependent boundary field has the same magnitude as but the opposite direction from the independent boundary field.

6. To reverse the direction of the vector, select the **Reverse Direction** check box.
7. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
8. Click **OK** to assign the boundary to the selected object.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2D DC Conduction Excitations

The following excitations are available for electric DC conduction problems:

Excitation	Type of Excitation
Voltage	The DC voltage on a surface or object.

Assigning a Voltage Excitation

For the 2D DC Conduction solver, this type of excitation sets the electric potential (voltage) on a surface or object to a specific value.

To set a voltage excitation:

1. Select the section of the geometry on which you want to apply the excitation.
2. Click **Maxwell 2D** > **Excitations** > **Assign** > **Voltage**.
3. In the **Voltage Excitation** dialog box that appears, enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the electric potential in the **Value** box, and select the units from the drop-down menu. You can enter a numerical value or a spatial function.
5. For a spatial function, select a coordinate system from the **Coordinate System** drop-down menu.
6. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
7. Click **OK** to assign the excitation to the selected object.

2D AC Magnetic Boundaries and Excitations

Specify at least one of the following as a source of AC magnetic fields in your model:

- The current in the cross-section (object), which can be either stranded or solid.
- The current to be divided among several parallel conductors.
- The current density in an object.
- The external magnetic field.

If currents or current densities are the only sources of AC magnetic fields in your model, set at least one outer boundary to the following:

- The default boundary conditions.
- An odd symmetry (flux tangential) boundary.
- An even symmetry (flux normal) boundary.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AC Magnetic Boundaries

The AC Magnetic (Eddy Current) field solver allows you to define the following types of boundaries:

Boundary Type	H-Field Behavior	Used to model…
Default Boundary Conditions (Natural and Neumann)	Field behaves as follows: • Natural boundaries — Tangential **H** and normal **B** are continuous across surfaces without current density distribution; tangential **H** has a jump if the surface has current density distribution. • Neumann boundaries — **Magnetic Field** is tangential to the boundary and flux cannot cross it.	Initially, object interfaces are natural boundaries; outer boundaries and excluded objects are Neumann boundaries.
Vector Potential	Sets the magnetic vector potential $A_z(t)$, or $rA_\phi(t)$, on the boundary. The behavior of H depends on whether $A_z(t)$ or $rA_\phi(t)$ is constant or functional.	Magnetically isolated structures.
Impedance	Includes the effect of induced currents beyond the boundary surface based on tangent component of **H** along the impedance boundary.	Conductors with very small skin depths.
Symmetry	Field behaves as follows: • Odd Symmetry (Flux Tangential) — **Magnetic Field** is tangential to the boundary; its normal components are zero. • Even Symmetry (Flux Normal) — **Magnetic Field** is normal to the boundary; its tangential components are zero.	Planes of geometric and magnetic symmetry.
Balloon	Models the case where the structure is “infinitely” far away from other magnetic fields or current sources.	Magnetically isolated structures.
Matching (Independent and Dependent)	The **Magnetic Field** on the dependent boundary is forced to match the magnitude and direction of the **Magnetic Field** on the independent boundary. If a plus sign is used, the fields oscillate in phase, if a minus sign is used the fields will oscillate 180 degrees out of phase.	Planes of symmetry in periodic structures where **Magnetic Field** is oblique to the boundary.

Default Boundary Conditions

These boundary conditions are automatically defined for an AC Magnetic model:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Natural boundaries are assigned to the surfaces between objects. Tangential **H** and normal **B** are continuous across surfaces without current density distribution; tangential **H** has a jump if the surface has current density distribution.
- Neumann boundaries are assigned to the outside edges of the problem region. **Magnetic Field** is tangential to the boundary and flux cannot cross it.

To leave a surface set to its default boundary condition, do nothing. Deleted boundary conditions and excitations automatically reset to the default boundary conditions.

Assigning a Vector Potential

To define a vector potential boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically an edge).
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Vector Potential**.
3. In the **Vector Potential Boundary** dialog box that appears, enter a name for the boundary in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the value of the potential. The value may be specified as a single numeric value or as a spatial function.
5. If a functional value is specified, select a **Coordinate System** from the drop-down menu:
6. Enter a value in the **Phase** text box, and select a unit from the drop-down menu.
7. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
8. Click **OK** to assign the boundary to the selected object.

The new boundary is added to the boundary list in the project tree.

Assigning an Impedance Boundary

This boundary condition is used to simulate the effect of induced currents in a conductor without explicitly computing them. Since the conductor must be excluded in the model (saving time needed to mesh and solve for currents), assign the impedance boundary condition to an outside edge of the problem region or to an excluded object. Exclude the object from the problem region and make the object a perfect conductor in the Material Manager.

To define an impedance boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically an edge).
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Impedance**.
3. In the **Impedance Boundary** dialog box that appears, enter a name for the boundary in the **Name** box, or accept the default.
4. Enter the conductivity (in inverse ohm-meters) in the **Conductivity** field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Enter the conductor's relative permeability in the **Permeability** field.
6. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
7. Click **OK** to assign the boundary to the selected object.

2D Impedance Boundary

Impedance boundaries allow you to simulate the effect of induced currents in a conductor without explicitly computing them. Use this boundary condition for models where:

- The skin depth in the conductor is less than two orders of magnitude smaller than the dimensions of the structure. In models like this, the mesh generator may not be able to create a fine enough mesh in the conductor to compute eddy currents.
- The magnetic field decays much more rapidly inside the conductor in the direction that's normal to the surface than it does in directions that are tangential to the surface.
- The AC current source is relatively far away from the surface where eddy currents occur, compared to the size of the skin depth.

The conductor itself must be excluded from the solution region. When setting up the model, do one of the following:

- For external boundaries, when drawing the model, make the surface along which eddy currents are to be computed an outer surface of the problem region.
- For internal boundaries: assign a perfect conductor to the object in the Material Manager, and also exclude the object from the solution by unchecking **Solve Inside** under the object attributes. The solver does not find solutions inside a perfect conductor.

Then, when defining boundaries, assign an impedance boundary to the individual surfaces of the problem region (for external boundaries) or to the entire object (for internal boundaries). By entering the conductivity, s, and the relative permeability, μ_r, of the object, you specify the skin depth of induced eddy currents. The simulator uses this skin depth value when computing the electromagnetic field solution. It assumes that the **Magnetic Field** falls off exponentially inside the conductor. The ohmic loss due to induced currents can then be computed from the **Magnetic Vector Potential** along the impedance boundary – on the surface of the object that you are interested in – according to the formula below or by using the quantity **EdgeLossDensity** in the 2D calculator.

$$EdgeLossDensity\left(W/m^2\right) = \pi f \sqrt{\frac{\pi f \sigma}{\mu_0 \mu_r}} \left[A_z \cdot A_Z^*\right]$$

where

A_z = z component of the Magnetic Vector Potential

A_z^* = complex conjugate of the z component of the Magnetic Vector Potential

μ_0 = permeability of free space

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

μ_r = relative permeability of the impedance boundary

σ = conductivity of the impedance boundary (S/m)

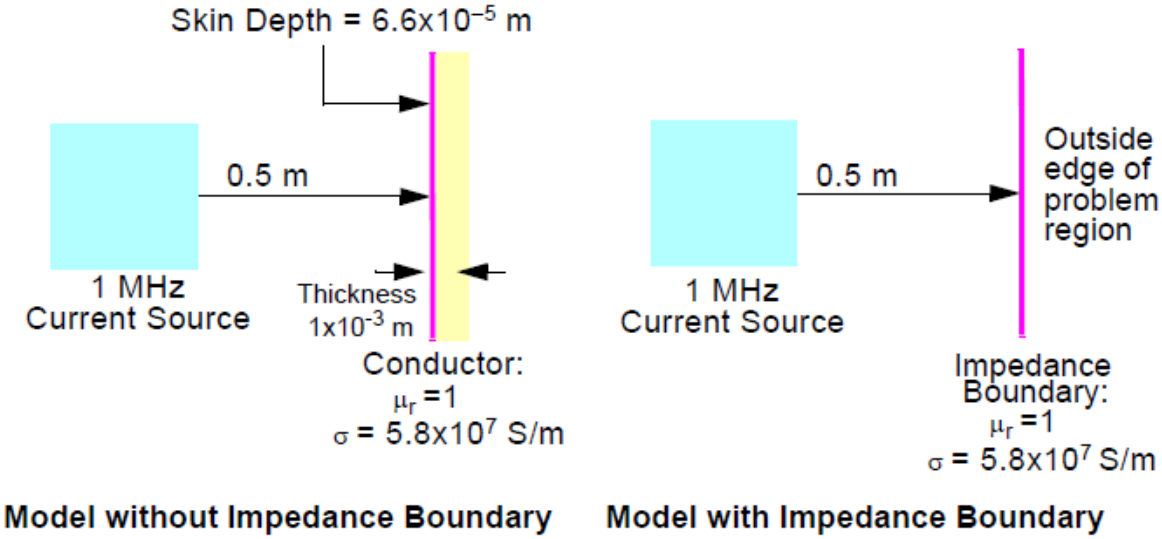
f = frequency (Hz)

You can also use **EdgeLossDensity** to plot the loss along an edge containing an impedance boundary.

> **Note:** An impedance boundary only approximates the effect of eddy currents acting at a shallow skin depth. It does not directly compute them. In general, the fields modeled using an impedance boundary will closely match the field patterns that would actually occur in the structure. However, at discontinuities in the surface (such as corners), the field patterns may be different.

When to Use Impedance Boundaries in 2D

A typical situation where impedance boundaries can be used to reduce the complexity of a model is shown below. Suppose you want to compute eddy current losses in the conductor next to the current source shown below on the left. If the source carries AC current at a frequency of 1 MHz, the skin depth in the conductor is 6.6 x 10–5 meters. This is several orders of magnitude smaller than the conductor's thickness. Since the conductor where currents are induced is also relatively far away from the current source, an impedance boundary can be used to model the induced currents — as shown on the right:



Model without Impedance Boundary **Model with Impedance Boundary**

The conductor must be excluded from the model by making it a perfect conductor. The outside boundary of the model is moved to the inside surface of the conductor. This outside surface is

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

defined as an impedance boundary, using the conductivity and permeability specified previously. Since the simulator does not have to actually compute a solution inside the conductor, the field solution is computed more quickly and uses less memory. After solving, you can compute the ohmic loss for the surface using the solution calculator and plot the loss density on the boundary.

Assigning a Symmetry Boundary

For the 2D AC Magnetic solver, this boundary condition defines a plane of geometric or magnetic symmetry in a structure. Assign it only to the outer surfaces of the problem region.

To set a symmetry boundary:

1. Select the section of the geometry on which you want to apply the boundary condition (typically an edge).
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Symmetry**.
3. In the **Symmetry Boundary** dialog box that appears, enter a name for the boundary in the **Name** box, or accept the default.
4. Select one of the following as the type of symmetry:

Odd (Flux Tangential)	Magnetic field is tangential to the boundary; its normal components are zero.
Even (Flux Normal)	Magnetic field is normal to the boundary; its tangential components are zero.

5. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
6. Click **OK** to assign the boundary to the selected object.

Assigning a Balloon Boundary Condition

To set a balloon boundary for the 2D AC Magnetic solver:

1. Select the section of the geometry on which you want to apply the boundary condition (typically an edge).
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Balloon**.
3. In the **Balloon Boundary** dialog box that appears, enter a name for the boundary in the **Name** box, or accept the default.
4. Click **OK** to assign the boundary to the selected object.

Assigning an Independent Boundary

For the 2D AC Magnetic solver, independent and dependent boundaries enable you to model planes of periodicity where the **Magnetic Field** at every point on the dependent boundary

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

surface is forced to match the **Magnetic Field** of every corresponding point on the independent boundary surface. The transformation used to map the **Magnetic Field** from the independent to the dependent is determined by specifying a coordinate system on both the independent and dependent boundaries.

To set an independent boundary:

1. Select the edge to which you want to assign the independent boundary.
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Matching** > **Independent**.
3. In the **Independent Boundary** dialog box that appears, enter a name for the boundary in the **Name** box, or accept the default.
4. To reverse the direction of the vector, select the **Reverse Direction** check box.
5. Click **OK**.

Assigning a Dependent Boundary

Assigning a dependent boundary is the second step in creating matching boundaries. The field on the independent boundary is mapped to the dependent boundary.

> **Note:** You must define an independent boundary before creating the dependent boundaries that are associated with it. Assign dependent boundaries only to the outer surfaces of the problem region.

To set a dependent boundary for the 2D AC Magnetic solver:

1. Select the section of the geometry on which you want to apply the boundary condition (typically an edge).
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Matching** > **Dependent**.
3. In the **Dependent Boundary** dialog box that appears, enter a name for the boundary in the **Name** box, or accept the default.
4. Select the independent boundary to which the dependent is to be assigned from the **Independent** drop-down menu.
5. In the **Relation** section, select one of the following to specify the field behavior on the boundary:

Bdep = Bind	Click this radio button if the magnetic fields on the dependent and independent boundaries have the same magnitude and the fields oscillate in phase.
Bdep = -Bind	Click this radio button if the magnetic fields on the dependent boundary have the same magnitude as the independent boundary field but the fields oscillate 180 degrees out

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of phase.

6. To reverse the direction of the vector, select the **Reverse Direction** check box.
7. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
8. Click **OK** to assign the boundary to the selected object.

AC Magnetic Excitations

The AC Magnetic (Eddy Current) solver allows you to define the following sources of AC magnetic fields:

Excitation	Type of Excitation
Current	The total current in a conductor
Parallel Current	The total current to be divided among several conductors in parallel. The method of dividing the current depends upon the choice of solid or stranded.
Current Density	The current density in a conductor. In this case, you must also define the current density terminals for the conductor.
Coil	Used to define one or more model windings.
End Connection	End connection is used to model details of the "squirrel cage" winding of induction machines, dampers in synchronous machines, etc.
Winding With Current	Current for both a stranded and solid conductor
Winding With Voltage	Voltage for both a stranded and solid conductor
Winding With External Circuit Connection	External circuit connection for both a stranded and solid conductor

> **Note:** The 2D AC Magnetic solver can be used to get harmonic losses on nonlinear systems and can be an alternative to a transient simulation. See Using the AC Magnetic Solver to Generate Harmonic Losses for more details.

Assigning a Current Excitation

For the 2D AC Magnetic solver, a current excitation specifies the total AC current in a conduction path. The conduction path may be contained completely within the problem region (for example, a coil), or may touch the edges of the problem region.

To specify the total AC current:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the 2D section of the geometry on which you want to apply the excitation.
2. Click **Maxwell 2D** > **Excitations** > **Assign** > **Current**.
3. In the **Current Excitation** dialog box that appears, enter a name for the excitation in the **Name** box, or accept the default.
4. Enter the current strength in the **Value** box.
5. Enter the phase for the current in the **Phase** box.
6. Select the **Solid** or **Stranded** radio button to specify the type of conductor.
7. Select either **Positive** or **Negative** as the **Ref. Direction** for the conductor.
8. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
9. Click **OK** to assign the excitation to the selected object.

Assigning a Parallel Current Excitation

For the 2D AC Magnetic solver, a parallel current excitation specifies the total AC current in a parallel conduction path consisting of at least two conductors.

To specify the total AC current:

1. Select the 2D sections of the geometry on which you want to apply the excitation. At least two section must be specified
2. Click **Maxwell 2D** > **Excitations** > **Assign** > **Parallel Current**.
3. In the **Parallel Current Excitation** dialog box that appears, enter a name for the excitation in the **Name** box, or accept the default.
4. Enter the current strength in the **Value** box.
5. Enter the phase for the current in the **Phase** box.
6. Select the **Solid** or **Stranded** radio button to specify the type of conductor.

   > **Note:** For parallel solid conductors, the current split will be based upon the overall field solution including eddy effects (the currents in individual parallel conductors will have complex values, which are proportional with the respective admittances, and in general will have different phase angles between them). If stranded conductors are specified, the current is split based upon the relative areas of the selected conductors.

7. Select either **Positive** or **Negative** as the **Ref. Direction** for the conductor.
8. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
9. Click **OK** to assign the excitation to the selected object.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning a Current Density Excitation

For the 2D AC Magnetic solver, this command specifies the applicable component (depending on XY or RZ solution type) of the AC current density. If the current density is a function of position, the value is entered in ampere/m^2, even if you change the units in the problem.

To define the current density:

1. Select the section of the geometry (for example, the conductor) in which you want to apply the excitation (typically a 2D object).
2. Click **Maxwell 2D** > **Excitations** > **Assign** > **Current Density**.
3. In the **Current Density Excitation** dialog box that appears, enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, specify the following options:
    a. Enter the current strength in the **Value** box. The value may be a single numeric value or a spatial function.
    b. If the value is specified as a spatial function, select the **Coordinate System** from the drop-down menu.
    c. Enter the phase angle, θ, of the current density in the **Phase** field, and select a unit from the drop-down menu.
5. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
6. Click **OK** to assign the excitation to the selected object.

Assigning a Coil

To assign a coil as an 2D AC Magnetic excitation:

1. Select the section of the geometry on which you want to apply the excitation (typically a 2D object).
2. Click **Maxwell 2D** > **Excitations** > **Assign** > **Coil**. (You can also right-click an existing winding in the project tree, and select **Assign Coil**.)
3. In the **Coil Excitation** dialog box appears, from the **General** tab, enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the number of conductors for the coil in the **Number of Conductors** box.
5. Select one of the following for the **Polarity**:
    - **Positive**
    - **Negative**
    - **Function**
6. If you selected **Function** as the **Polarity**, enter a function in the text box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Optionally, on the Defaults tab, click **Save Defaults** to save the current settings or click **Revert to Standard Defaults** to revert to the default values in the dialog box.
8. Click **OK** to assign the coil to the selected object.

To add this coil to an existing winding:

1. In the Project Manager tree, right-click the coil terminal, and select **Add to Winding**. The **Add to Winding** dialog box appears.
2. Select the winding to which you want to add the coil terminal, and click **OK**.

To remove a coil terminal from an existing winding:

- In the Project Manager tree, right-click the coil terminal, and select **Remove from Winding**. The terminal is removed from the winding and moved up one level in the project tree (directly beneath **Excitations**).
- Alternatively, all coils assigned to a winding may be deleted by right-clicking the winding in the project tree and selecting **Delete All Coils**.

Assigning an End Connection

For the 2D AC Magnetic solver, an End Connection excitation is used to model details of the "squirrel cage" winding of induction machines, dampers in synchronous machines, *etc*.

To define an end connection:

1. Select the section of the geometry (*i.e.*, the conductor) across which you want to apply the excitation.
2. Click **Maxwell 2D** > **Excitations** > **Assign** > **End Connection**.
3. In the **End Connection Excitation** dialog box that appears, enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, specify the following options:
    a. Enter the **End resistance between adjacent conductors**, and select a unit of measurement from the drop-down menu.
    b. Enter the **End inductance between adjacent conductors**, and select a unit of measurement from the drop-down menu.
5. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
6. Click **OK** to assign the excitation to the selected object.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning a Winding Setup

> **Note:**
> If you wish to define a winding for a 2D transient solution type, refer to Assigning a Winding Setup for a 2D Transient Solver.
> If you wish to define a winding for a 2D magnetostatic solution type, refer to Assigning a Winding Setup for a 2D Magnetostatic Solver.

For the 2D AC Magnetic solver, you can use the predefined coil terminal(s) to define one or more current or voltage windings.

To define a winding for your model:

1. Click **Maxwell 2D** > **Excitations** > **Add Winding** to open the **Winding** dialog box.
2. Enter a name for the winding in the **Name** box, or accept the default.
3. In the **Parameters** section:
   a. Select **Current**, **Voltage**, or **External** from the **Type** drop-down menu. The winding type selected enables only those fields (Initial Current, Resistance, Inductance, Voltage, Phase) applicable for the chosen type.
   b. Select the **Solid** or **Stranded** radio button to specify the type of conductor.
   c. For the 2D AC Magnetic solver, there is a **Get Databases from FFT** button. This feature allows the user to get frequency domain signals from transient signals, and solve on the frequencies in AC Magnetic designs. For more information, see Using an AC Magnetic Excitation to Generate Harmonic Losses.
   d. Enter values in the following fields (enabled according to winding type), and select the desired units:

For a current winding	For a voltage winding[1]	For an external winding
Current	Resistance	(no settings required)
Phase	Inductance	
	Voltage	
	Phase	

> **Note:** You can also type a function as an expression for any of these fields except Initial Current.

   e. Enter a value in the **Number of parallel branches** text box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** Parallel branches for solid windings assume there are no circulating currents in the parallel branches.

4. Optionally, click **Use Defaults** to revert to the default values in the dialog box
5. Optionally, on the Defaults tab, click **Save Defaults** to save the current settings or click **Revert to Standard Defaults** to revert to the default values in the dialog box.
6. Click **OK** to assign the excitation to the selected object.

The definition of a winding is final only after specifying the coils which belong to it. **To add a coil terminal to this winding**:

1. In the **Project Manager** tree, right-click the winding, and select **Add Coils**. The **Add Terminals** dialog box appears, listing all coil terminals that do not already belong to that winding.
2. Select the coil terminal (s) you want to add. To select multiple terminals, press **Ctrl** and click each terminal.
3. Click **OK**.

To assign a coil terminal excitation and add it to this winding:

1. Select the section of the geometry on which you want to apply the coil terminal excitation.
2. In the **Project Manager** tree, right-click the winding, and select **Assign Coil**. The **Coil Excitation** dialog box appears.
3. Enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the number of conductors for the coil in the **Number of Conductors** box.

> **Note:** This value represents only the number of conductors inside of the selected geometry. If the Coil Terminal is cut due to symmetry, then only enter the Number of Conductors in the portion modeled.

5. Select one of the following for the **Polarity**:
   - **Positive**
   - **Negative**
   - **Function**
6. If you selected **Function** as the **Polarity**, enter a function in the text box.
7. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
8. Click **OK** to assign the coil terminal excitation. The coil terminal excitation is assigned and is added to the winding.

To delete all coil terminal excitations that belong to this winding:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- In the **Project Manager** tree, right-click the winding, and select **Delete All Coils**. All coil terminal excitations are removed from the winding and deleted from the excitations.

---

1. For a solid winding, the resistance term can represent: the resistance of a portion of the winding which is not modeled (for instance, end-effects), the leads connecting the winding to the source, or the source resistance. (The main winding resistance is calculated directly by the solver.) For a stranded winding, the resistance term is the complete DC resistance of the winding (since the solver does not determine resistance of a stranded winding) as well as the resistance of the end-effects, leads, source, etc. For both solid and stranded windings, the inductance term can represent: the extra inductance for a portion of the winding of the winding which is not modeled (for instance, end-effects), the leads connecting the winding to the source, or the source inductance. (The main winding inductance itself is calculated directly by the solver.)

External Circuit Setup

The Maxwell Circuit Design Editor is used to set up the external circuit excitation for windings in Maxwell 2D and 3D AC Magnetic solutions. Because the AC Magnetic solver is a frequency domain solver, only linear circuit components are supported, which include Resistance, Inductance, Capacitance, and Transformer. In addition, only sinusoidal sources should be used as sources. Sinusoidal Current Source and Sinusoidal Voltage Source, designed for transient solutions, can also be used for AC Magnetic solutions. For example, to use the sinusoidal current source, you should set the parameters as follows:

- Set **I0**, **Td**, and **Df** to zero, where I0 is offset current in Amps, Td is the delay time in seconds, and Df is the damping factor in 1/seconds.
- Set **TIME** as the **Type**.
- Use **Ia** to set the peak amplitude in Amps.
- Use **IFreq** to set the signal frequency. Note that setting IFreq is not necessary as it will be internally set to the frequency defined in the Maxwell design setup.
- Use **Phase** to set the phase delay.

With these specifications, the time signal can be written as

$$I(t) = I_a \sin\left(2\pi I_{freq} - Phase\right)$$

Note that “Phase” is the signal phase delay. The complex source in the AC Magnetic solver is represented in phasor form as

$$I = I_a \; e^{j\theta}$$

and the time signal can be written in terms of (I_a, θ) as

$$I\left(t\right) = Im\left\{I_a \; e^{j(2\pi I_{freq} + \theta)}\right\}$$

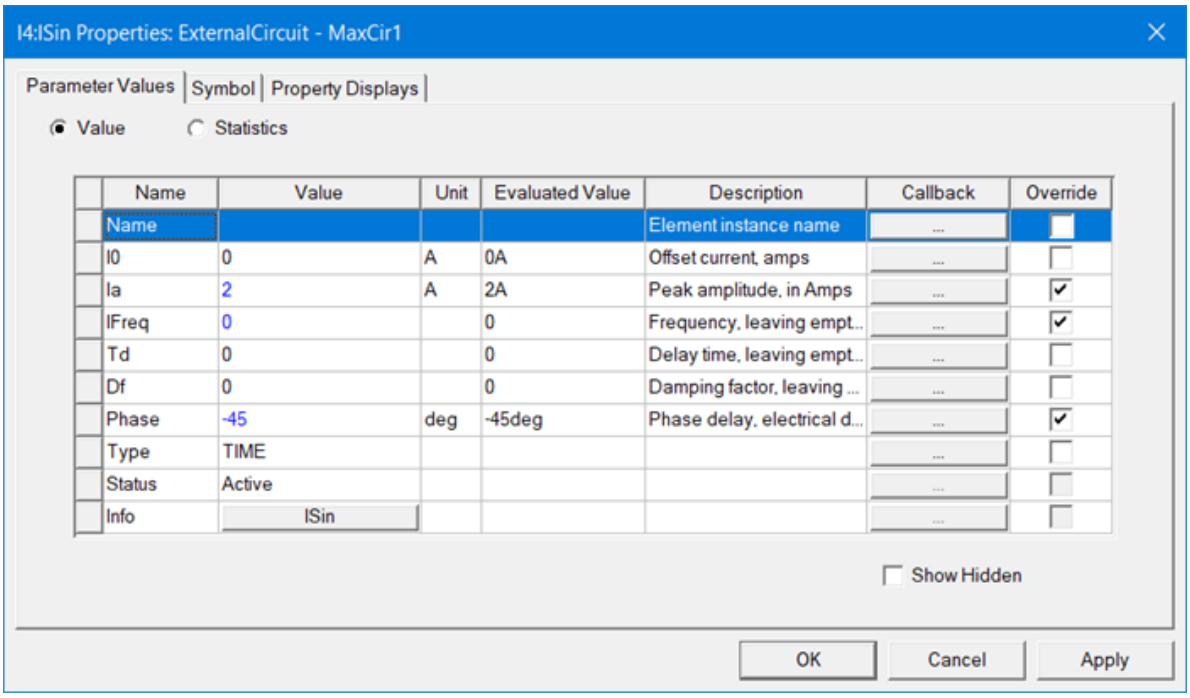
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the complex expression, θ is defined as phase, while in Maxwell Circuit, *Phase* is defined as phase delay. Thus,

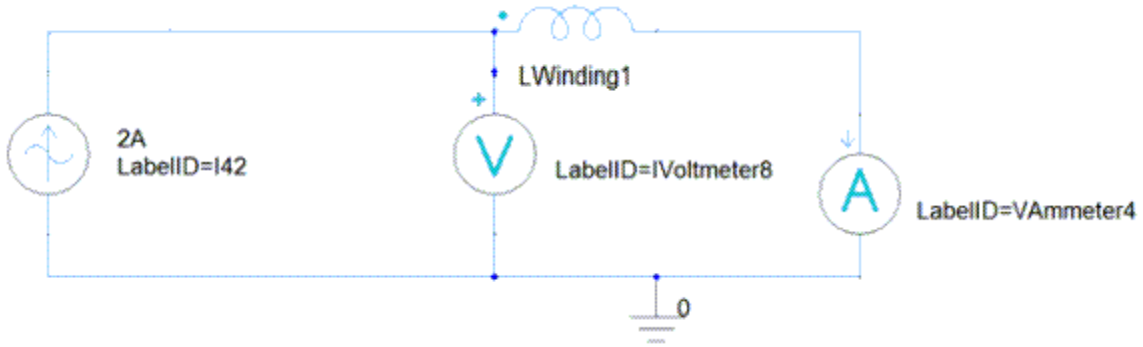
$$Phase = -\theta$$

As a result, you should enter the negative of phase of the complex signal as the Phase value.

An example of such a setup for $I = 2e^{j45^\circ}$ is shown below.



An example of an external circuit excitation for the winding for an AC Magnetic solver is shown below.



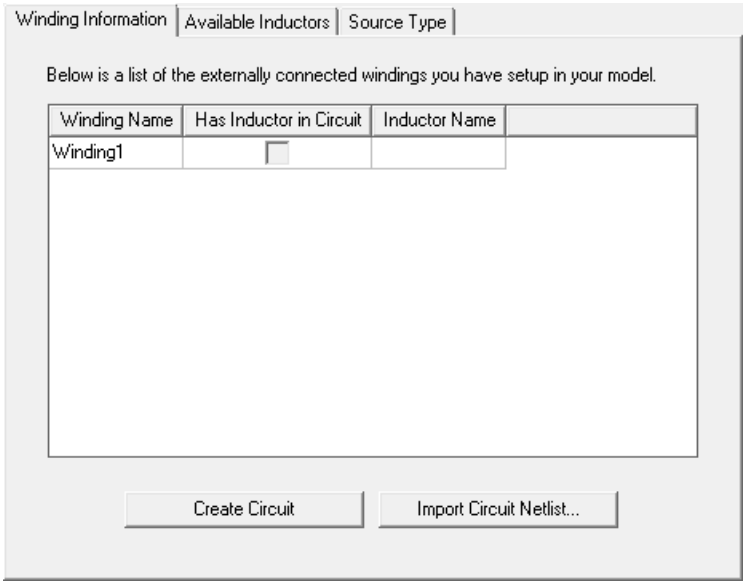
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** For AC Magnetic solutions, only linear circuit components are supported, which include Resistor, Inductor, Capacitor, and Transformer. Only sinusoidal sources should be used as sources.

To set up an external circuit connection for an External winding type:

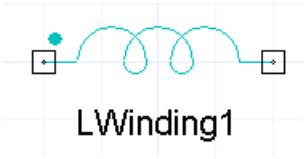
1. Click **Maxwell 2D** > **Excitations** > **External Circuit** > **Edit External Circuit** to open the **Edit External Circuit** dialog box. You can also right-click on **Excitations** in the Project Manager tree and select **External Circuit** > **Edit External Circuit**.

   a. If no external winding connections have been set up for your model, the **Winding Information** tab lists the external winding(s) in the design. Continue with step 2.



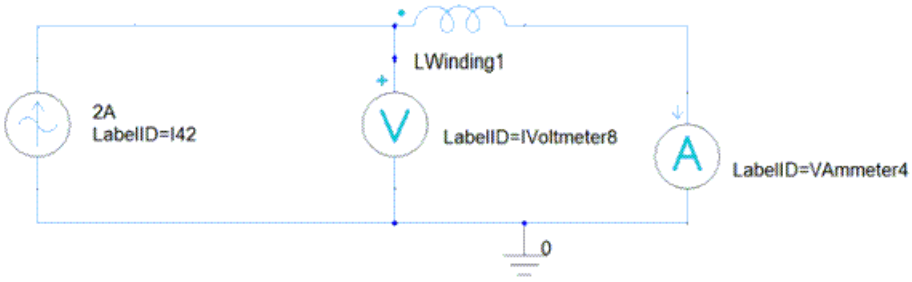
   b. If external windings have already been set up, continue with step 7.

2. On the **Winding Information** tab, click **Create Circuit** to add a circuit design to the project. The new circuit design automatically opens for editing in the Maxwell Circuit Editor. The circuit includes one inductor for each external winding in the original Maxwell transient design. The following example is for a design having one external winding.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Add the components needed to complete the external circuit design such as shown the following example.



4. When the circuit design is finished, click **Maxwell Circuit** > **Export Netlist** and export the netlist **.sph** file to the desired location.
5. Return to the Maxwell design and click **Maxwell 2D** > **Excitations** > **External Circuit** > **Edit External Circuit** to open the **Edit External Circuit** dialog box. You can also right-click on **Excitations** in the Project tree and select **External Circuit** > **Edit External Circuit**.
6. On the **Winding Information** tab, click **Import Circuit** to import the circuit.
7. To view a list of inductors in the imported circuit, click the **Available Inductors** tab.
8. To view the sources used in the externally connected windings, click the **Source Type** tab.
9. If variables are used in the imported circuit, they and their design values are listed on the **Parameter Values** tab. You can link to variables within the imported circuit as follows:
    a. On the **Parameter Values** tab, click in the **Value** column of the parameter to be mapped.
    b. Enter a new local variable name and press the `Enter` key to open the **Add Variable** dialog box.
    c. Select a **Unit Type** and **Unit**, and enter a **Value** for the variable and click **OK**. The new Maxwell variable is now mapped to the variable in the imported circuit, and can be varied directly by Maxwell or used for Optimetrics analyses.
10. If the imported file is a .**sph** file, you can click the **Circuit Path** tab to view the original project and design names.

> **Note:** After you import a circuit netlist, the **Create Circuit** button changes to **Update Netlist from Circuit**. Use **Update Netlist from Circuit** to reimport the netlist to keep it current.

11. Click **OK** to close the **Edit External Circuit** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2D Transient Boundaries and Excitations

Define at least one of the following as a source of magnetic fields:

- The stranded or solid windings, with a voltage or current supply or connected to an external circuit.
- Permanent magnet(s).

You will need to set at least one outer boundary to the following:

- Value (vector potential) boundary conditions.
- An odd symmetry (flux tangential) boundary.
- An even symmetry (flux normal) boundary.

2D Transient Boundaries

The transient field solver allows you to define the same types of boundary conditions as magnetostatic ones.

Boundary Type	H-Field Behavior	Used to model...
Default Boundary Conditions (Natural and Neumann)	Field behaves as follows: • Natural boundaries — Tangential **H** and normal **B** are continuous across surfaces without current density distribution. Tangential **H** has a jump if the surface has current density distribution. • Neumann boundaries — Magnetic field is normal to the boundary.	Initially, object interfaces have natural boundaries; outer boundaries, and excluded objects have Neumann boundaries.
Vector Potential	Sets the magnetic vector potential A_Z, or $rA\phi$, on the boundary. The behavior of H depends on whether A_Z or $rA\phi$ is constant or functional.	Outer boundaries at specific vector potentials; externally applied magnetic fields.
Symmetry	Field behaves as follows: • Odd Symmetry (Flux Tangential) — **Magnetic Field** is tangential to the boundary; its normal components are zero. • Even Symmetry (Flux Normal) — **Magnetic Field** is normal to the boundary; its tangential components are zero.	Planes of geometric and magnetic symmetry.
Balloon	Models the case where the structure is	Magnetically isolated

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	“infinitely” far away from other magnetic fields, permanent magnets, or current sources.	structures.
Matching (Independent and Dependent)	The **Magnetic Field** on the dependent boundary is forced to match the magnitude and direction (or the negative of the direction) of the **Magnetic Field** on the independent boundary.	Planes of symmetry in periodic structures where **Magnetic Field** is oblique to the boundary.

Default Boundary Conditions

These boundary conditions are automatically defined for a 2D transient model:

- Natural boundaries are assigned to the surfaces between objects. Tangential **H** and normal **B** are continuous across surfaces without current density distribution. Tangential **H** has a jump if the surface has current density distribution.
- Neumann boundaries are assigned to the outside edges of the problem region. Magnetic field is normal to the boundary.

To leave a surface set to its default boundary condition, you do not need to set any boundary conditions. Deleted boundary conditions and excitations automatically reset to the default boundary conditions.

Assigning a Vector Potential

To define a vector potential boundary for a 2D Transient solver:

1. Select the section of the geometry on which you want to apply the boundary condition (typically an edge). Maxwell 2D requires that the **Vector Potential** be assigned on a 1D object such as an edge or a line.
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Vector Potential**.
3. In the **Vector Potential Boundary** dialog box that appears, enter a name for the boundary in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the value of the potential. The value may be specified as a single numeric value or a spatial function.
5. If a functional value is specified, select a **Coordinate System** from the drop-down menu:
6. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
7. Click **OK** to assign the boundary to the selected object.

The new boundary is added to the boundary list in the Project Manager tree.

Assigning a Symmetry Boundary

This boundary condition defines a plane of geometric or magnetic symmetry in a structure. Assign it only to the outer surfaces of the problem region.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To set a symmetry boundary for the 2D Transient solver:

1. Select the section of the geometry on which you want to apply the boundary condition (typically an edge).
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Symmetry**.
3. In the **Symmetry Boundary** dialog box that appears, enter a name for the boundary in the **Name** box, or accept the default.
4. Select one of the following as the type of symmetry:

Odd (Flux Tangential)	**Magnetic Field** is tangential to the boundary; its normal components are zero.
Even (Flux Normal)	**Magnetic Field** is normal to the boundary; its tangential components are zero.

5. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
6. Click **OK** to assign the boundary to the selected object.

Assigning a Balloon Boundary Condition

To set a balloon boundary for the 2D Transient solver:

1. Select the section of the geometry on which you want to apply the boundary condition (typically an edge).
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Balloon**.
3. In the **Symmetry Boundary** dialog box that appears, enter a name for the boundary in the **Name** box, or accept the default.
4. Click **OK** to assign the boundary to the selected object.

Assigning an Independent Boundary

Independent and dependent boundaries enable you to model planes of periodicity where the **Magnetic Field** at every point on the dependent boundary surface is forced to match the **Magnetic Field** of every corresponding point on the independent boundary surface.

To set an independent boundary for the 2D Transient solver:

1. Select the edge to which you want to assign the independent boundary.
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Matching** > **Independent**.
3. In the **Independent Boundary** dialog box that appears, enter a name for the boundary in the **Name** box, or accept the default.
4. To reverse the direction of the vector, select the **Reverse Direction** check box.
5. Click **OK**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Maxwell solver computes the **Magnetic Field** on this boundary and maps it to the dependent boundary using the transformation defined by the independent and dependent coordinate systems.

Assigning a Dependent Boundary

Assigning a dependent boundary is the second step in creating matching boundaries. The field on the independent boundary is mapped to the dependent boundary.

> **Note:** You must define an independent boundary before creating the dependent boundaries that are associated with it. Assign dependent boundaries only to the outer surfaces of the problem region.

To set a dependent boundary for the 2D Transient solver:

1. Select the section of the geometry on which you want to apply the boundary condition (typically an edge).
2. Click **Maxwell 2D** > **Boundaries** > **Assign** > **Matching** > **Dependent**.
3. In the **Dependent Boundary** dialog box that appears, enter a name for the boundary in the **Name** box, or accept the default.
4. Select the independent boundary to which the dependent is to be assigned from the **Independent** drop-down menu.
5. In the **Relation** section, select one of the following to specify the field behavior on the boundary:

Bdep = Bind	Click this radio button if the magnetic field on the dependent and independent boundaries have the same magnitude and direction.
Bdep = -Bind	Click this radio button if the magnetic field on the dependent boundary has the same magnitude as but the opposite direction from the independent boundary field.

6. To reverse the direction of the vector, select the **Reverse Direction** check box.
7. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
8. Click **OK** to assign the boundary to the selected object.

2D Transient Excitations

Transient problems use the following types of excitations:

Excitation	Type of Excitation

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Current	The total current in a conductor
Current Density	The known current density distribution in a conductor
Coil	Used to define one or more model windings
End Connection	End connection is used to model details of the "squirrel cage" winding of induction machines, dampers in synchronous machines, etc.
Winding With Current	Current for both a stranded and solid conductor
Winding With Voltage	Voltage for both a stranded and solid conductor
Winding With External Circuit Connection	External circuit connection for both a stranded and solid conductor

In addition, permanent magnet materials serve as sources of magnetic fields.

> **Note:** A winding is a versatile type of excitation. Six different combinations can be used: Current, Voltage, and External Circuit — each with a solid or stranded conductor. The Coil Terminal type of excitation needs to be defined prior to setting up a winding.

Assigning a Current Excitation

For the 2D Transient solver, this excitation specifies the total transient current in an object.

To specify the total transient current:

1. Select the 2D section of the geometry on which you want to apply the excitation.
2. Click **Maxwell 2D** > **Excitations** > **Assign** > **Current**.
3. In the **Current Excitation** dialog box that appears, enter a name for the excitation in the **Name** box, or accept the default.
4. Enter the current magnitude, time dependent, in the **Value** box, and select a unit from the drop-down menu.
5. Select the **Solid** or **Stranded** radio button to specify the type of conductor.
6. Select either **Positive** or **Negative** as the **Ref. Direction** for the conductor.
7. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
8. Click **OK** to assign the excitation to the selected object.

Assigning a Current Density Excitation

For the 2D Transient solver, this command specifies the applicable component (depending on the XY or RZ type of application) of the transient current density in an object. If the current

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

density is a function of position, the value is entered in ampere/m^2, even if you change the units in the problem.

To define the current density:

1. Select the section of the geometry (*i.e.*, the conductor) in which you want to apply the excitation (typically a 2D object).
2. Click **Maxwell 2D** > **Excitations** > **Assign** > **Current Density**.
3. In the **Current Density Excitation** dialog box that appears, enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, specify the following options:
    a. Enter the current strength in the **Value** box. The value may be a single numeric value or a spatial function.
    b. If a spatial function is entered for the **Value**, select the **Coordinate System** from the drop-down menu.
5. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
6. Click **OK** to assign the excitation to the selected object.

Assigning a Coil

To assign a coil as a 2D transient excitation:

1. Select the section of the geometry on which you want to apply the excitation (typically a 2D object).
2. Click **Maxwell 2D** > **Excitations** > **Assign** > **Coil**. (You can also right-click an existing winding in the project tree, and select **Assign Coil**.)
3. In the **Coil Excitation** dialog box that appears, enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the number of conductors for the coil in the **Number of Conductors** box.
5. Select one of the following for the **Polarity**:
    - **Positive**
    - **Negative**
    - **Function**
6. If you selected **Function** as the **Polarity**, enter a function in the text box.
7. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
8. Click **OK** to assign the coil to the selected object.

To add this coil to an existing winding:

1. In the project tree, right-click the coil terminal, and select **Add to Winding**.
2. In the **Add to Winding** dialog box that appears, select the winding to which you want to add the coil terminal, and click **OK**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To remove a coil terminal from an existing winding, do one of the following:

- In the Project Manager tree, right-click the coil terminal, and select **Remove from Winding**.
  The terminal is removed from the winding and moved up one level in the Project Manager tree (directly beneath **Excitations**).
- Alternatively, all coils assigned to a winding may be deleted by right-clicking the winding in the Project Manager tree and selecting **Delete All Coils**.

Assigning an End Connection

The End Connection is used to model details of the "squirrel cage" winding of induction machines, dampers in synchronous machines, etc.

To define an end connection for the 2D Transient solver:

1. Select the section of the geometry (i.e., the conductor) across which you want to apply the excitation.
2. Click **Maxwell 2D** > **Excitations** > **Assign** > **End Connection**.
3. In the **End Connection Excitation** dialog box that appears, enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, specify the following options:
   a. Enter the **End resistance between adjacent conductors**, and select a unit of measurement from the drop-down menu.
   b. Enter the **End inductance between adjacent conductors**, and select a unit of measurement from the drop-down menu.
5. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
6. Click **OK** to assign the excitation to the selected object.

Assigning a Winding Setup

> **Note:**
>
> - If you wish to define a winding for a 2D AC Magnetic solution type, refer to Assigning a Winding Setup for a 2D AC Magnetic Solver.
> - If you wish to define a winding for a 2D Magnetostatic solution type, refer to Assigning a Winding Setup for a 2D Magnetostatic Solver.

You can use the predefined coil terminal(s) to define one or more current or voltage windings.

To define a winding for the 2D Transient solver:

1. Click **Maxwell 2D** > **Excitations** > **Add Winding** to open the **Winding** dialog box.
2. Enter a name for the winding in the **Name** box, or accept the default.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In the **Parameters** section, select **Current**, **Voltage**, or **External** from the **Type** drop-down menu.

   > **Note:** Maxwell transient designs can be dynamically coupled to Twin Builder components through the Twin Builder user interface. You can enable this feature on the Advanced Product Coupling tab of the **Design Settings** dialog box. Source windings set to **External** are available as conservative pins in Twin Builder. For more information, refer to the Help for Twin Builder.

4. Select the **Solid** or **Stranded** radio button to specify the type of conductor.
5. Enter values in the following fields, and select the desired units:

For a current winding	For a voltage winding[1]	For an external winding
Current	Initial Current	Initial Current
	Resistance	
	Inductance	
	Voltage	

   > **Note:** You can also type a function as an expression for any of these items except initial current.

6. Enter a value in the **Number of parallel branches** text box.

   > **Note:** Parallel branches for solid windings assume there are no circulating currents in the parallel branches.

7. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
8. Click **OK** to assign the excitation to the selected object

The definition of a winding is final only after specifying the coils which belong to it. **To add a coil terminal to this winding**:

1. In the Project Manager tree, right-click the winding, and select **Add Coils**.

   The **Add Terminals** dialog box appears, listing all coil terminals that do not already belong to that winding.

2. Select the coil terminal (s) you want to add. To select multiple terminals, press **Ctrl** and click each terminal.
3. Click **OK**.

To assign a coil terminal excitation and add it to this winding:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the section of the geometry on which you want to apply the coil terminal excitation.
2. In the Project Manager tree, right-click the winding, and select **Assign Coil**.
3. In the **Coil Excitation** dialog box that appears, enter a name for the excitation in the **Name** box, or accept the default.
4. In the **Parameters** section, enter the number of conductors for the coil in the **Number of Conductors** box.

   > **Note:** This value represents only the number of conductors inside of the selected geometry. If the Coil Terminal is cut due to symmetry, then only enter the Number of Conductors in the portion modeled.

5. Select one of the following for the **Polarity**:
   - **Positive**
   - **Negative**
   - **Function**
6. If you selected **Function** as the **Polarity**, enter a function in the text box.
7. Optionally, click **Use Defaults** to revert to the default values in the dialog box.
8. Click **OK** to assign the coil terminal excitation.
   The coil terminal excitation is assigned and is added to the winding.

To delete all coil terminal excitations that belong to this winding: In the Project Manager tree, right-click the winding, and select **Delete All Coils**. All coil terminal excitations are removed from the winding and deleted from the excitations.

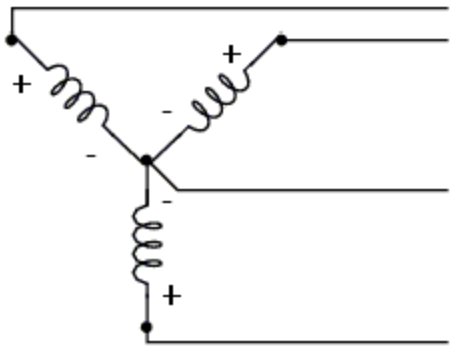
---

1. For a solid winding, the resistance term can represent: the resistance of a portion of the winding which is not modeled (for instance, end-effects), the leads connecting the winding to the source, or the source resistance. (The main winding resistance is calculated directly by the solver.) For a stranded winding, the resistance term is the complete DC resistance of the winding (since the solver does not determine resistance of a stranded winding) as well as the resistance of the end-effects, leads, source, etc. For both solid and stranded windings, the inductance term can represent: the extra inductance for a portion of the winding of the winding which is not modeled (for instance, end-effects), the leads connecting the winding to the source, or the source inductance. (The main winding inductance itself is calculated directly by the solver.)

Setting Up a Y Connection

The Y Connection function available in 2D and 3D Transient solution types allows multiple windings to be connected in a classical Y (sometimes referred to as wye) configuration with the negative terminals connected to a common node as illustrated below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Setup Y Connection requires voltage winding definitions be set and the negative voltage terminal for each winding will be connected to form the common node of the circuit.

Each voltage source specified in the winding description drives the positive side of the coil. Y Connections are commonly used in motor and transformer applications driven by three-phase voltage; however, any number of windings may be connected to a common node in Maxwell.

Note:

- Only one Y Connection is allowed in a given Maxwell design.
- For 3D designs, only stranded voltage windings in a Y-connection are supported.
- For 2D transient designs, both stranded and solid windings are supported, but a mix of stranded winding and solid winding in one Y-connection group is not allowed.
- A mix of PWM voltage winding and (normal) voltage winding in one Y-connection group is not allowed.
- Duplicate windings are not allowed.

To define a Y-connection:

1. Click **Maxwell 2D** (or **Maxwell 3D**) > **Excitations** > **Setup Y Connection**.

   The **Setup Y Connection** dialog box appears.
2. In the **Winding** list on the left, select the windings to be grouped into a Y connection.
3. Click the **Group** button to create a Y Connection group with the selected windings.
4. Click **OK**.

Setting Up an SVPWM Excitation

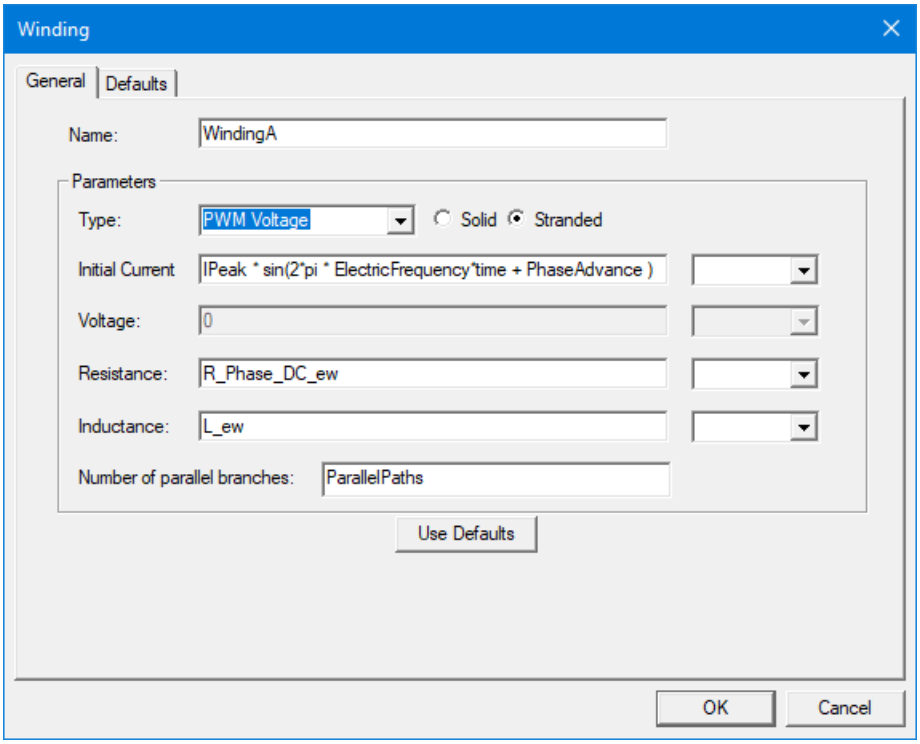
Pulse Width Modulation (PWM) is a term used for any method that produces a square wave signal with varying duty cycle and frequency to control the average power delivered to a load.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PWM is widely used in applications such as light dimming, switching power supplies, and electric motor drive. In these types of designs, the AC motor needs to be powered by an alternating current excitation, which is supplied by the DC supply voltage through an inverter circuit using power device switches. Controlling how the power switches are turned on and off is accomplished by PWM.

For 3D and 2D magnetic transient T-Omega solutions, Space Vector Pulse Width Modulation (SVPWM) can be directly created in Maxwell for three-phase voltage-excited windings. For all three phases, Maxwell will calculate the PWM voltage waveforms based on user inputs and automatically set up the winding excitations in the transient simulations. The generated waveform represents the Phase-Neutral voltage, and it is applicable to Y(star)- or delta-connected winding.

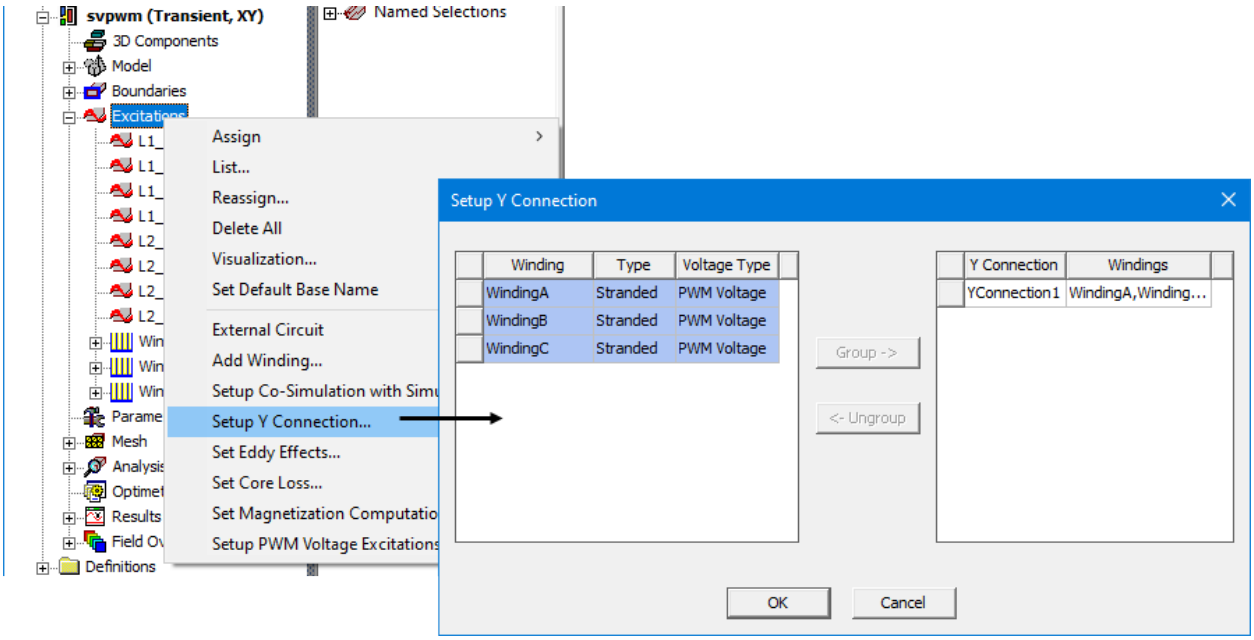
1. Before setting up the SVPWM excitation:
    a. Define three PWM voltage windings. For the **Type** option in the Winding window, make sure to select **PWM Voltage**:



    Note that when the **PWM Voltage** type is selected, the Voltage field is grayed out because it will use the data generated by PWM voltage excitation setup.

    b. If you will be using a Y connection for the winding, right click on the **Excitations** entry in the Project Manager, and select **Setup Y Connection**. Select the three windings, and click **Group**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



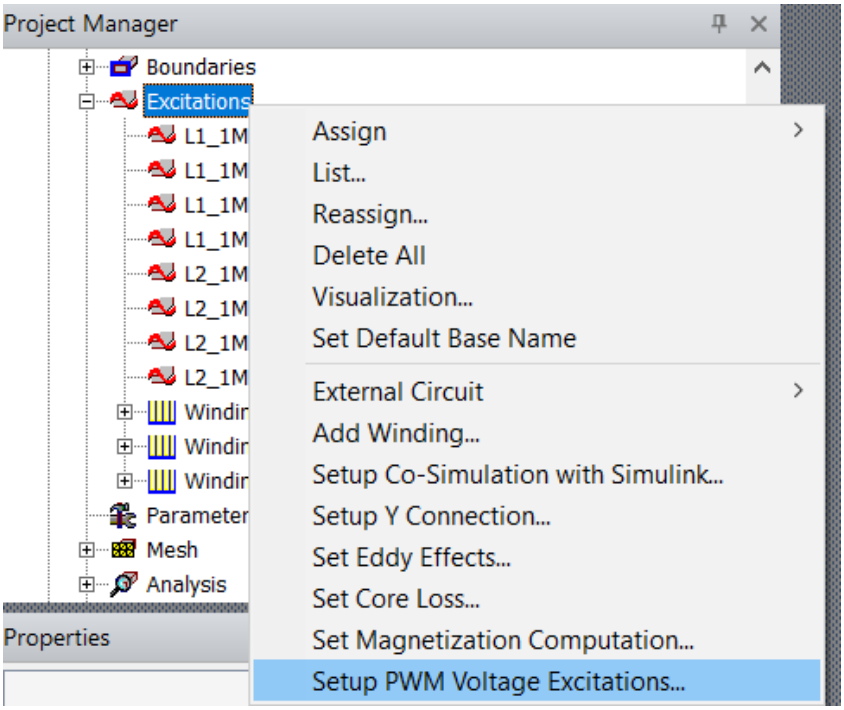
Note:

- For 3D transient designs, only stranded windings are allowed for the Y-connection.
- For 2D transient designs, both stranded and solid windings are supported, but a mix of stranded winding and solid winding in one Y-connection group is not allowed.
- A mix of PWM voltage winding and (normal) voltage winding in one Y-connection group is not allowed.
- Only one Y Connection is allowed in a given Maxwell design.

2. In the Project Manager tree, right click **Excitations**, and select **Setup PWM Voltage Excitations**.

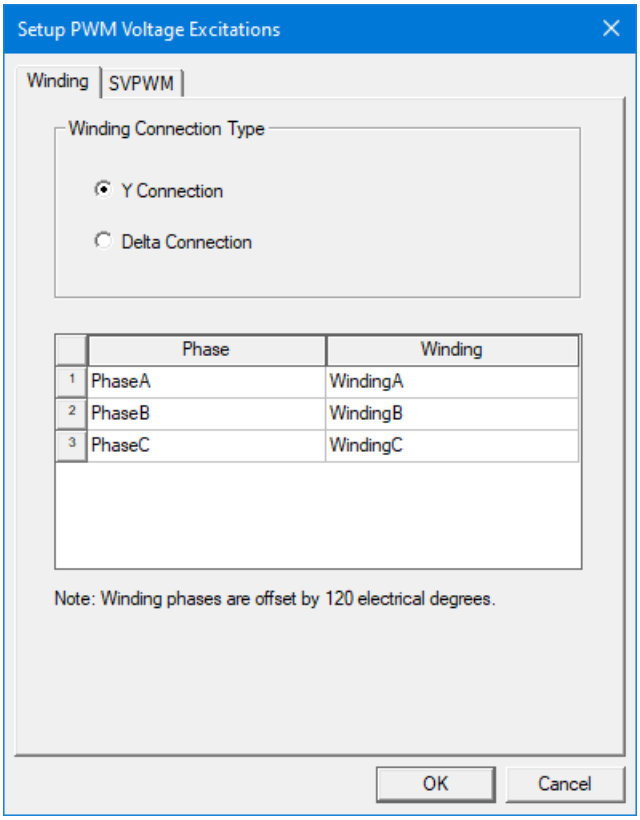
   Alternatively, from the menu bar, select **Maxwell 2D/3D** > **Excitations** > **Setup PWM Voltage Excitations**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. In the Setup PWM Voltage Excitations window that opens, select either the **Y Connection** or the **Delta Connection**, and arrange the three windings to represent the three phase windings (PhaseA, PhaseB and PhaseC) accordingly. Note that the **Winding** drop-down menu will only include windings that were defined as the **PWM Voltage** type.

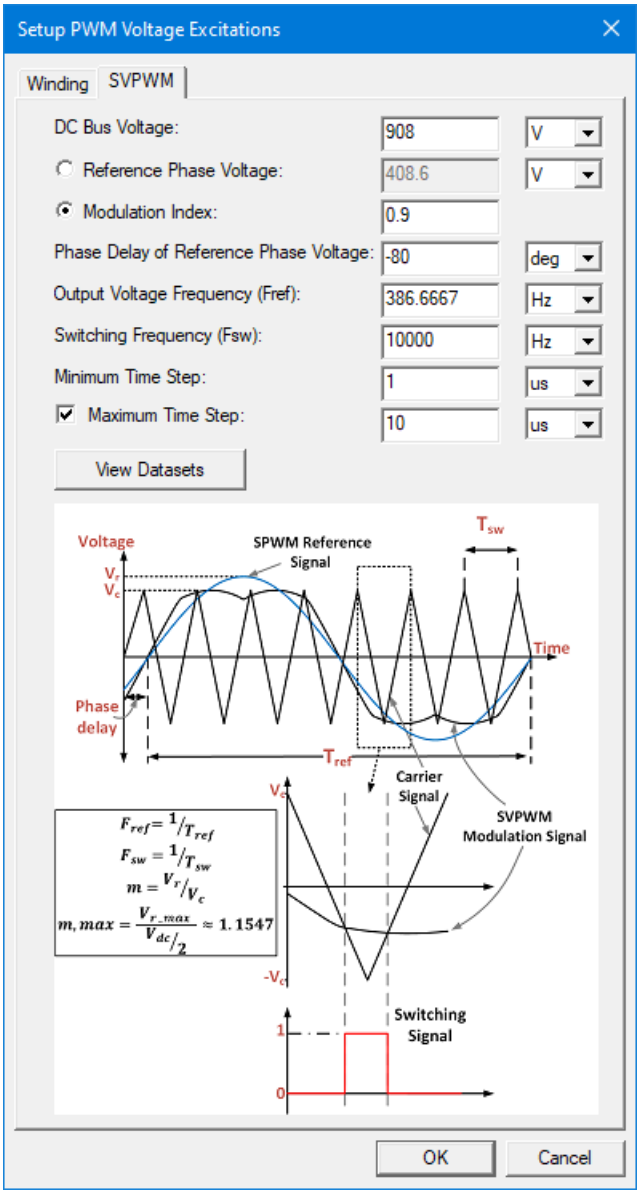
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:** For Delta Connection winding, the PhaseA, PhaseB, and PhaseC voltages are different than for the Y connection winding: if voltage values for Y connection are Va, Vb, Vc, then the voltage values for delta connection are Va-Vb, Vb-Vc, Vc-Va.

4. Click on the **SVPWM** tab, and input the corresponding parameters for SVPWM. Example parameters are shown below:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note: All the parameters of the Setup PWM Voltage Excitations window can be defined with a variable **except** Modulation Index.

- **DC Bus Voltage** – Specifies the voltage available at the main DC bus as input to the inverter.
    - **Reference Phase Voltage** – Defines the reference voltage signal, seen as the blue curve (labeled SVPWM Reference Signal) in the diagram in the Setup PWM Voltage Excitations window. If this option is selected, the **Modulation Index** setting is grayed out and automatically calculated

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

according to ModulationIndex = ReferencePhaseVoltage/(DCBusVoltage/2).

- **Modulation Index** (m) – Specifies the ratio between the reference signal to the carrier signal; it determines the amplitude of the fundamental output voltage with respect to the available DC bus voltage as described by the equation below.

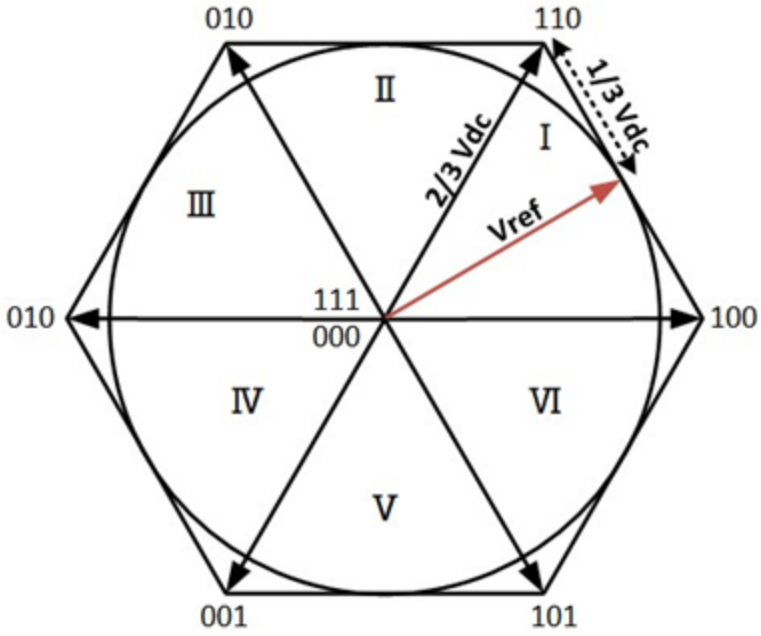
$$V_{peak,\ Phase-Neutral} = m * \frac{V_{DC}}{2}$$

The range of the modulation index is 0 < modulation index ≤ 2/sqrt(3) (*i.e.*, approx. 1.1547) for SVPWM.

> **Important:** For a small modulation index, care must be taken when setting the minimum time step to control numerical error.

If this option is selected, the **Reference Phase Voltage** setting is grayed out and automatically calculated according to ReferencePhaseVoltage = ModulationIndex*(DCBusVoltage/2).

To understand how the maximum modulation index is calculated, start with the figure below, which shows the space vector hexagon (represented by six sectors and eight vectors). Each vector represents a different switching state of the inverter.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The vector V_{ref} is the reference revolving voltage vector formed from the three phase sinusoidal voltages; at each sector, this vector is realized by alternating between the two nearest space vectors along with the zero vectors.

The circle inside the hexagon represents the maximum value possible for V_{ref} (and hence the output voltage) that can be produced with the SVPWM technique.

Knowing that the magnitude of each vector is 2/3 V_{DC}, the maximum value of the output voltage (V_{ref}) can be computed as

$$V_{RefMax} = \sqrt{\left(\frac{2}{3}V_{dc}\right)^2 - \left(\frac{1}{3}V_{dc}\right)^2}$$

$$V_{RefMax} = \frac{\sqrt{3}}{3}V_{DC}$$

From these definitions, the maximum modulation for SVPWM can be derived as

$$m_{Max} = \frac{V_{RefMax}}{\frac{V_{DC}}{2}} = \frac{2*\sqrt{3}}{3} \approx 1.1547$$

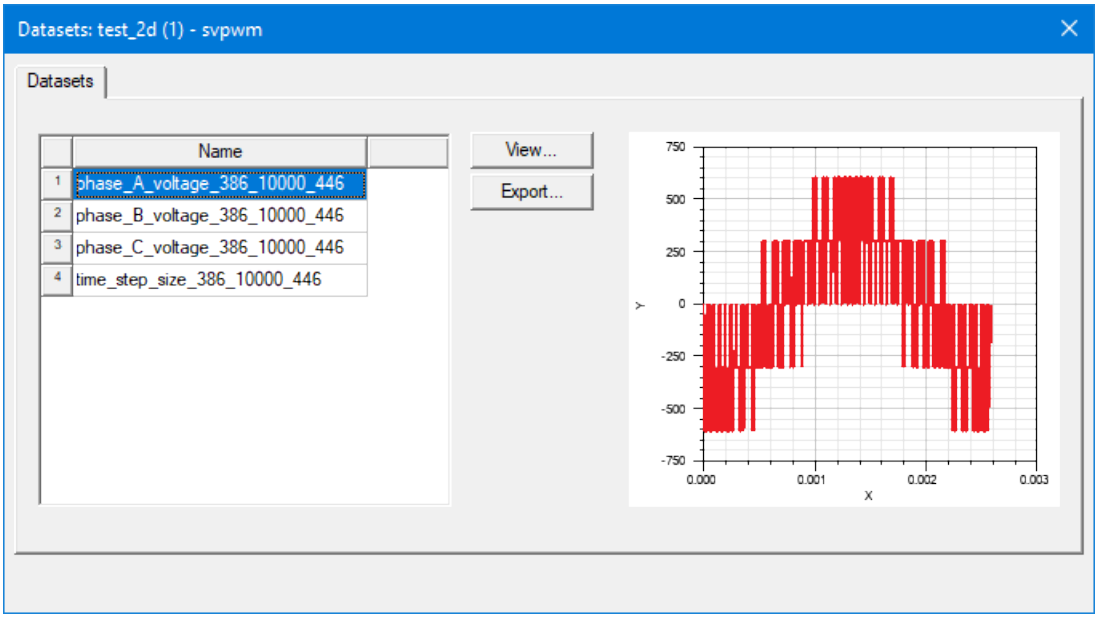
- **Output Voltage Frequency (Fref)** – Defines the fundamental frequency of the reference signal, and therefore, the PWM output voltage.
- **Switching Frequency (Fsw)** – Defines the carrier signal frequency that is used to generate the PWM switching signal. A typical value for traction inverters is between 10 – 20 kHz.
- **Phase Delay** – Defines the phase delay angle of the reference signal (and hence the output voltage). The phase delay depends on the operating point, and it can be provided from system-level simulation or by running a transient simulation with an ideal sinusoidal current and checking the phase delay on the output of the phase voltage.
- **Minimum Time Step** – Defines the rate at which the carrier signal and the reference signal are compared to generate the switching signals. This value is important to correctly capture the switching events. The recommended value is 1/(Fsw*100) or smaller.
- **Maximum Time Step –** Optional setting that specifies the largest time step that can be used to generate the switching signal; it is used to modify the time step dataset; see the explanation that follows. If left unchecked, there will be no maximum time step considered when the time step dataset is created. If checked, the default value is 10*Minimum Time Step.
- **View Datasets –** Click to view the new datasets generated by the PMW Voltage Excitation setup.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Three datasets of the voltage (Phase_X_voltage_<Fref>_<Fsw>) vs. time are generated based on the settings. The datasets are generated by comparing the reference signal to the triangular carrier at a sampling frequency defined by the minimum time step. Subsequently, the datasets are reduced by removing the timesteps where no change of the voltage values occurs.

The datasets represent the three normalized PWM phase voltages for one electrical cycle. The phase sequence of the generated dataset is A-B-C with Phase B and C lagging Phase A with 120 deg and 240 deg, respectively. If a variable was used to define a setting, the View Dataset function will only display nominal values.

A time step dataset is also created. Note that when creating the time step dataset, the solver considers the Maximum Time Step (*dt*), and it will add *N* more time steps (equal spaced) between the two time stamps Y(n-1) and Y(n) (say *deltaT* = Y(n) - Y(n-1)) when *deltaT* > *dt*, and *N* = floor(*deltaT*/*dt*). For example, with timeStep = [1.7, 2, 3.3] and *dt* = 0.4, then newTimeStep = [1.7, 2, 2.325, 2.65, 2.975, 3.3]. Three new time steps are added: 2.325, 2.65, and 2.975, and they are equally spaced between 2 and 3.3.



Note: The Export button can be used to export the datasets as design datasets.

5. Click **OK**.

The technique used by the Maxwell application to implement SVPWM excitation computes the pattern of voltage pulses for each of the three datasets by comparing a triangular carrier wave with frequency Fsw and amplitude 1 to three reference functions.

Unlike for sinusoidal PWM, where the reference functions are sinusoidal, for this SVPWM implementation, the three reference functions (V_{a_Ref}, V_{b_Ref}, V_{c_Ref}) need to be constructed as

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a first step. This is achieved by superposing the common mode voltage (V_{CM}) to the three sinusoidal signals (V_{a_0}, V_{b_0}, V_{c_0}) of a three-phase system:

$$V_{a_Ref} = V_{a_0} + V_{CM}$$

$$V_{b_Ref} = V_{b_0} + V_{CM}$$

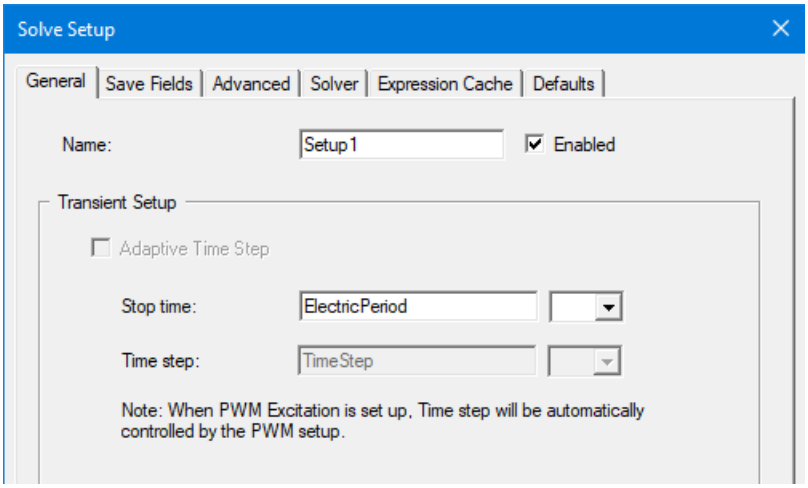
$$V_{c_Ref} = V_{c_0} + V_{CM}$$

V_{a_0} is defined as ModulationIndex*sin(2*pi*Fref*time-Phase Delay). V_{b_0} and V_{c_0} lag V_{a_0} by 120° and 240°, respectively.

The amplitude V_{max} can be calculated as (DC Bus Voltage)/2 * Modulation Index.

At every timestep, the common mode voltage is calculated as half of the mid-valued function among V_{a_0}, V_{b_0}, V_{c_0}.

The dataset time_step_size_<Fref>_<Fsw> tracks the exact instance the switching happens and will be used to define the variable time step in the transient setup to ensure that all switching events are captured with the minimum number of time steps possible. This dataset will be used in the Solve Setup Time step value field, so the **Time step** value field is grayed out. For a 2D analysis, the **Adaptive Time Step** check box is grayed out as well.



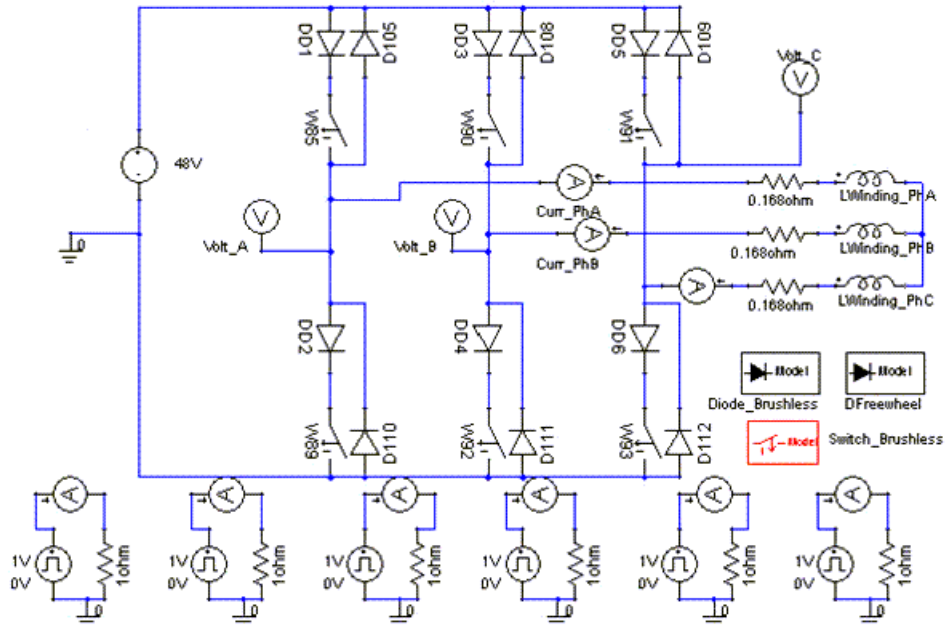
Setting Up an External Circuit

> **Note:** Use of external circuits is indicated when more complex excitation circuits such as rectifiers, inverters, etc., containing components such as diodes, transistors, complex sources, etc. are needed. There is no need to use an external circuit for simple current or voltage sources supplying the windings.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: External circuits intended for use with the Time Decomposition Method (TDM) can only support basic circuit elements that are solution-independent. This is because, for TDM, all timesteps (or all timesteps in a subdivision) are solved simultaneously - and thus the circuit topology and its parameters must be known before solving. Therefore some circuit elements, such as solution-dependent switches, diodes, capacitors, and inductors which are not directly connected to windings in series, are not supported.

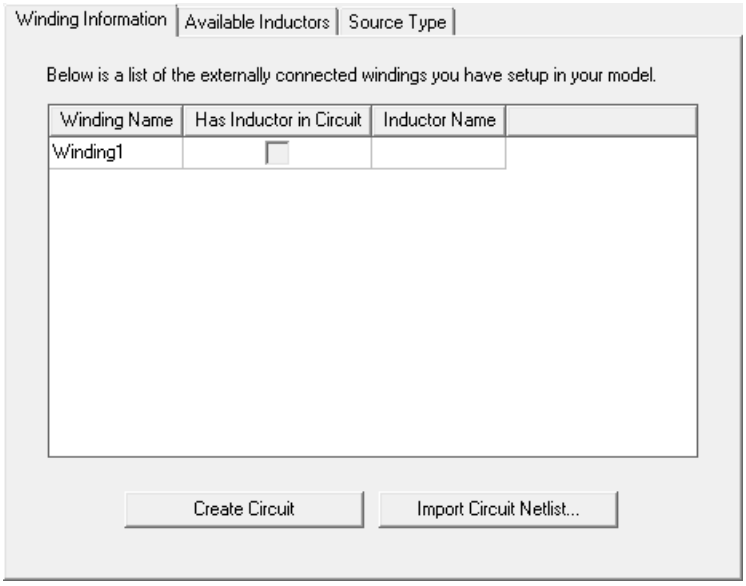
An external circuit can be complex as shown in the example below.



For the 2D Transient solver, to set up an external circuit connection for an External winding type:

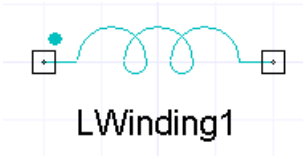
1. Click **Maxwell 2D** > **Excitations** > **External Circuit** > **Edit External Circuit** to open the **Edit External Circuit** dialog box. You can also right-click on **Excitations** in the Project tree and select **External Circuit** > **Edit External Circuit**.
    a. If no external winding connections have been set up for your model, the **Winding Information** tab lists the external winding(s) in the design. Continue with step 2.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

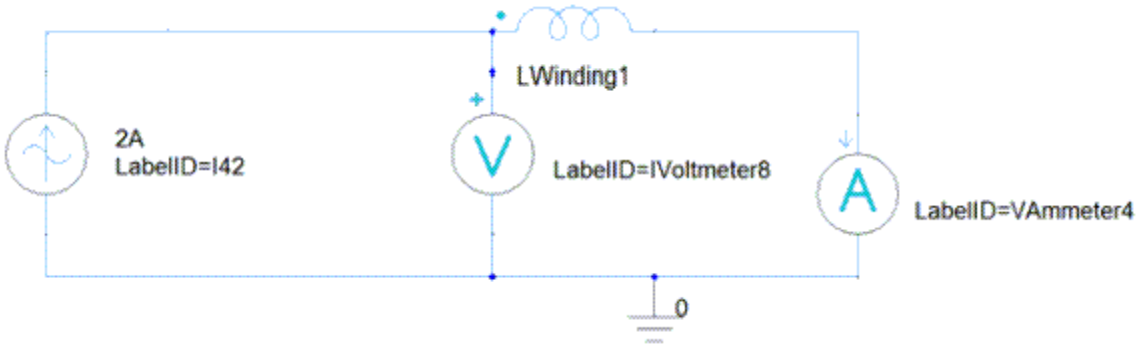


b. If external windings have already been set up, continue with step 7.

2. On the **Winding Information** tab, click **Create Circuit** to add a circuit design to the project. The new circuit design automatically opens for editing in the Maxwell Circuit Editor. The circuit includes one inductor for each external winding in the original Maxwell transient design. The following example is for a design having one external winding.



3. Add the components needed to complete the external circuit design such as shown the following example.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. When the circuit design is finished, click **Maxwell Circuit** > **Export Netlist** and export the netlist **.sph** file to the desired location.
5. Return to the Maxwell design and click **Maxwell 2D** > **Excitations** > **External Circuit** > **Edit External Circuit** to open the **Edit External Circuit** dialog box. You can also right-click on **Excitations** in the Project tree and select **External Circuit** > **Edit External Circuit**.
6. On the **Winding Information** tab, click **Import Circuit** to import the circuit.
7. To view a list of inductors in the imported circuit, click the **Available Inductors** tab.
8. To view the sources used in the externally connected windings, click the **Source Type** tab.
9. If variables are used in the imported circuit, they and their design values are listed on the **Parameter Values** tab. You can link to variables within the imported circuit as follows:
    a. On the **Parameter Values** tab, click in the **Value** column of the parameter to be mapped.
    b. Enter a new local variable name and press the `Enter` key to open the **Add Variable** dialog box.
    c. Select a **Unit Type** and **Unit**, and enter a **Value** for the variable and click **OK**. The new Maxwell variable is now mapped to the variable in the imported circuit, and can be varied directly by Maxwell or used for optimetrics analyses.
10. If the imported file is a .**sph** file, you can click the **Circuit Path** tab to view the original project and design names.

> **Note:** After you import a circuit netlist, the **Create Circuit** button changes to **Update Netlist from Circuit**. Use **Update Netlist from Circuit** to reimport the netlist to keep it current.

11. Click **OK** to close the **Edit External Circuit** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: The user-set timesteps can be modified when external circuits are used to drive the windings of the transient finite element model. Following is a list of situations that will lead to a time-step change:

- All power electronic switching instances that do not coincide with user-specified solve times request a new solution time from the transient solver. The respective switching generating a new solve time request for the transient solver can be time, position, or speed dependent.
- When current and/or voltage sources with a piecewise linear variation are used, a new solution time is requested from the transient solver at each (time) definition point used in the corresponding source definition table.
- When the change in any winding inductance value is excessive, a new timestep (smaller) is calculated and a new solution generated (recalculated) accordingly.

If a user-specified save field time is missed because of the above reasons, the next solved time-step fields are saved instead.

Setting Up an External Circuit Using Simulink

In addition to using the Maxwell Circuit Editor to set up an external circuit, you also can use MATLAB® Simulink® to set up an external circuit for both Maxwell 2D and 3D transient designs.

Supported Simulink versions are

- R2019b
- R2020a
- R2020b
- R2021a
- R2021b
- R2022a
- R2022b
- R2023a

Refer to Setting Up an External Circuit Using Simulink for detailed information.

Setting Up Motion for Transient Projects

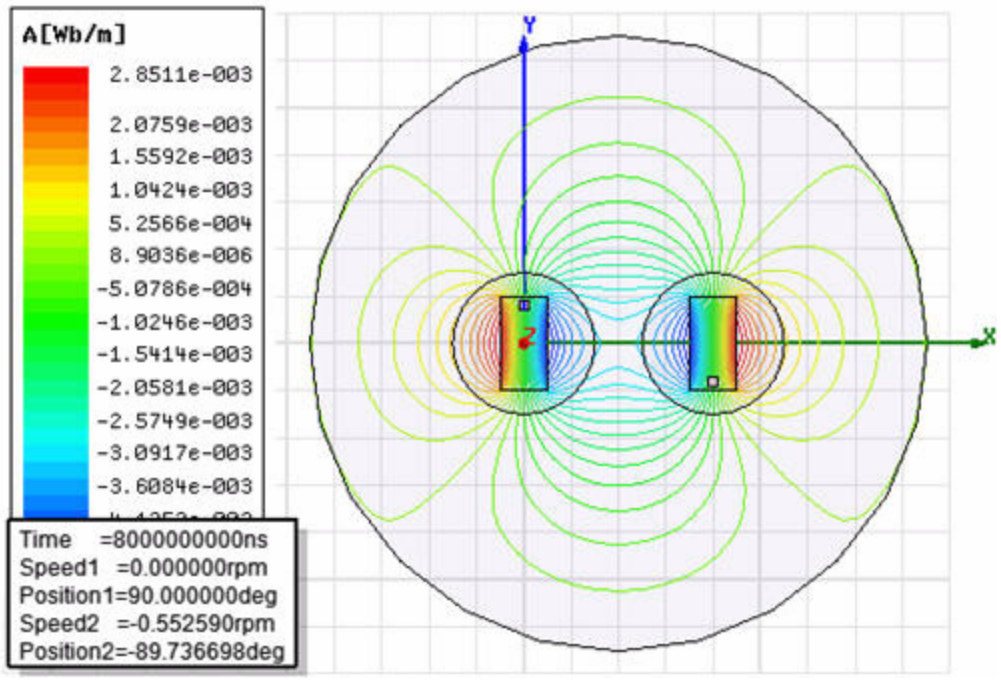
For projects where you selected transient as the solution type, you can define how the model components move in relation to one another. This is accomplished by assigning a **MotionSetup** to a **Band** object. Two or more motion bands may be assigned, each with its own independent motion characteristics. Multiple motion bands are used primarily for certain types of motors

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where multiple rotors may rotate about a common central axis; however, other applications are possible as shown in the following examples.

Example - Two Bands of Motion

The project shown below comprises two rectangular magnets, each surrounded by a band object with a MotionSetup assigned to it. When one magnet moves, the other magnet realigns itself. Also note that the solution context display shows the position and speed for both motion setups.

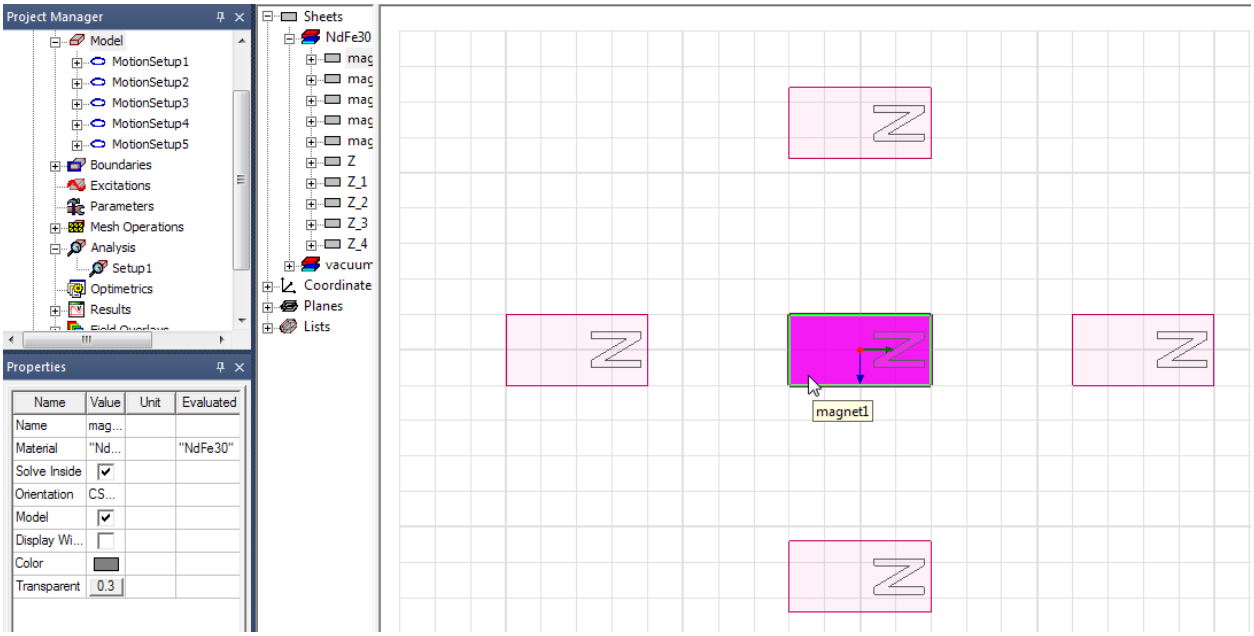


Note: The two moving bands must not touch each other. There must be some stationary region between the two bands, even if it is filled only with vacuum.

Example - Multiple Bands of Motion

The magnet project shown below comprises five bar magnets. When the center magnet (MotionSetup1) moves, the other magnets also move.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:** The moving bands may not touch each other. There must be some stationary region between the bands, even if it is filled only with vacuum.

Assigning a Band of Motion

You can assign a band of motion in both Maxwell 2D and Maxwell 3D projects. For both Maxwell 2D and 3D projects, two or more motion bands may be assigned.

> **Note:** Overlapping bands are not supported so they cannot be located inside of each other.

For both rotational and translational problems, the band object must always enclose all of the moving objects. For rotational problems, this means the band object should be a solid cylinder or a solid wedge – not a hollow shell or segment of a shell. If the band object encloses all moving aspects, it is easier to identify the moving objects and reduces errors related to misidentifying them.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: An inner region (or wrapper object) enclosed by the band can normally be used as a “fake” moving object for two reasons:

- The first is to avoid creating holes inside the band which would result in the band having multiple connected domains.
- The second is due to the torque/force computation being carried out by one layer of elements inside the band region that directly touches moving objects. If an inner region is not included, the torque/force computation will be very sensitive to mesh noise - thus leading to an inaccurate solution.

Note: For 2D and 3D translational motion, here are some additional recommendations:

- An all-encompassing inner region (non-magnetic, non-conductive wrapper such as air or vacuum) must be used around all moving objects in a band.
- The inner region wrapper object, like the Band itself, should be a simplified shape, such as a rectangle or polygon in 2D, or a polyhedral or segmented sweep section in 3D. Any simplifications to the inner region and band object will result in better re-meshing for each time-step and faster overall simulation times.
- In order to give more accurate forces, if there is sufficient clearance between moving and stationary objects, you can provide some padding between the inner region and the moving objects and the band. At a minimum, the inner region should be the same size as the outside of the enclosed objects (like shrink wrap) with no gap between the wrapper and objects inside.
- Ansys highly recommends that the band not touch any outer stationary objects, with the only exception being for symmetry boundaries on the outer region. It is suggested to locate the band halfway in the air-gap between fixed and moving objects or equally spaced with the inner region object if there is sufficient clearance at the gap.
- In order to get stable force results, length-based mesh operations (approximately equal to the distance between the band and the inner region) should be applied using the Length-Based-Inside mesh refinement assigned to the Band object so that the mesh length is refined in the band region that is re-meshed for each time-step.

To assign the band of motion:

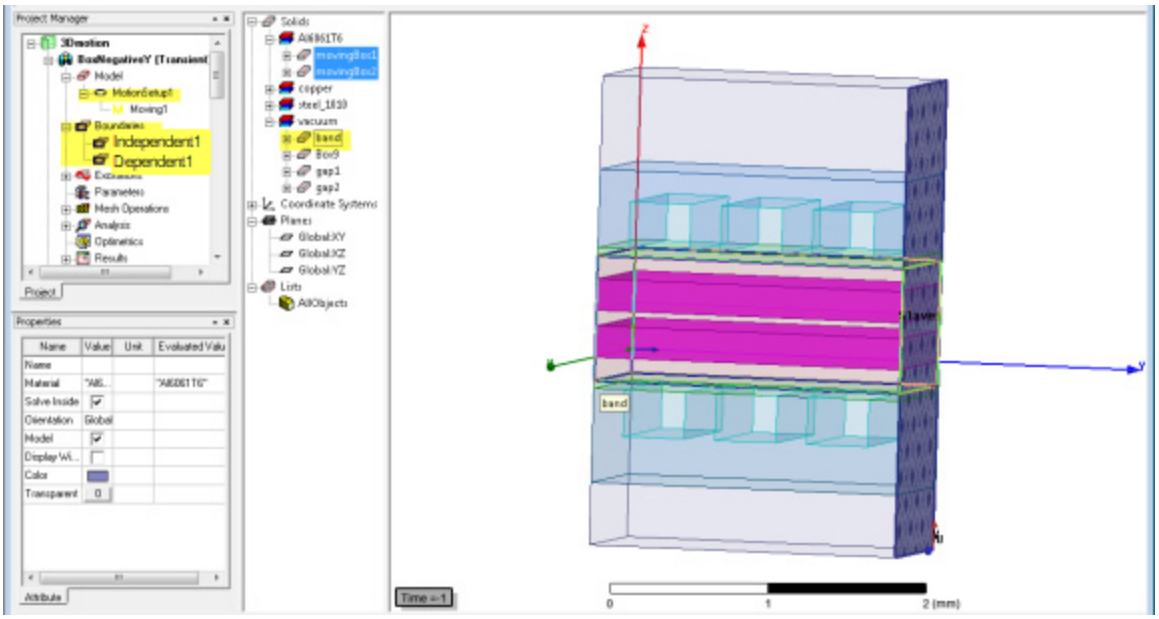
1. Select the band object.
2. Click **Maxwell** > **Model** > **Motion Setup** > **Assign Band**.

   The **Motion Setup** window appears.
3. On the **Type** tab: select either **Translation** or **Rotation** as the **Motion Type**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For **Rotation** about a fixed point you may select the **Non-Cylindrical** check box.
- For **Translation**, you may select the **Periodic** check box.

For **Periodic** motion, you must define independent and dependent boundary conditions. Both band and the object(s) inside it move, and both should touch the independent and dependent boundaries as shown in the following example.



4. Set the motion direction:
   a. For **Translation** (including **Periodic**), select the desired X, Y, or Z (for 3D) axis of motion from the **Moving Vector** drop-down menu; then select either **Positive** or **Negative** direction.
   b. For **Rotation**, from the **Moving Axis** drop-down menu, select an axis of rotation. Use the radio buttons to select either **Positive** or **Negative** rotation.

   > **Note:**
   > - For 2D XY rotation, only **Positive** rotation is allowed.
   > - If the expected moving vector does not exist, click **Modeler** > **Coordinate System** > **Create** > **Relative CS** to create a new coordinate system.

5. On the **Data** tab: type a value in the **Initial Position** box, and select the units from the drop-down menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. To set the motion limits, do the following in the **Translate Limit** or **Rotate Limit** section:

   Note: For Periodic motion, you do not need to define motion limits because the solver does not consider the limits for periodic motion; thus the **Translate Limit** section is not present.

   a. For rotational motion, click the **Rotate Limit** check box.
   b. For translational (non-**Periodic**) or rotational motion, type a value in the **Negative** box, and select the units from the drop-down menu.
   c. For translational (non-**Periodic**) or rotational motion, type a value in the **Positive** box, and select the units from the drop-down menu.

7. On the **Mechanical** tab, do one of the following to specify the object velocity (dynamic or constant):

   Note: When you select **Consider Mechanical Transient**, you are telling the solver to use the calculated force and user-specified mass, damping, and other parameters to dynamically determine how the object moves, rather than giving it a constant velocity.

   Note:
   - For Maxwell 2D/3D designs for motor applications, the computed output mechanical power at given constant speed may not reach the desired rated power. Therefore, it is useful to apply constant power as the mechanical load. In Maxwell 2D/3D designs created by RMxprt, the mechanical transient is set up according to the rated mechanical output power, and equivalent damping is added to accelerate the process approaching to the rated output power. This setup is visible when **Consider Mechanical Transient** is checked.
   - The **Consider Mechanical Transient** setup is only for the sake of getting the operating point with given output power. it is not related to any mechanical load type you may set in RMxprt.
   - 2D/3D designs generated from RMxprt include the setup data, but **Consider Mechanical Transient** is unchecked initially. To ensure that the setup data is preserved, make sure that you check **Consider Mechanical Transient** before saving the project.

   - To dynamically change how the object moves, click the **Consider Mechanical Transient** check box, complete the following fields, and select any corresponding units:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Initial Velocity** (translational) or **Initial Angular Velocity** (rotational)
- **Mass** (translational) or **Moment of Inertia** (rotational)

Use the **Calculate** button to have Maxwell calculate the Mass or Moment of Inertia with the following restrictions:

- For 2D problems, the moving bodies are assumed to be of model-unit thinness.
- Rotating bodies must be symmetric about the axis of rotation. Partial models (e.g., half a motor with implied symmetry) are not supported.
- **Damping** (rotational motion) – A value for damping for rotational motion (for electric machines) can be calculated by the windage and friction loss in Watts divided by the speed in rad/sec (at which those losses are defined); the resulting units are in W / $(rad/sec)^2$ or N*m * sec/rad.
- **Damping** (linear motion) – A value for damping for linear motion can be calculated by the windage and friction loss in Watts divided by the speed in m/sec (at which those losses are defined); the resulting units are in W / $(m/sec)^2$ or N*m * sec/m.
- **Load Force** (translational) or **Load Torque** (rotational)

  Note:
  - For translational motion, the load force is positive if it is applied in the same direction as the moving vector and negative if applied in the opposite direction.
  - For rotational motion, the load torque sign is determined based on the moving vector, using the right-hand rule. Point your thumb in the direction of the moving vector. If the load torque is applied in the same direction as your fingers, then the load torque is positive. If it is applied in the opposite direction, then the load torque is negative.

- To set a constant velocity (and disable the mechanical transient), clear the **Consider Mechanical Transient** check box, type a value in the **Velocity** or **Angular Velocity** box, and select the units from the drop-down menu.

8. For translational motion, a **Post-Processing** tab is available to set the coordinate system for post processing calculation.
9. Click **OK**.

- A **MotionSetup*x*** is created and added to the **Model** folder in the **Project Manager** window.
- For Maxwell 3D projects only, if cylindrical **Rotation** was specified and a **CylindricalGap** mesh operation does not already exist, a **CylindricalGap** mesh operation is automatically added to the **Mesh** folder. Only one **CylindricalGap** mesh operation is permitted.

Modifying Motion Setup Properties

To modify existing motion setup properties:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the Project Manager **Model** folder, right-click on **MotionSetup*x*** and select **Properties** from the context menu to open the **Motion Setup** dialog box.
   - Alternatively, you can double-click the **MotionSetup*x*** icon.
2. Edit the desired properties in the **Motion Setup** dialog box. Refer to *Assigning a Band of Motion* for detailed information on the various **Motion Setup** dialog box settings.

Note:

For Maxwell 3D projects only:

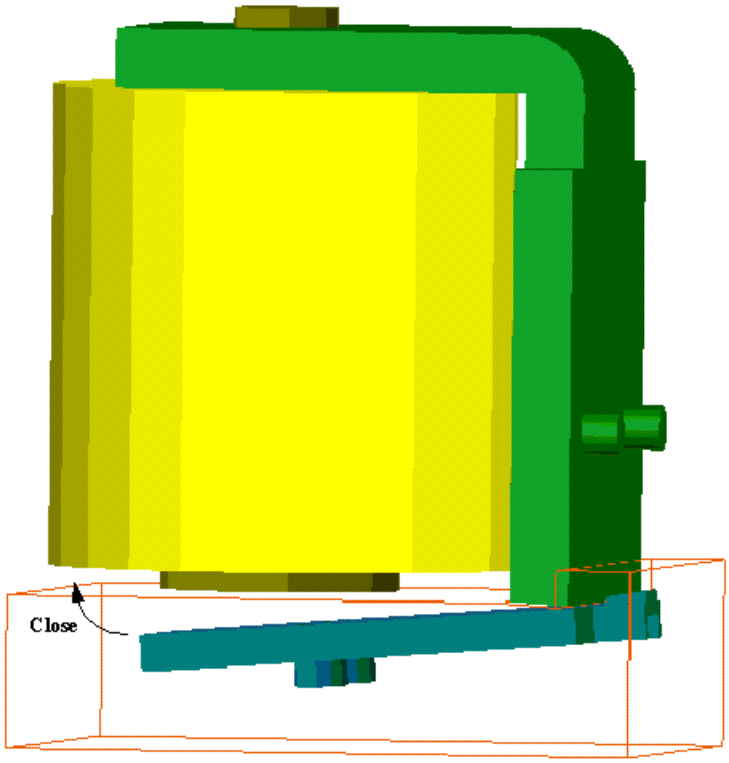
- If you change the motion type to cylindrical **Rotation** and a **CylindricalGap** mesh operation does not already exist, a **CylindricalGap** mesh operation is automatically added to the **Mesh** folder. Only one **CylindricalGap** mesh operation is permitted.
- If you change a cylindrical **Rotation** motion type to another motion type, the **CylindricalGap** mesh operation remains in the **Mesh Operation** folder.
- Alternatively, you can edit many of the motion setup properties in the **Properties** window.

Non-Cylindrical Rotational Motion

With cylindrical rotation, the moving objects are free to rotate completely, and the mesh remains fixed as the moving objects rotate. With non-cylindrical motion, the geometry shape of the band region changes as the moving objects rotate, and the mesh is recreated.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- A motor is an example of cylindrical motion.
- A relay is an example of non-cylindrical motion.



Reassigning a Band of Motion

When a **MotionSetup** has been created, the object that the setup is assigned to is changed using the **Reassign** command, allowing you to identify the motion setup parameters with a new object without losing the motion setup parameters themselves. To reassign a **MotionSetup**:

1. Select the object that the **MotionSetup** will be reassigned to in the modeler window or in the history tree.
2. Right-click on **MotionSetup*x*** in the Project Manager window and select **Reassign**.

> **Note:** An existing **CylindricalGap** mesh operation's assignment will be synced up to a rotational cylindrical motion setup's assignment when you reassign the band object of the motion setup.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adding and Deleting Objects from a Band of Motion

To add objects to, or delete objects from a motion setup without changing the parameters of the **MotionSetup** do the following:

1. Select the object to be added or deleted from the **MotionSetup** in the modeler window or in the history tree.
2. Right-click **Model** > **MotionSetup*x*** > **Moving*x*** in the Project Manager window and choose **Add Selected Objects** or **Remove Selected Objects** from the shortcut menu.
3. To verify the addition or deletion of the object from the MotionSetup definition, use **Select Assignment** from the shortcut menu to view the objects assigned to the band.

Showing and Hiding Motion Objects

1. Click **Maxwell** > **Model** > **Motion Setup** > **Visualization**.
2. From the **Motion Setup Visualization Options** window that appears, select **All**, **Band**, or **Moving** from the **Select** drop-down menu.
3. Click **Deselect All** to hide all motion objects.
4. Click **Close** to close the window.

Setting the Visibility for Motion Objects

To show or hide a parameter:

1. Click **View** > **Active View Visibility**.
2. From the **Active View Visibility** dialog box that appears, click the **Motion** tab.
3. Select or clear the **Visibility** check box for each motion object listed.
4. Click **Done**.

Modeling Maxwell Motion Eccentricity

You can model motion eccentricity for Maxwell 2D and 3D transient designs that have cylindrical rotational motion set up. Ansys Electronics Desktop provides a **Maxwell Eccentricity Wizard** with settings for

- rotating part eccentricity
- rotation axis eccentricity

For rotating part eccentricity modeling, the eccentricity is only applied to rotating objects, not to a motion axis. More specifically, these rotating objects are all objects enclosed by the band, but not the band itself.

For rotation axis modeling, the eccentricity is only applied to the motion axis, which is equivalent to moving the band itself, but not any rotating objects inside the band.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using these two basic settings, you can model the following physics.

- **Static eccentricity**: To model static eccentricity, you need to set the rotating part eccentricity and rotation axis eccentricity to the same values.
- **Dynamic eccentricity**: To model dynamic eccentricity, set the rotating part eccentricity values only.
- **Mixed eccentricity**: This is the combination of static and dynamic eccentricities. To model mixed eccentricity, set both rotating part eccentricity and rotation axis eccentricity, but with different values.

Using the Maxwell Eccentricity Wizard

> **Note:**
>
> The **ACT Extensions** window and the design wizards it contains (*5G Wizard, HFSS Antenna Design Toolkit, HFSS-EMA3D Link,* and *Maxwell Eccentricities*) are only available for the Windows version of the Ansys Electronics Desktop software. These items are not available when the software is installed on a Linux platform.

You can use the Maxwell Eccentricity wizard to model both 2D and 3D transient designs that have cylindrical rotational motion setup.

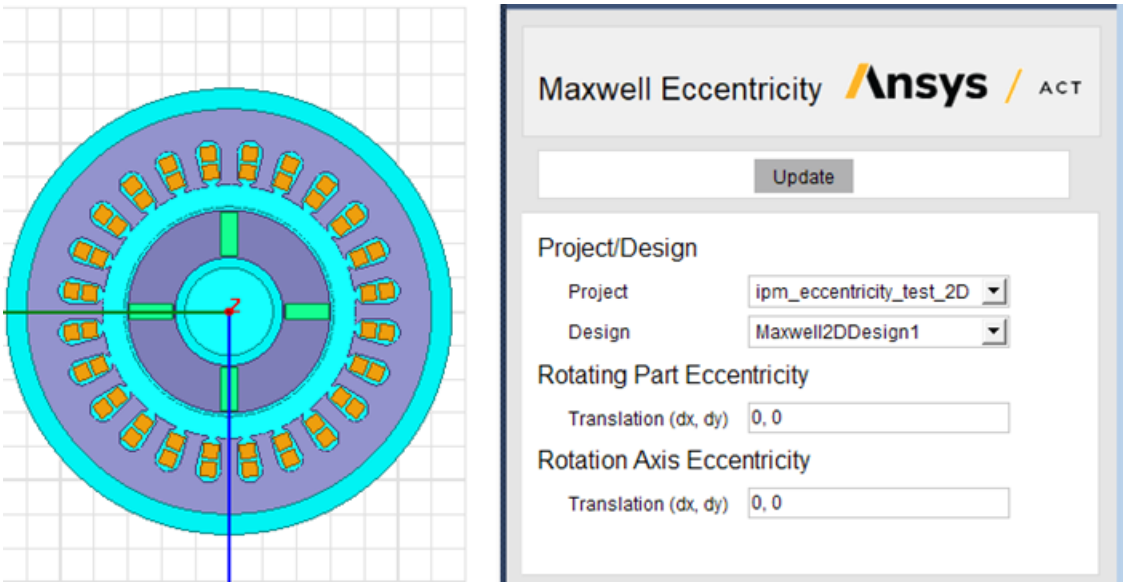
> **Note:**
>
> If you want to generate a 3D rotational eccentricity motor design from an RMxprt-generated Maxwell 3D design, you must enter **Eccentricity 1** in the RMxprt **Design Setting** window's **User Defined Data tab** for the RMxprt design *before* generating the Maxwell 3D design.

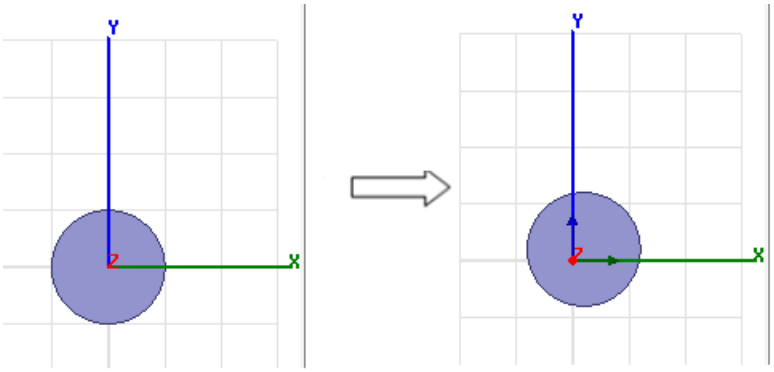
1. Select **View** > **ACT Extensions** to open the ACT Extensions window.
2. Click the **Wizards** button to open the Wizards page, then click **Maxwell Eccentricity** to load the Maxwell Eccentricity setup page.
3. Select the project and design you want to use.

   > **Note:**
   >
   > - The **Update** button synchronizes the project and design selection lists with those currently loaded in the Desktop.
   > - You can click **Exit Wizard** to close the eccentricity wizard and return to the wizard start page without executing any changes.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

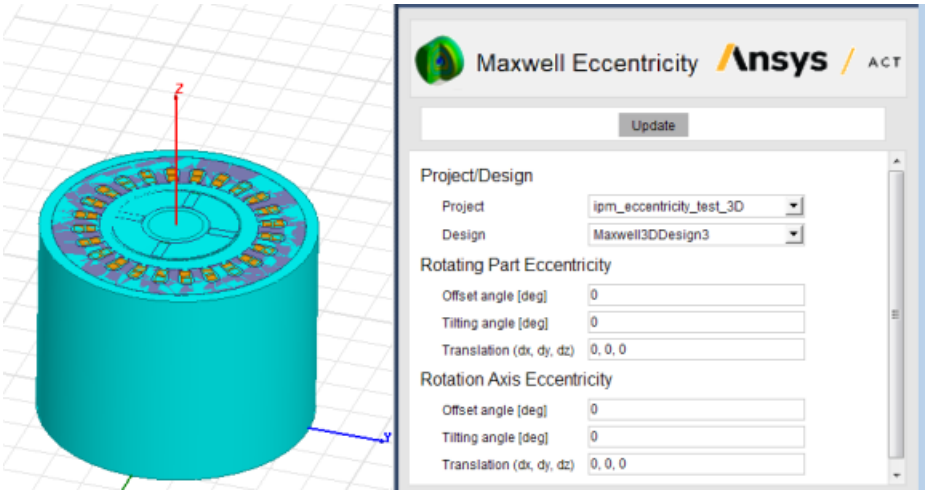
4. Depending on whether your design is 2D or 3D, do one of the following:
    - For a 2D Maxwell project, enter the desired dx and dy translation values to set up the Rotating Part Eccentricity and Rotation Axis Eccentricity. The unit of measure for translation values is the model's unit, which is set using **Modeler** > **Units**.



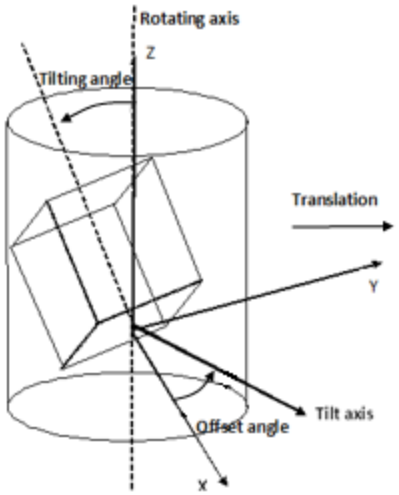


    - For a 3D Maxwell project, enter the desired dx, dy , and dz translation values, the offset angle, and the tilting angle for the rotating part eccentricity axis and rotation axis. The unit of measure for translation values is the model's unit, which is set using **Modeler** > **Units**.

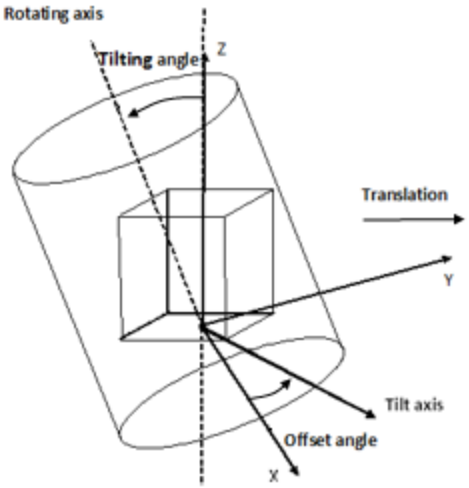
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Rotation Part Eccentricity



Rotation Axis Eccentricity



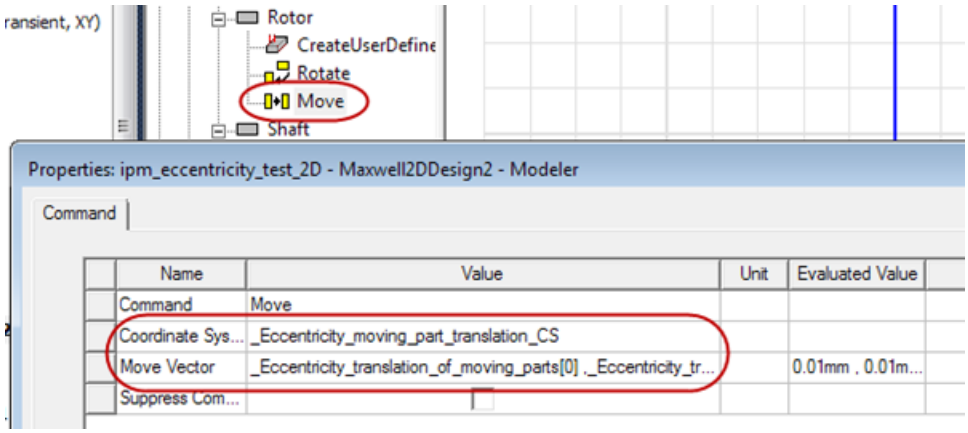
5. Click the **Finish** button at the bottom of the page. The page closes and the wizard adds a new design in the same project, in which the Maxwell geometry has been updated in accordance with the eccentricity setup values you entered, as explained in the sections that follow.

For 2D Designs

- Design variables are created for each of the data entered in the wizard.
  - For rotating part eccentricity: _Eccentricity_translation_of_moving_parts
  - For rotation axis eccentricity:_Eccentricity_translation_of_rotating_axis

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The moving parts in the design motion setup are shifted according to the data defined for **Rotating Part Eccentricity** in the Wizard. New coordinate systems are generated as the object coordinate systems for moving parts, according to the same operations.
- The band is translated according to the data defined for **Rotation Axis Eccentricity** in the wizard.
- The **Moving Vector** entry in the design's motion setup is replaced with the new direction (represented by an axis of a new coordinate system) generated according to the data defined for **Rotation Axis Eccentricity** in the Wizard.



- New design coordinate systems are created.

  _Eccentricity_rotating_axis_translation_CS

  _Eccentricity_moving_part_translation_CS

  _Eccentricity_moving_part_CS_<*part name*> – one for each moving part

Note:

- All the above operations of translation, as well as the creation of new coordinate systems, are dependent on the eccentricity design variables.
- You can modify these design variables to adjust the eccentricity.
- The newly added coordinate systems should not be changed manually by the user.
- Using the wizard to modify a design that has already been set up for eccentricity is not recommended.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For 3D Designs

- Design variables are created for each of the data entered in the wizard.

  For rotating part eccentricity:

  - _Eccentricity_offset_angle_of_moving_parts
  - _Eccentricity_translation_of_moving_parts
  - _Eccentricity_tilting_angle_of_moving_parts

  For rotation axis eccentricity:

  - _Eccentricity_offset_angle_of_motion_axis
  - _Eccentricity_translation_of_motion_axis
  - _Eccentricity_tilting_angle_of_motion_axis
- The moving parts in the design motion setup are rotated and shifted according to the data defined for **Rotating Part Eccentricity** in the Wizard. New coordinate systems are generated as the object coordinate systems for moving parts, according to the same operations.
- The band is rotated and translated according to the data defined for **Rotation Axis Eccentricity** in the wizard.
- The **Moving Vector** entry in the design's motion setup is replaced with the new direction (represented by an axis of a new coordinate system) generated according to the data defined for **Rotation Axis Eccentricity** in the Wizard.
- New design coordinate systems are created.

  For the motion moving vector:

  - _Eccentricity_motion_axis_CS
  - _Eccentricity_motion_axis_tilting_axis_CS

  For the moving parts:

  - _Eccentricity_moving_part_tilting_axis_CS
  - _Eccentricity_moving_part_translation_CS
  - _Eccentricity_moving_part_CS_*<part name>* – one for each moving part

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- All the above operations of rotation and translation, as well as the creation of new coordinate systems, are dependent on the eccentricity design variables.
- You can modify these design variables to adjust the eccentricity.
- The newly added coordinate systems should not be changed manually by the user.
- Using the wizard to modify a design that has already been set up for eccentricity is not recommended.

Note:

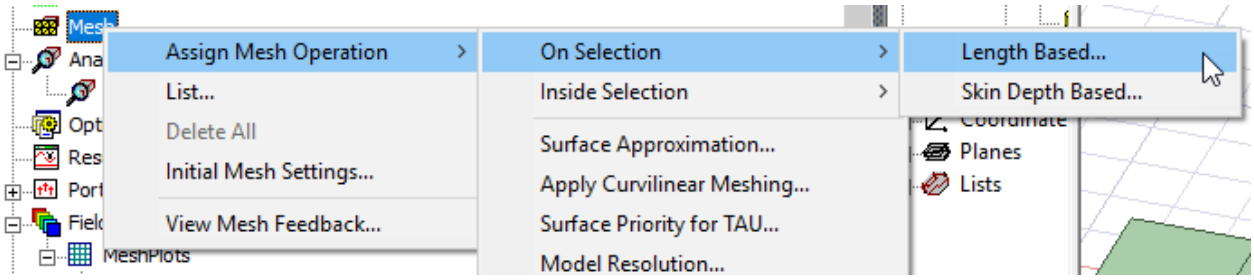
When modeling motion eccentricity, the position and direction of the moving objects and/or band are adjusted based on the eccentricity setup. In such cases, all the adjustments are referenced to the original position and direction of the geometry of each object, As a result, after you finish the eccentricity setup and click the **Finish** button, all the geometries in the design created by the wizard are updated to new positions based on the eccentricity setup. If you want to edit geometry, you should do so in the original design. After editing the geometry, you can then redo the eccentricity setup to create a new design.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

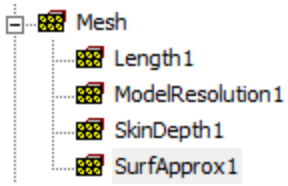
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12 - Defining Mesh Operations

In Project Manager, mesh operations are optional mesh refinement settings that provide the solvers with mesh construction guidance. This technique of guiding mesh construction is referred to as "seeding" the mesh. Seeding is performed using the **Assign Mesh Operation** commands on the respective solver menu.



Each mesh operation you define appears in the Project tree.



Note:

There are no restrictions against assigning multiple mesh operations to the same entities. The *type* of mesh operation can differ between overlapping assignments. For example, length-based refinement could be assigned inside a body, and skin-depth seeding on one or more of its faces. Additionally, overlapping mesh operations of the *same type* can have differing, or even redundant, parameter values (such as element length) without causing an error condition.

Typically, the operation that dictates the finest local meshing results will control the outcome in overlapping assignment areas or volumes.

When defining a mesh, you typically assign the mesh operations first (using the **Maxwell 3D** > **Mesh** or **Maxwell 2D** > **Mesh** commands) and then create the mesh (using the **Maxwell 3D** or **Maxwell 2D** menu and selecting **Analysis Setup** > **Generate Mesh** command). However, you can also refine the mesh after the initial mesh has been created.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can instruct the solvers to reduce the maximum size of tetrahedral elements on a surface or within a volume until they are below a certain value (length-based mesh refinement) or you can instruct solvers to refine the surface triangle length of all tetrahedral elements on a surface or volume to within a specified value (skin depth-based mesh refinement). These types of mesh operations are performed on the *current mesh*, that is, the most recently generated mesh.

In a few circumstances, you may also want to create a mesh operation that modifies a solver's surface approximation settings for one or more faces. Surface approximation settings are only applied to the *initial mesh*, that is, the mesh that is generated the first time a design variation is solved.

In some circumstances, you may also want to assign a cylindrical gap treatment mesh operation. For Maxwell 2D, which uses the TAU mesher is selected in the Initial Mesh Settings dialog box, you can define the Skin Depth Layers mesh setting corresponding to the selected edges.

You can also specify Initial Mesh Settings to apply to all objects; however, if you apply separate surface approximation mesh operations to specific objects, the object settings take precedence over the general setting.

> **Note:** Mesh operations on the target design of a mu (permeability) link are ignored.

You can also choose to override automatic choice of which mesher a product uses, by using the [*solverName*] > Mesh > Initial Mesh Settings command.

See the technical notes for more details about Ansys Electronics Desktop's application of mesh operations.

What do you want to do?

- Length Based Mesh Refinement
- Perform length-based mesh refinement on object faces
- Perform length-based mesh refinement inside objects
- Perform skin depth-based mesh refinement on object faces
- Assign Cylindrical Gap Treatment
- Apply Skin Depth Layer Settings
- Modify surface approximation settings for one or more faces
- Modifying the Model Resolution (Maxwell 3D only)
- Apply Curvilinear Elements
- Specify automatic or specified model resolution for a selection
- Specify the initial mesh settings
- TAU Flex Meshing

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specifying Initial Mesh Settings

You can specify the initial mesh settings, including the surface approximation and the meshing approach for both Maxwell 3D and Maxwell 2D designs. Initial Mesh Settings apply to all objects; however, if you apply separate surface approximation mesh operations to specific objects, the object settings take precedence over the general setting.

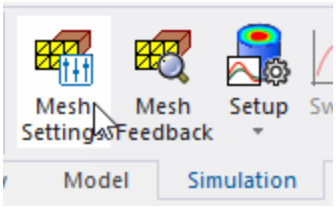
> **Note:**
>
> When you have defined mesh linking between a source and target design, all parameters within the *Initial Mesh Settings* dialog box are ignored for the target design. The mesh is copied from the source design without modification.

For most Maxwell 3D designs, you can let the solver automatically choose which meshing approach to take. Each solver predicts which one gives the best results, balancing mesh reliability, speed, quality, size and design characteristics.

In most cases, Maxwell uses TAU mesh, rather than the classic mesh. In general, it looks for specific features (for example, stacks of large planar parallel facets with small gaps) and situations where the initial Tau mesh is four times larger than the Classic mesh. For Maxwell 2D designs, the TAU mesh is used.

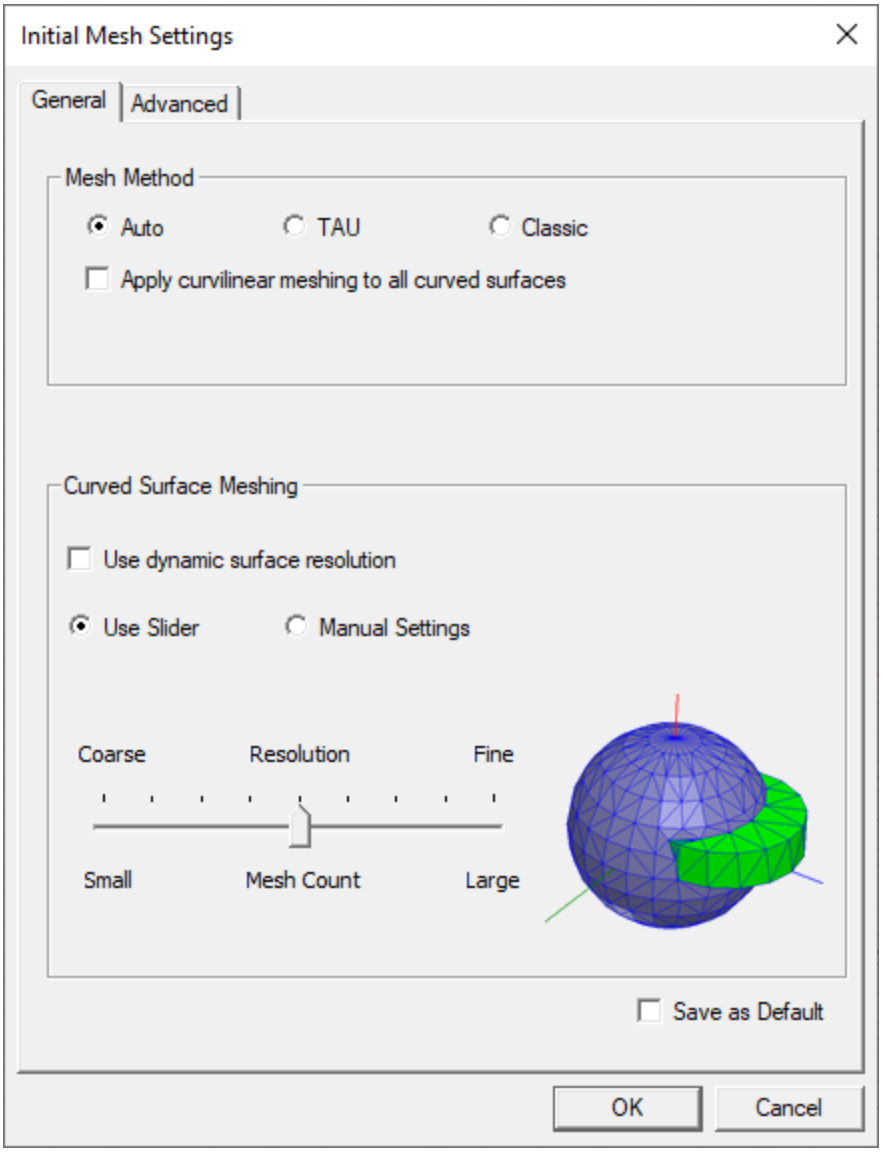
In a few cases, you may decide to override the automatic choice and designate the mesher to use. To do so:

1. Access the **Initial Mesh Settings** one of these ways:
   - Select **Maxwell** > **Mesh** > **Initial Mesh Settings...**
   - In the Project Manager, right-click **Mesh**, and select **Initial Mesh Settings** from the shortcut menu.
   - On the **Simulation** tab of the ribbon, select **Mesh Settings**.



   The **Initial Mesh Settings** window appears, with the **General** tab selected.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

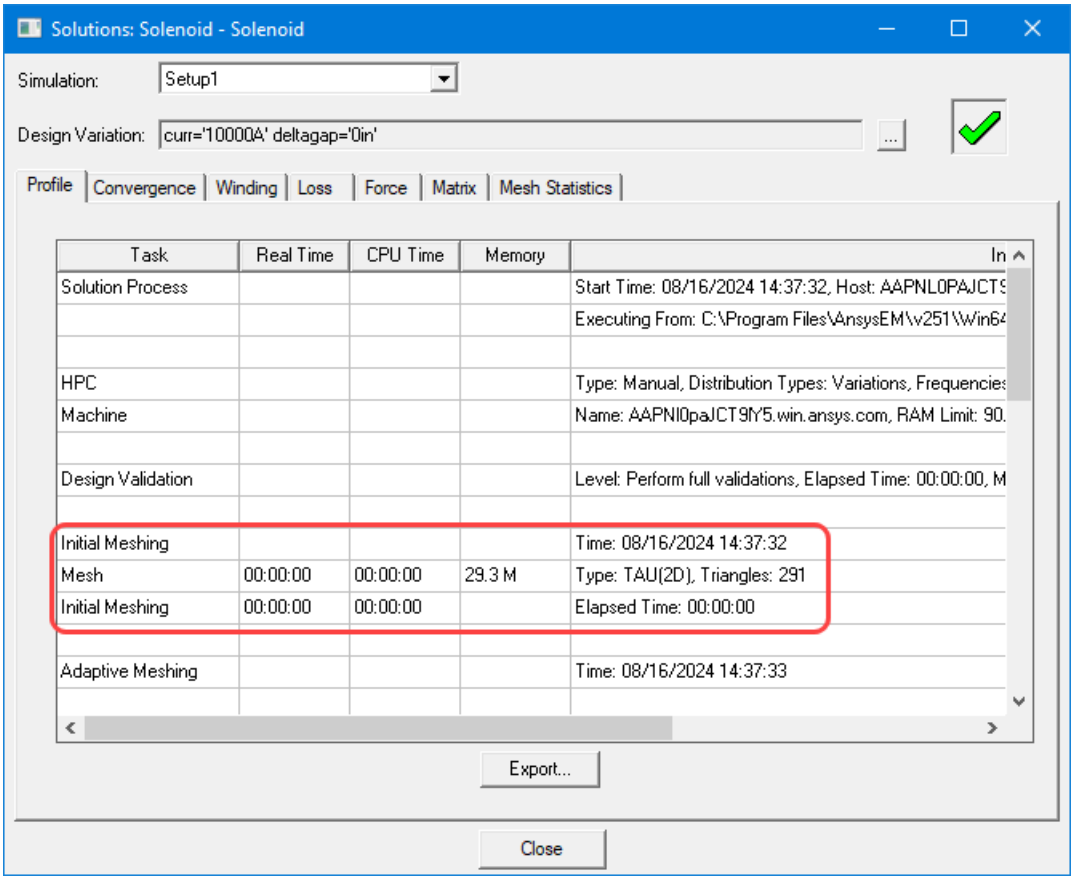


2. For Maxwell, depending on if it is a 2D or 3D design, the **Mesh Method** options you can select are

   - **Auto** – the solver automatically selects the mesher. This is the default setting. In most cases, this will be TAU.

     Note if this option selected, you can view the type of mesher used for the solution in the **Results** > **Solution Data**\ **Profile** tab.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **TAU** – Use for models that do not contain densely layered geometry. TAU is preferred for models with curved surfaces, and for complex geometry. With this method, only specific curve faces will be remeshed (for example, equation-based axisymmetric faces). If a curve face connects to non-remeshed curve faces, the curve face will not be remeshed. If a curve face cannot be remeshed, faceting triangles will be used as the surface mesh. Mesh quality depends on how the faceting triangles are made. If the surface mesh is generated by TAU, it shows in the profile as Mesh TAU (Surface). If you check **Use alternative mesh methods as fall back** on the **Advanced** tab, this can fall back to Mesh TAU(Wrapper). This, in turn, can fall back to Mesh (stitch), which is Classic.
- **Classic** – Use for models that are well defined, have fewer curved surfaces, and may contain some densely layered geometry. It can also be used for ECAD / MCAD assemblies that do not mesh with Phi or Phi Plus. Classic mesh uses a Bowyer algorithm to create compact meshes for the model with very large length scales. It represents the model very accurately. Depending on the model, Classic mesh may converge using fewer resources whereas TAU or Phi Plus may converge faster.

  This setting must be used if you wish to use the Phi Mesher, for which you must also check **Allow Phi for layered geometry (Classic only)** on the **Advanced** tab. The

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Phi mesher is suitable for layered geometry that is drawn in the XY plane and swept in the Z direction. This method is not suitable for geometry with faces that are not oriented in a vertical or horizontal direction.

For Maxwell 2D, the TAU mesher is used; there are no selection options.

- **Phi Plus** – this option is only available if the Maxwell design has layout components. It is best for complex assemblies that include layered geometry, but are not suitable for Phi. See Meshing Recommendations for Layout Components for more details.

3. For Maxwell models with curved surfaces, you can select or deselect **Apply curvilinear meshing to all curved surfaces**. This increases accuracy, but costs more memory.
4. Under **Curved Surface Meshing**, you can choose to **Use dynamic surface resolution**, select **Use Slider** or specify **Manual Settings**.

   If you select **Use dynamic surface resolution**, the **Manual Settings** option is disabled.

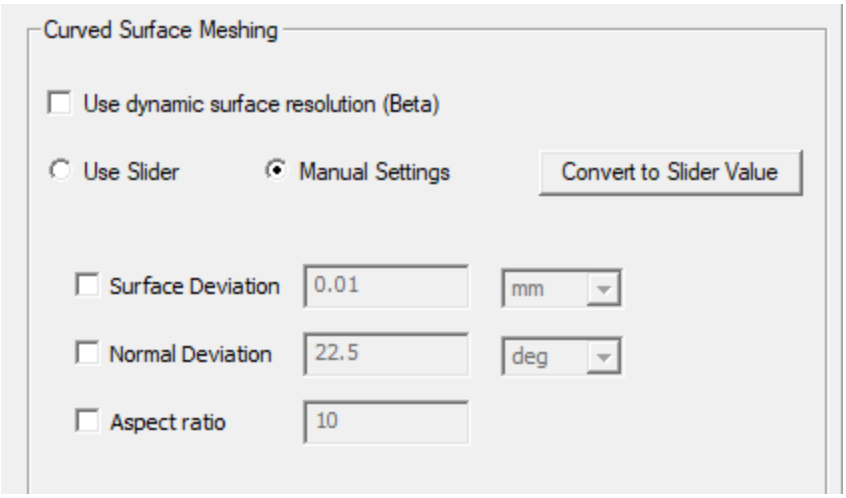
   The slider includes a visual representation of your choice, ranging from a ***Coarse*** *Resolution* (with a ***Small*** *Mesh Count*) through a nine position scale to a ***Fine*** *Resolution* (with a ***Large*** *Mesh Count*. Here, "*Mesh Count*" refers to the number of elements comprising the mesh.

   Selecting **Use dynamic surface resolution** in the *Curved Surface Meshing* section specifies the best-practice mesh operations over the geometric models. This mesh operation will support 3D volume mesh and surface mesh in all products. The default mesh operations or user-defined mesh operations may or may not be replaced by the optimized mesh operations with model analysis. In general, large curve faces, curve faces with small gaps, or skewed cables get more smooth curvature representation while small curve faces, such as fillets, small curve objects, and so forth, get relatively coarse triangulations so that the overall mesh count will be reduced.

   If you select dynamic surface resolution, you can optionally use the slider bar to adjust surface representation levels as: coarse (1-3), normal (4-6), and fine (7-9). You can also adjust the surface representation level for a specific group of faces by specifying slider bar, as described in Modifying Surface Approximation Settings.

5. If you choose **Manual Settings**, the window changes to show text fields. See Modifying Surface Approximation Settings.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

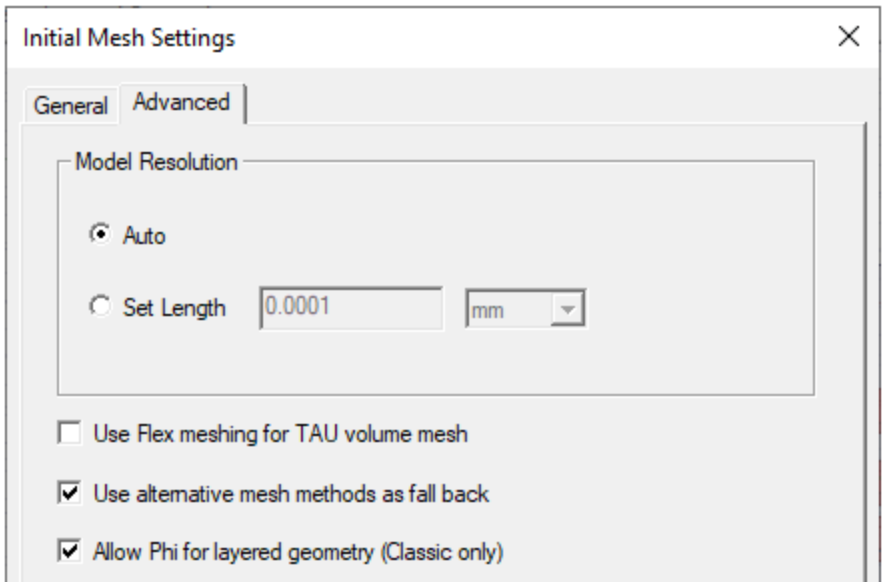


Use the check boxes to enable the fields and specify values:

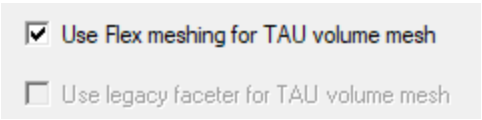
- **Surface Deviation** – the distance between the true surfaces of the selected faces and the meshed faces.
- **Normal Deviation** – the angular distance between the normal of the true surface and the corresponding mesh surface. Maxwell 3D's default normal deviation setting for the selected faces is 15 degrees. Maxwell 2D's default normal deviation setting is 3 degrees.
- **Aspect Ratio** – For curved faces, this setting determines the shape of the triangles. The higher the value, the thinner the triangles. Values close to 1 will result in well-formed, wide triangles.
- Clicking **Convert to Slider Value** converts the manually entered values to an equivalent slider setting, and returns the panel to the slider view.

6. To make your choices the default, use the **Save as default** check box.
7. Optional: use the **Advanced** tab to specify a **Set Length** for **Model Resolution**. This is for experienced users who have a good understanding of how particular values will affect their models. In general, the **Auto** setting provides good results.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



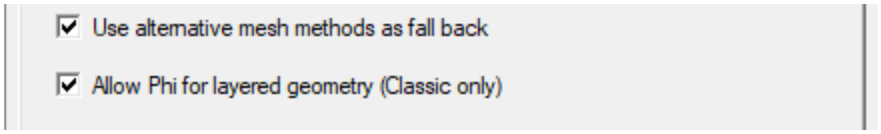
8. **Use Flex meshing for TAU volume mesh** enables a version of the TAU mesher that will rarely fail to generate a mesh (for Maxwell 3D models only). In most cases, the TAU Flex mesh is as accurate as traditional TAU or Classic meshes. However, for some complex models with bad translation or poorly defined surfaces that would fail to produce a strict mesh on all objects, relaxed tolerances will be applied. In these cases the user should review the mesh to evaluate whether it is acceptable for simulation. See TAU Flex meshing for more details.



> **Note:**
>
> TAU Flex meshing does not support RMxprt models.

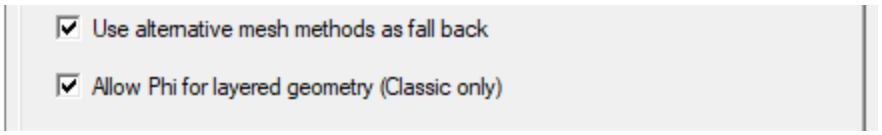
9. If your selected mesh method (Classic/TAU) failed to generate initial mesh, the solver can either stop or fall back on alternative mesh methods if you select **Use alternative methods as fall back**. You can also select **Phi for layered geometry (Classic only)**. For example, specifically, the mesher might go like this: When mesh method is TAU: TAU (>Phi)>Classic. When mesh method is Classic: Classic(>Phi)>TAU. Note: the options do not affect the Auto mesh setups.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



10. For Maxwell 3D models only, selecting **Use alternative mesh methods as fall back** enables the **TAU** and **Classic** mesh methods (chosen on the **General** tab) to fall back to alternative mesh methods if the solver fails to generate the initial mesh.

    You can also select **Allow Phi for layered geometry (Classic only)**, which enables the **Classic** mesh method (chosen on the **General** tab) to use the Phi mesher for layered geometry.

    For example, the the sequence of mesher methods might go like this: When the mesh method is **TAU**: TAU(>Phi)>Classic. When mesh method is **Classic**: Classic(>Phi)>TAU. Note: the options do not affect the Auto mesh setups.

11. Click **OK** to apply your choices.

The settings will be applied to the initial mesh generated.

Length Based Mesh Refinement

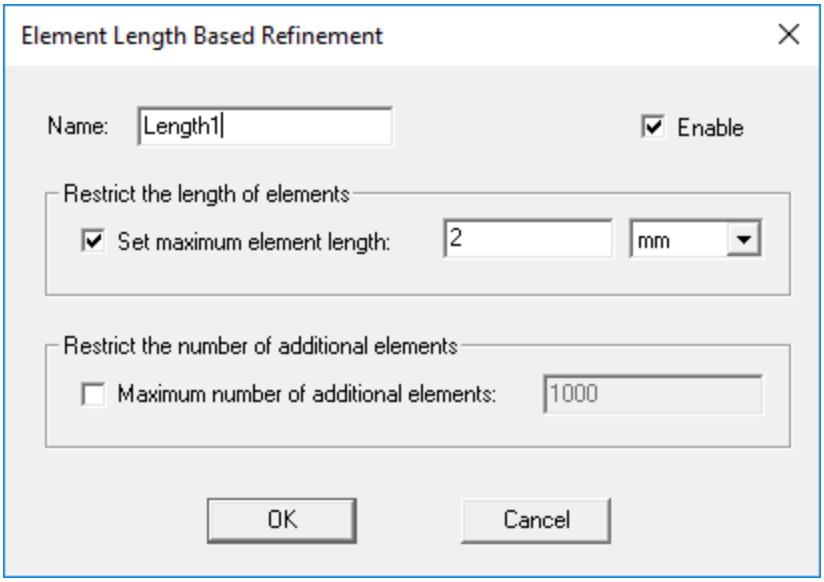
Length-based mesh refinement settings help control the length of the tetrahedral elements and the growth of the whole mesh. If you are not sure how big a refinement is needed, by way of mesh refinement you can set a ceiling on the mesh growth with the option **Restrict the Number of Additional Elements**.

To apply a length based mesh operation either to an object face or inside an object:

1. Select the object in the modeling workspace.
2. In the Project Manager, right-click **Mesh** > **Assign Mesh Operation** > **[On Selection / Inside Selection]** > **Length Based**.

   The **Element Length Based Refinement** dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Enter a **Name** for the mesh operation, or leave the default.
4. Select the **Enable** check box to set the operation as active. You can later deselect this to deactivate it.
5. To restrict the length of tetrahedra edges, select the **Set maximum element length** check box and specify values and units using the drop-down menus.
6. To restrict the number of elements added during refinement, select the **Maximum number of additional elements** check box and specify a value.
7. Click **OK**.

When you set a mesh operation on a selection of objects (more than one) and set a maximum length, it applies to the whole mesh. Even if you selected just one object, when you refine its boundary, elements are added both inside and outside of that object.

When there are no more elements to refine or when no more elements can be added, the refinement process stops because it has either met the growth limit or the element length target. However, there are more quality improvement swaps, smoothing, matching boundary issues performed at the finishing stage that might slightly alter the mesh size.

If you restrict the number of elements to say 10000, it will add 10000 elements. If you have multiple mesh operations each for 10000 elements, each mesh operation will add 10000 elements (assuming there was no element length targets).

If you set a restriction of say 2000 elements on an object which creates 1100 elements after lambda refinement without any mesh operation, the starting mesh will contain more than 3000 elements.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning Length-Based Mesh Refinement on Object Faces

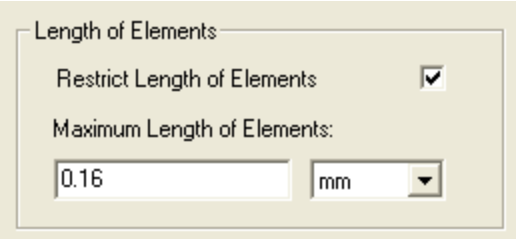
1. Select the faces you want to refine.

   Alternatively, select an object if you want to refine every face on the object.

2. Click **Maxwell** > **Mesh** > **Assign Mesh Operations** > **On Selection** > **Length-Based** or right-click **Mesh** in the Project Manager and select **Assign Mesh Operation** > **On Selection** > **Length-Based.**

   The *Element Length-Based Refinement* dialog box appears.

3. Type a name for the mesh operation in the **Name** text box or accept the default name.
4. To restrict the length of tetrahedra edges touching the faces:

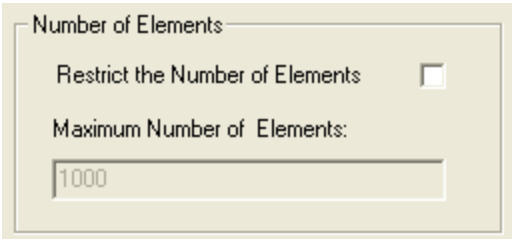


   a. Select **Restrict Length of Elements**.
   b. Type the maximum length of the tetrahedral edges touching the faces in the **Maximum Length of Elements** text box.

   The solver will refine the element edges touching the selected faces until their lengths are equal to or less than this value.

   The default value is set to 20% of the maximum edge lengths of the bounding boxes of each selected face.

5. To restrict the number of elements added during refinement of the faces:



   a. Select **Restrict the Number of Elements**.
   b. Enter the **Maximum Number of Elements** to be added.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Click **OK**.

When the mesh is generated, the refinement criterion you specified is used. If the maximum number of elements is reached, some elements may exceed the requested maximum element length.

You can also specify Initial Mesh Settings to apply to all objects; however, if you apply separate surface approximation mesh operations to specific objects, the object settings take precedence over the general setting.

Assigning Length-Based Mesh Refinement Inside Objects

Length-based mesh operations inside volumes can be applied to non-model box primitives. If an error occurs, you may be trying to assign a mesh to a non-model box that is not a box primitive (such as boxes with chamfers or non-model cylinders).

To instruct a solver to refine every face of an object and its interior:

1. Select the object you want to refine.
2. Click **Maxwell** > **Mesh** > **Assign Mesh Operation** > **Inside Selection** > **Length-Based**, or right-click **Mesh** in the Project Manager, and select **Assign Mesh Operation** > **Inside Selection** > **Length-Based...**.

   The *Element Length-Based Refinement* dialog box appears.
3. Type a name for the mesh operation in the **Name** text box or accept the default name.
4. To restrict the length of the tetrahedral element edges inside the object:
   a. Select **Restrict Length of Elements**.
   b. Type the maximum length of the edges inside the object in the **Maximum Length of Elements** text box.

   The default value is set to 20% of the maximum edge lengths of the bounding boxes of each selected object's faces.

   The solver will refine the element edges inside the object until they are equal to or less than this value.
5. To restrict the number of elements added during the refinement inside the object:
   a. Select **Restrict the Number of Elements**.
   b. Enter the **Maximum Number of Elements** to be added.
   c. Click **OK**.

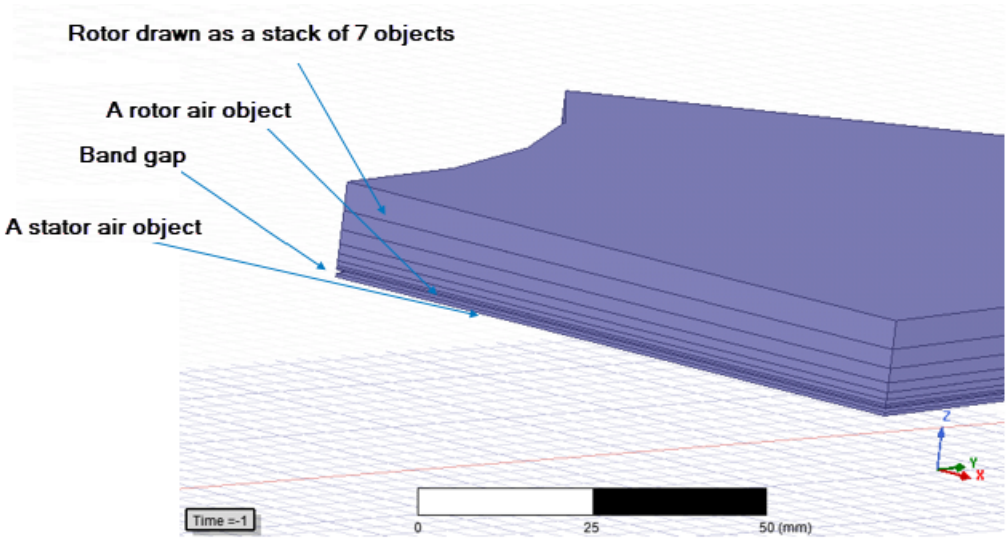
When the mesh is generated, the refinement criterion you specified will be used. If the maximum number of elements are reached, it may result in some elements exceeding the requested maximum element length.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also specify Initial Mesh Settings to apply to all objects; however, if you apply separate surface approximation mesh operations to specific objects, the object settings take precedence over the general setting.

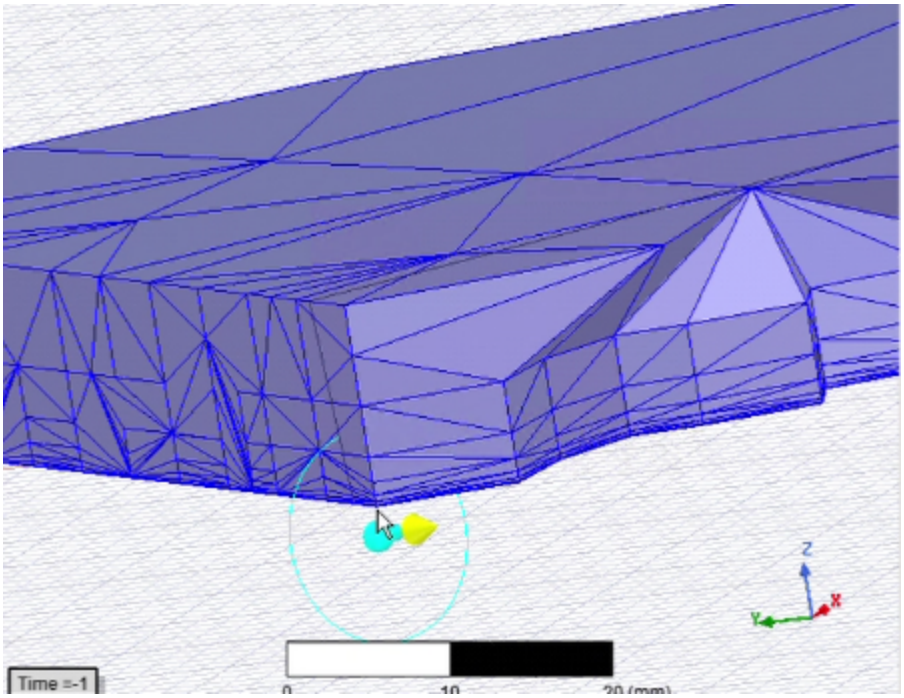
Assigning Skin Depth-Based Mesh Refinement on Object Faces

The Skin Depth-based mesh refinement lets you calculate or specify a skin-depth for mesh refinement. You can also specify the number of layers of elements for refinement where the skin depth is the total depth of all layers combined. These layers provide an easy, alternative approach than by creating physical models of each layer, pseudo-sheet bodies. Whereas creating and adjusting a complex, layered physical model is difficult, changing the skin depth parameters and number of layers is very easy. For example, consider a Maxwell rotor, drawn as a stack of seven objects:



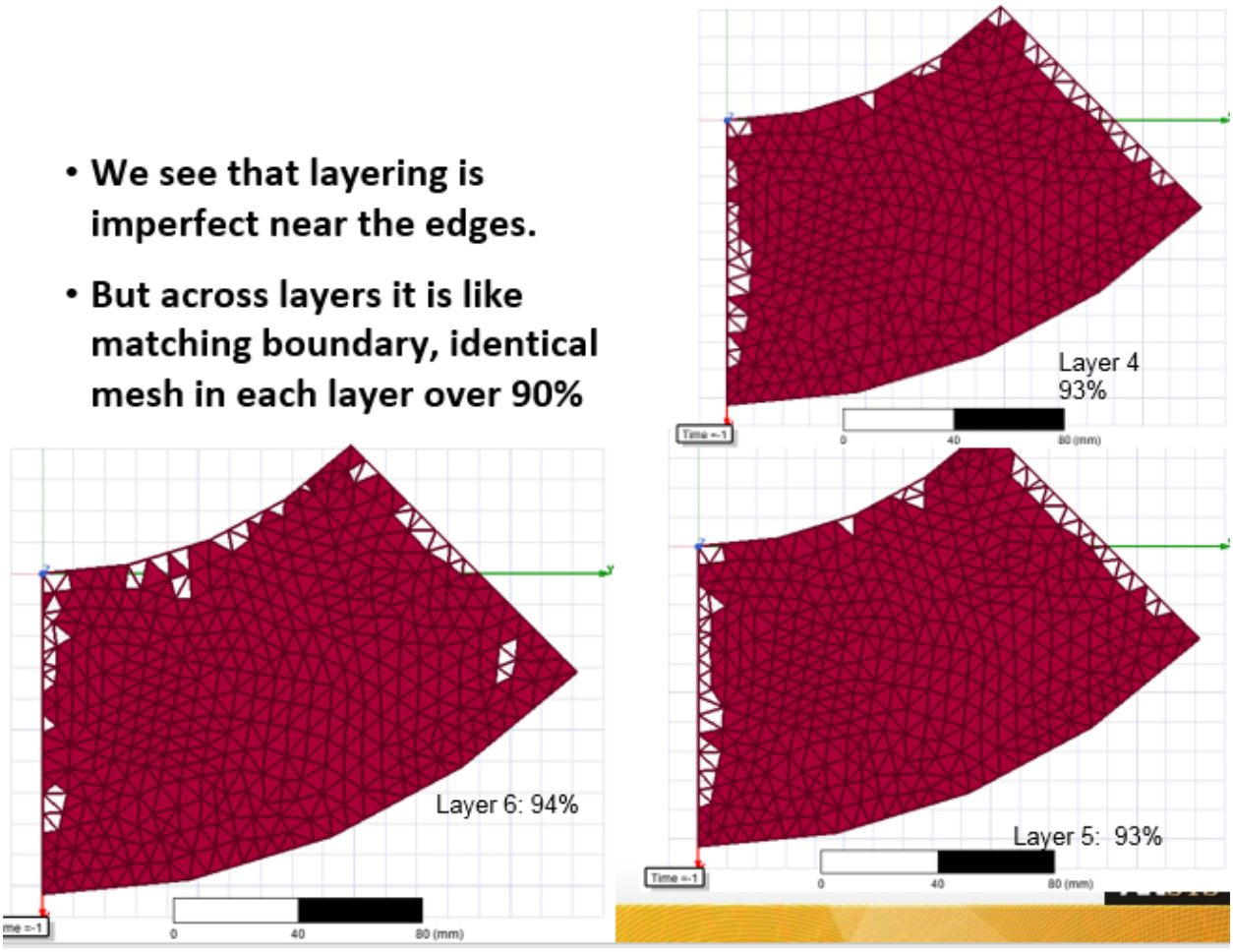
To test various stacking models with pseudo bodies involves recreating the model with different thicknesses and remaking the entire initial mesh. The Tau mesher may not be available for all layer heights. You can achieve comparable results by using skin depth based refinement by reverting to the initial mesh, changing settings and applying mesh operations. The following figure shows a cut-plane image of the interior of a model that uses skin-depth based refinement and layers of elements. The Skin Depth is the total depth of all layers combined. Layered elements apply to the selected faces of solid bodies. The elements are stretched parallel to face, and are compressed in the normal direction.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



While in some places the edges have imperfect layering, the interior is very good. You can also view the results layer by layer by using the Model Analysis dialog.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The layered skin depth mesh operation provides a tool toward finding a good solution. You can try different settings to improve results toward a good solution. For instance, for this example, these different settings were tried. For comparison, a model built with seven layers required 85K tets.

Tri length target mm	Total skin depth mm	Number of layers	Mesh Size element count	Perfect layering by tri count	Perfect layering by area
15.0	6.3	6	63K	43%	65%
10.0	6.3	6	68K	95%	94%
8.0	6.3	6	76K	96%	93%
6	6.3	6	97K	97%	94%

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The goal is not to have perfect layering, but to have a good solution that is much easier to achieve.

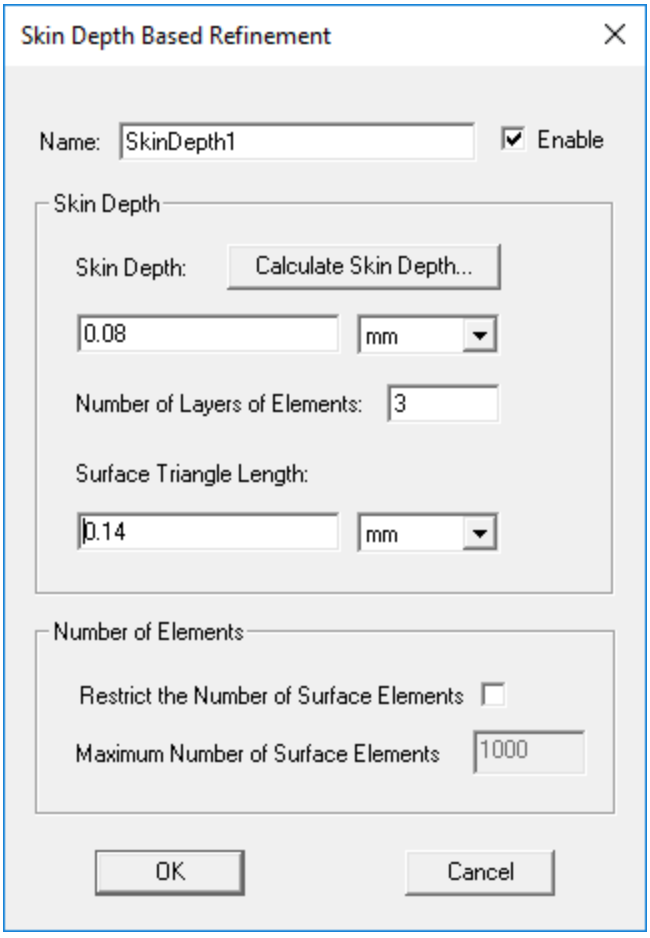
To use Skin Depth-Based Refinement:

1. Select the faces you want to be refined.

   > **Note:**
   >
   > It is possible to select a body and convert it to selecting all faces of the body. The user can use this method to select all faces and toggle a few faces out of selection. Selecting the whole body might select very large regions for refinement and increase the element count a lot.

2. Click **Maxwell** > **Assign Mesh Operation** > **On Selection** > **Skin Depth Based**.

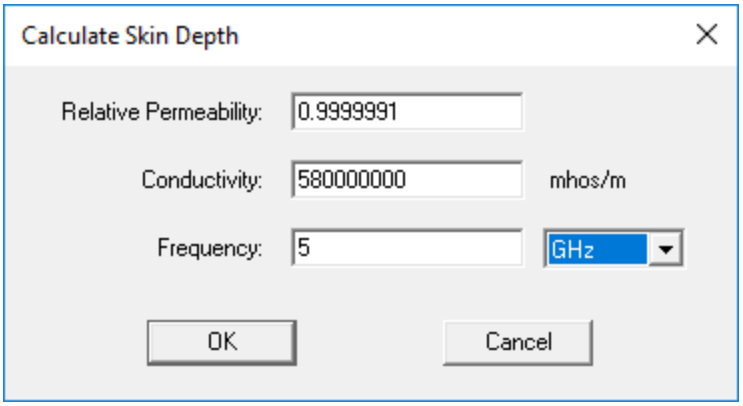
   The *Skin Depth Based Refinement* dialog box appears.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. To calculate the skin depth based on the object's material permeability and conductivity and the frequency at which the mesh will be refined, click **Calculate Skin Depth**.

   The **Calculate Skin Depth** dialog box appears with values based on the selected object's material properties and the solution setup frequency.



   Accept or edit these values. When you click **OK** the solver calculates the skin depth and enters its value in the Skin Depth text box. You can accept the calculated values or provide your own.

4. In the **Number of Layers of Elements** text box, type the number of layers to add perpendicular to the object's surface. The skin depth is the total depth of all layers combined.

   The solver will add an equivalent number of mesh points to each layer. For example, if HFSS added 10 points to satisfy the **Surface Triangle Length**, it will add 10 points to each layer.

5. Optionally, provide the maximum edge length of the surface mesh in the **Surface Triangle Length** text box. The default value is set to 20% of the maximum edge lengths of the bounding boxes of each selected face.

   The solver will refine the surface triangle mesh (the faces of the tetrahedra touching the surface) until their edge lengths are less than or equal to the specified value.

6. By default, the Restrict Number of Surface Elements setting is unchecked. This allows the mesher to use symmetry more effectively. If you Restrict the number of surface elements, this may affect symmetry but can be used to protect against runaway refinement in specific cases. To restrict the number of elements added during refinement on the faces uncheck the box to enable the field for Maximum Number of Surface Elements. With the box unchecked, the field, unavailable, the number in the box provides an estimate of mesh growth.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the mesh is generated, the refinement criteria you specified will be used. This operation will be approximately the same as having slabs of tetrahedra, but it is not guaranteed to prevent tetrahedra from crossing slab interfaces. Caution should be used with this mesh operation, as very thin layers may cause a reduction in mesh quality or unnecessarily cause the generation of a very large mesh. Further regions refined under this operation and its close neighbors do not participate in solution adaptive refinement. This is another reason to use this seeding operation with caution.

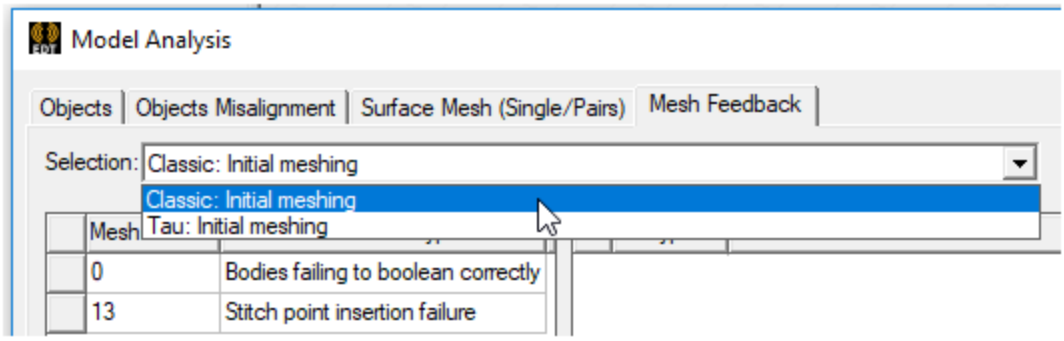
You can also specify Initial Mesh Settings to apply to all objects; however, if you apply separate surface approximation mesh operations to specific objects, the object settings take precedence over the general setting.

Visualization for Skin Depth Refinement Mesh Results

You can visualize the mesh to assess results either by using the clip-plane feature on a model with a mesh plot, or by using the Model Analysis dialog box as follows:

1. Select the object on which you have generated a meshplot, and click **Modeler** > **Model Analysis** > **Show Analysis Dialog** > **View Mesh Feedback**.

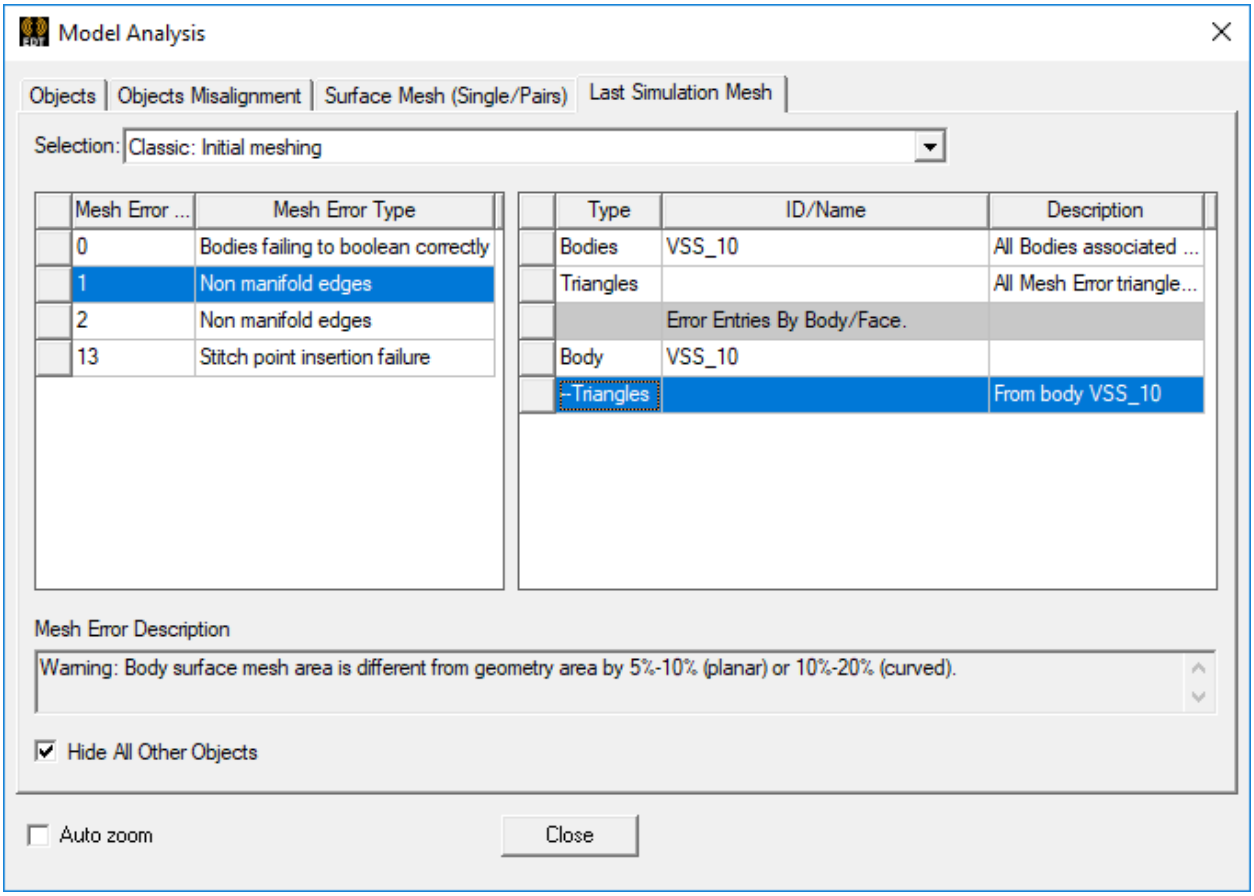
   The **Model Analysis** dialog box appears. If the project has solutions by different solvers, the **Selection** drop-down menu lets you select which mesh to view.



2. On the upper left table, under Mesh Error, select 0 through n , where 0 is the selected body, 1 is the layer closest to the face, 2 is the next, up to layer "n" as farthest.

   Selecting one of these rows causes a **Mesh Error Description** to display for the selected layer or body. The message gives the Total Expected count for triangles, as compared with the Success percentage by count and by area.

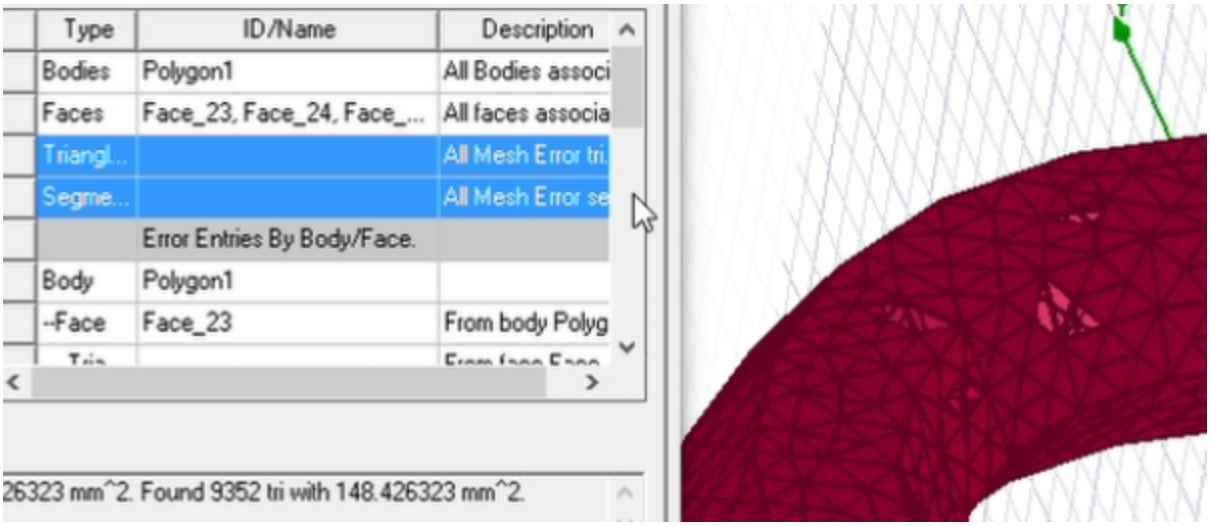
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. On the upper right table, select the body, face, or triangle desired, scrolling the list to find any object, face, triangle, or area of interest.

   The following figure shows Triangle and Segment both selected (using Shift+click). Visually, the unfilled triangles represent places where the triangle is not on the desired layer. However the mesh is still very good and supports an accurate solution.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Check boxes allow you to **Hide All Other Objects** checked by default, and **Auto zoom to selection**, unchecked by default.

Assigning Edge Cut Mesh Operation

1. Select the faces you want Maxwell to refine.

   Alternatively, select an object if you want Maxwell to refine every face on the object.

2. Click **Maxwell 3D** or **2D** > **Mesh** > **Assign Mesh Operation** > **Inside Selection** > **Edge Cut Based**.

   The **Edge Cut Based Refinement** dialog box appears.

3. Type a name for the mesh operation in the **Name** box, or accept the default name.
4. Specify the **Layer Thickness**, and select the units.
5. Click **OK**.

When the mesh is generated, the refinement criteria you specified are used. When the maximum number of elements is reached, some elements may exceed the requested maximum element length.

Refer to Layer Mesh Generation for 3D Rotational Models for information on edge cut for 3D designs.

You can also specify Initial Mesh Settings to apply to all objects; however, if you apply separate surface approximation mesh operations to specific objects, the object settings take precedence over the general setting.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning Cylindrical Gap Treatment

1. Select the object you want refined.
2. Click **Maxwell 3D or Maxwell 2D** > **Mesh** > **Assign Mesh Operation** > **Cylindrical Gap Treatment**

   A **Cylindrical Gap** mesh operation is applied to the selected object, and a **Cylindrical Gap** icon appears in the Project Manager's Mesh Operations folder.

   The Cylindrical Gap operation only works for the TAU (tolerant) mesher. It is a specific mesh operation assigned to the “Band” object for a rotational transient simulation.

   While you can select any object and assign the Cylindrical Gap operation to it in any Maxwell or Mechanical–Thermal simulation, the Cylindrical Gap operation is effective only for rotational transient simulations, for which cases the TAU (tolerant) mesher will be chosen as the default mesher; or for which you have manually selected the TAU (tolerant) mesher.

   > **Note:** For 2D transient simulations of rotational models only, refer to TAU2D: Band Mapping Angle Mesh.

   The TAU (tolerant) mesher analyzes the “Band” object, calculates the surface normal angle for the band rotational surface, and automatically assigns the desired mesh operation; or adjusts the existing mesh operations to the adjacent true surfaces at both static and moving sides to ensure a smooth and uniform mesh for the “Band” mesh.

   If the Cylindrical Gap mesh operation is assigned to a regular object, TAU analysis finds the object is not suitable for transient mesh, in which case the regular TAU mesh will be applied. A Warning message is then displayed in the message window.

   > **Note:**
   >
   > - There can be only one cylindrical gap operation per design.
   > - The operation cannot be assigned to multiple objects.

3. If you wish to change the name of the mesh operation, you can double-click the **Cylindrical Gap** icon to open the **Cylindrical Gap Mesh Operation** dialog box in which you can type a name for the mesh operation in the **Name** box.
4. Optionally, for 3D transient simulations of rotational models only, if you wish to apply a clone mesh operation, check **Clone Mesh**. This option is unchecked by default.
   - When **Clone Mesh** is selected, the **Band Mapping Angle** setting is enabled. The valid range of angles is 0.1 through 3.0 degrees, inclusive. The default band mapping angle is 3 degrees. (For more information on this setting, refer to *Band*

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mapping Angle.)

> **Note:**
>
> If the model is not suitable to make a clone mesh or the clone mesh process fails, the program automatically falls back to the regular TAU 3D mesher. Because the clone mesh is generated in the initial mesh, the initial mesh is deleted and regenerated once this option is changed and you click **OK**.

- When **Clone Mesh** is selected, the **Number of Layers** setting is enabled. You can input the number of mesh layers on the **Moving Side** and **Static Side**, respectively. The valid ranges for these settings are integers ≥ 1. The default values are 1. (For more information on this setting, refer to *Layers in Band Mesh*.)

3D Clone Mesh Generation

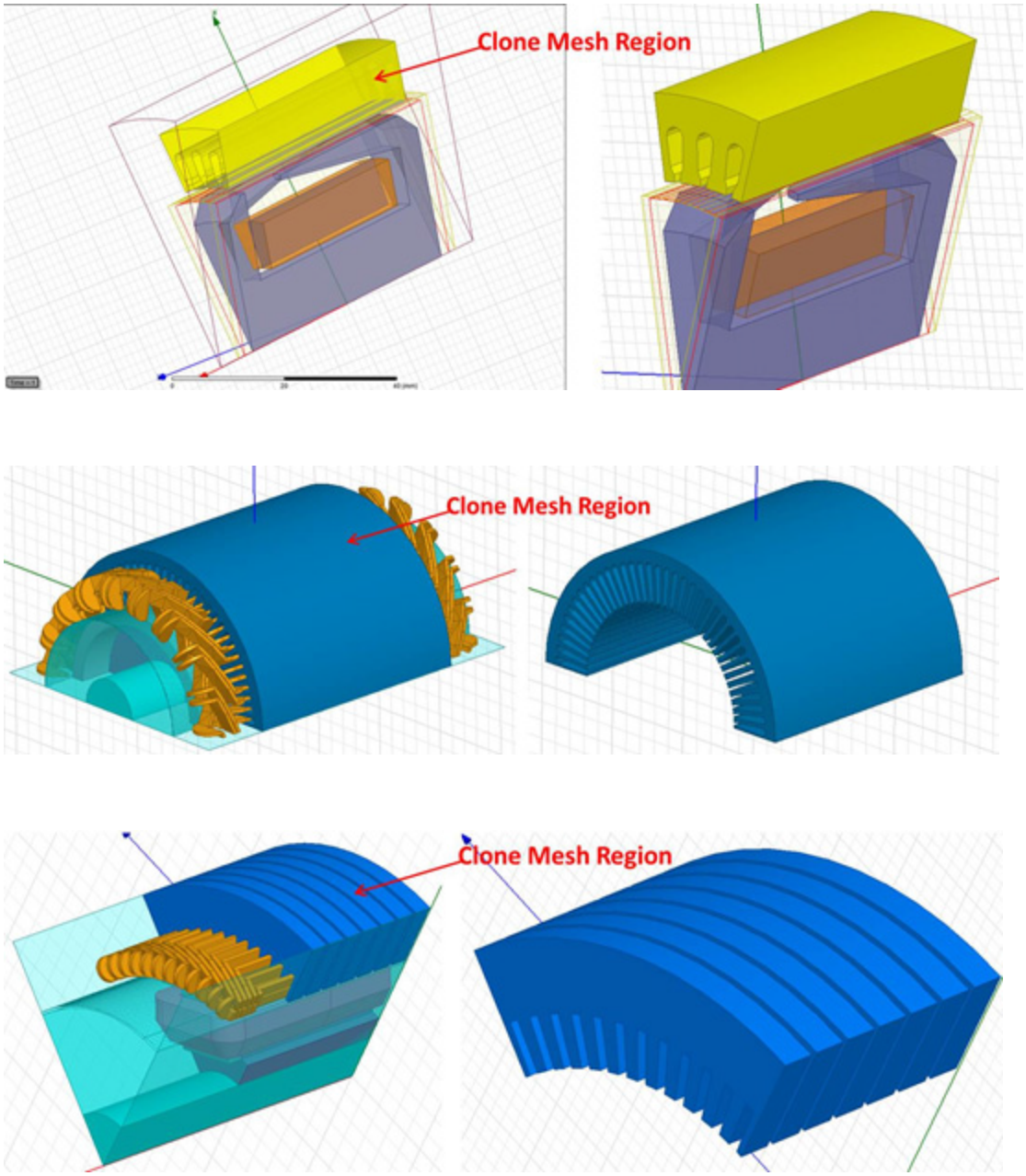
The 3D clone mesh is generated only when the following three conditions are met:

- The Clone Mesh check box is checked.
- The CylindricalGap mesh operation has been assigned.
- The model has a non-skewed stator with more than two teeth.

> **Note:**
>
> When Clone Mesh is used, the only additional mesh operation that can be used is surface approximation. If any other mesh operation is used, the clone mesh operation will be ignored.

The solid bodies adjacent to the band are detected to identify the clone object. If the object has more than two teeth that are close to the band, the teeth will be verified to see they have the identical geometric features. If the teeth are skewed, TAU will fall back to the regular mesh. Otherwise the region from the radius of teeth to the outer edge of the object will be further classified and verified to see if the object has the multiple 3D identical sub-regions.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



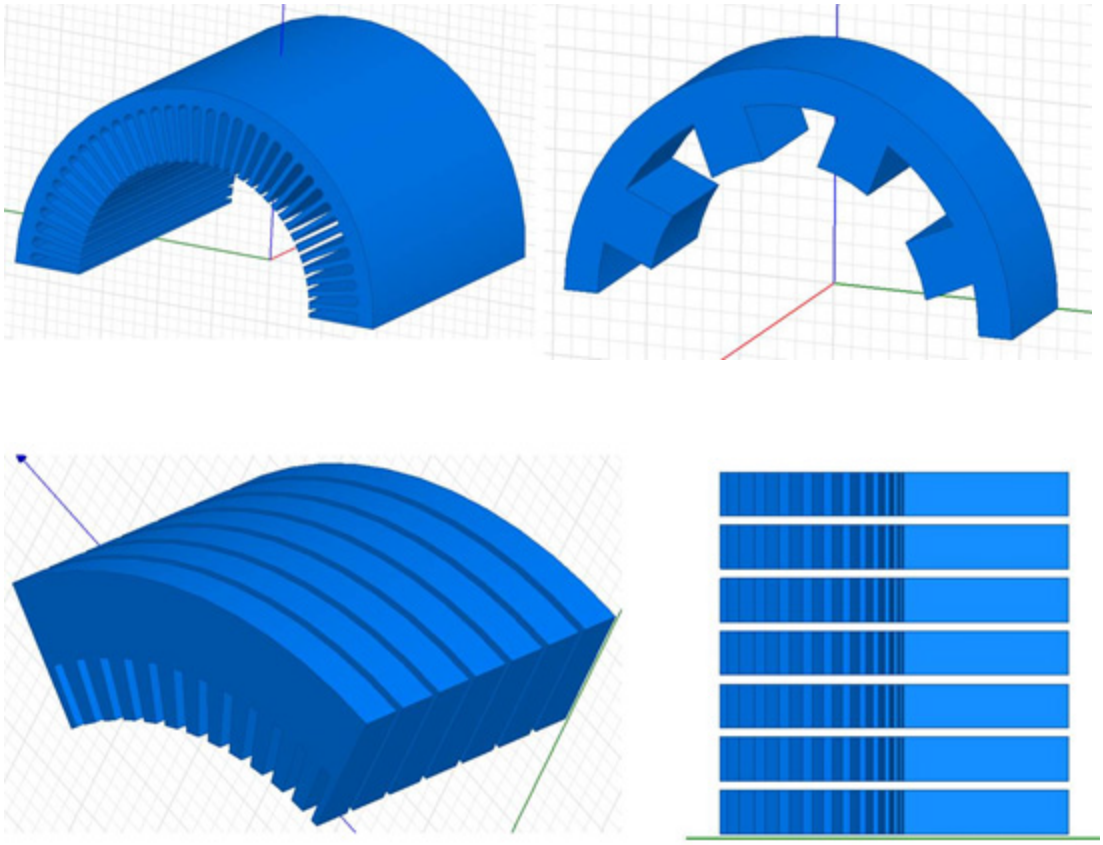
If a 3D clone region has been identified, TAU will classify the 3D rotational model into a static region and a moving region by the band object radius. The Static region includes the clone region and skewed coils regions at one side or two sides. Clone mesh method will then be used to generate the mesh in the clone region, while the TAU regular mesh method will be used to generate the mesh at other skewed regions.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The moving region includes skewed coils regions in which the TAU regular mesh method is used to generate the mesh, and also the non-skewed sweep regions in which the TAU sweep mesh method is used to make the mesh.

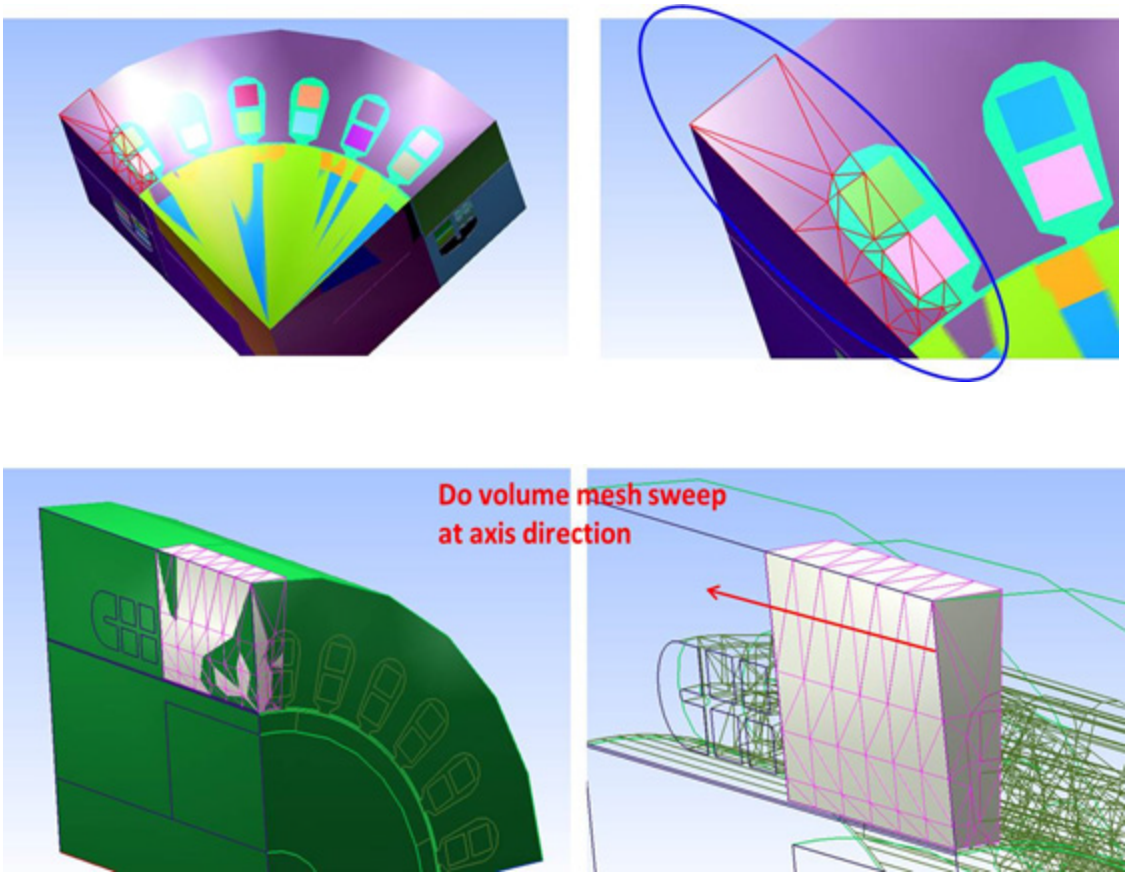
3D Clone Mesh

A 3D clone region may have only one simple object, or several sections of clone objects with the gap between as shown in the following images.



To ensure the identical volume mesh in each sub-region, the volume mesh is generated in the Master region, then by using a copy-and-paste method to duplicate the mesh at all child regions. In the 3D master region, if the region is also symmetric, then a symmetric mesh is generated. “Sweeping mesh” technology is used to make the volume mesh in the master region. In the half of the master region, a surface mesh is generated at the axis side. The volume mesh is generated by sweeping the surface mesh in the axis direction to fulfill the clone master region.

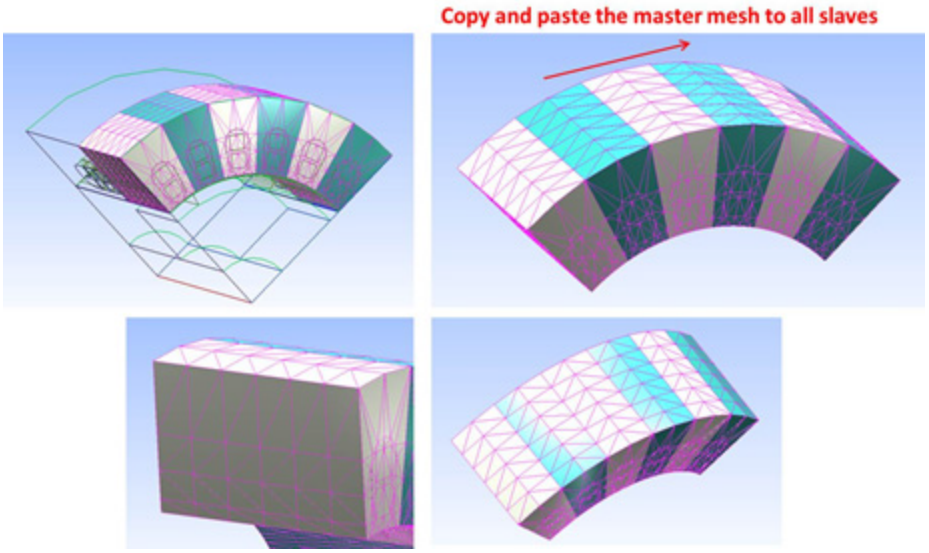
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The volume mesh will be mirrored to the other half of the master region to produce the symmetric mesh in the master region.

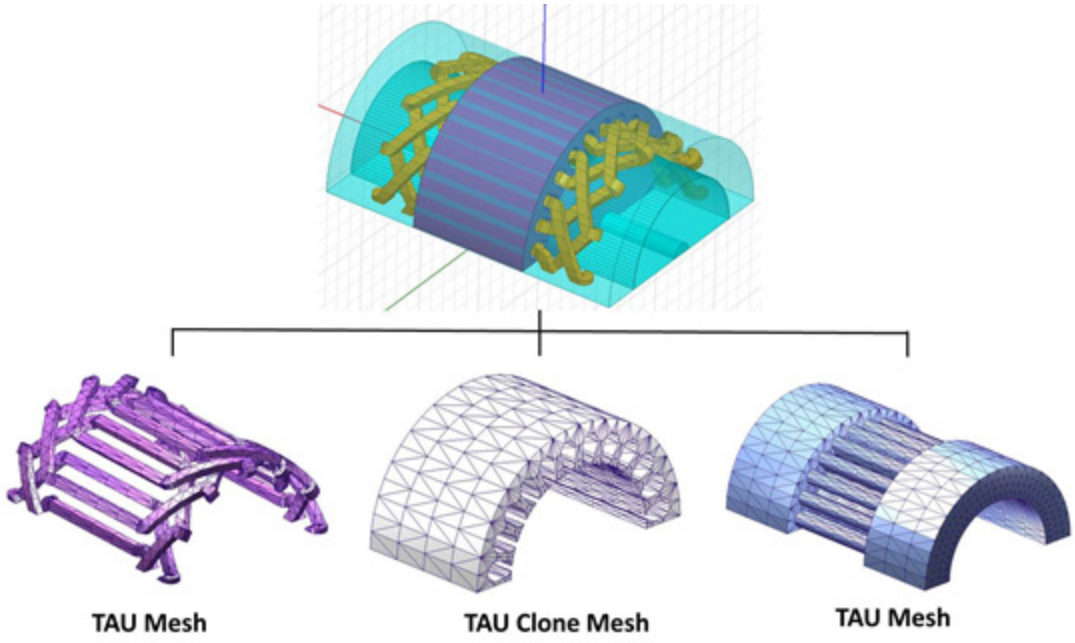
The volume mesh in the master region is then duplicated and translated to the child regions with copy and paste technology in TAU to complete the 3D clone region mesh.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3D TAU Mesh

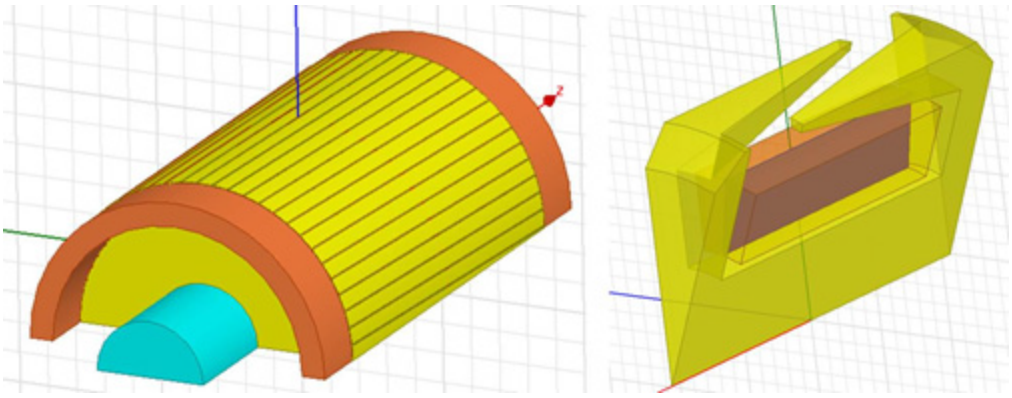
Beyond the 3D clone region, normally the regions have skewed coils. Therefore, the TAU regular mesher will be used to make the mesh.



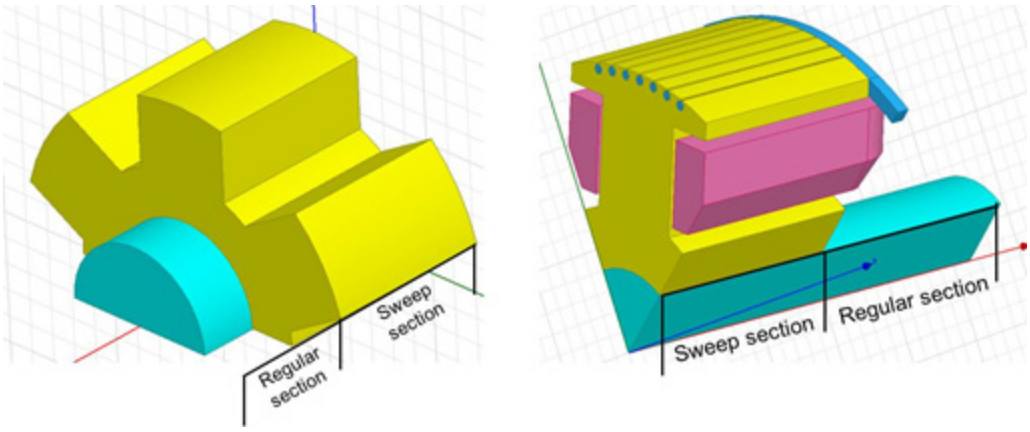
3D Sweep Mesh

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The rotational model for a moving region may have different structures. It may have all skewed coils and rotors. In this case, the TAU regular mesh will be used to make the mesh.

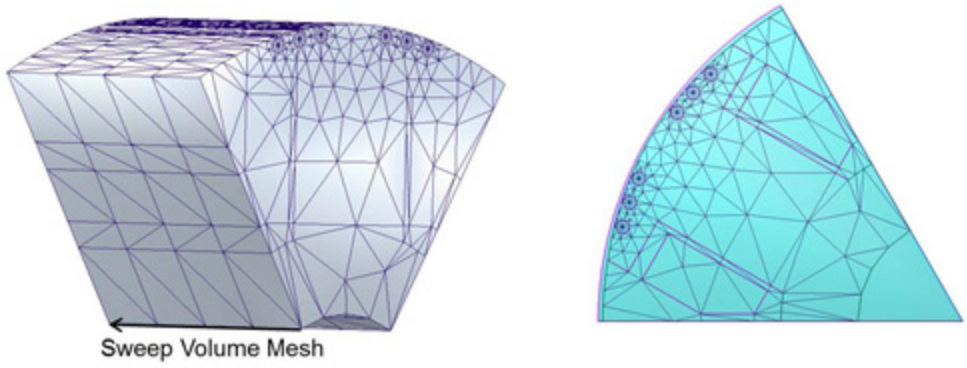


At the axis direction, the moving region can be classified into several specific regions. If the region has skewed objects, it is called a “regular section,” and the TAU regular mesh is applied. If the region can be geometrically sweeping at the axis direction, the region is called a “sweep region,” and the TAU sweep mesh method is applied.

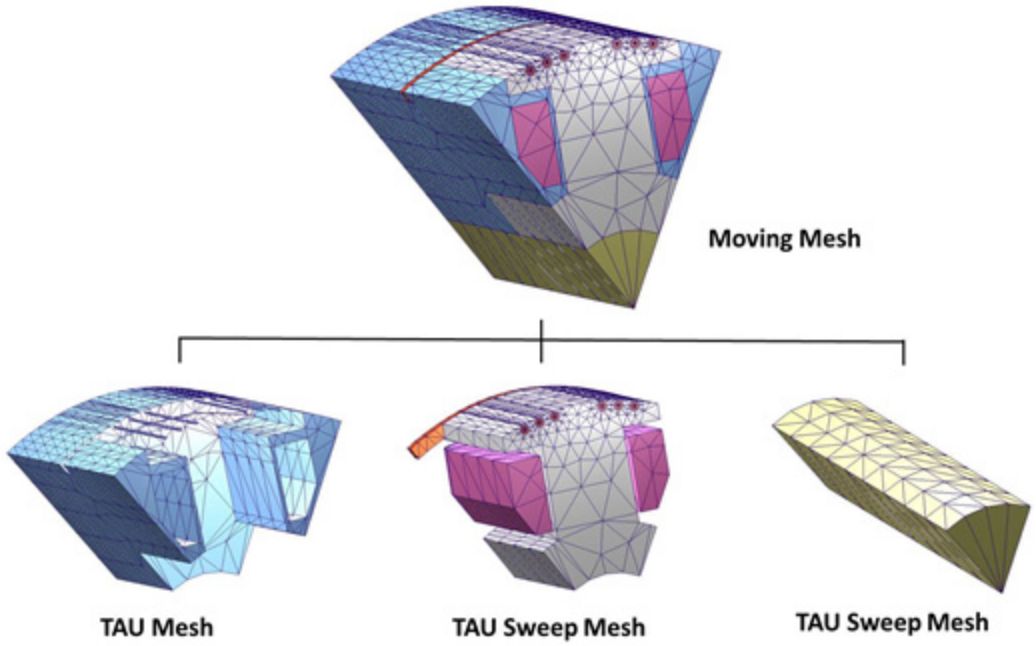


For the “sweep section,” a surface mesh is generated at the side, then the volume mesh is made by sweeping the surface mesh in the axis direction. This ensures a uniform mesh in the rotational direction, while maintaining the long aspect ratio in the axis direction to keep the mesh size small.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The volume mesh for the rest of the regions is made by the TAU regular mesher.



Band Mapping Angle

Because mesh mapping errors can occur at the interface between the band of motion and the stationary objects when mesh segmentation is performed, the band mapping angle allows these errors to be corrected by making the rotating angle per time step equal to the angle of each mesh segment in the direction of rotation. Thus, the band mesh will match the mesh on the stationary region for each time step. Reducing the mesh mapping error also reduces the noise for the torque calculation and improves simulation accuracy.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Band Mapping Angle** setting on the Cylindrical Gap Mesh Operation dialog box, enabled when **Clone Mesh** is selected, supports 3D transient designs for which:

- The band of motion is a true surface.
- The model is not skewed.
- The model is suitable for clone mesh – that is, the model can be segmented into identical sections in the direction of rotation.

The recommended band mapping angle (the angle the rotor rotates in one time step) typically equals the rotational speed multiplied by the time step. For example, if the rotational speed is 1.5 degrees per second and the time step is 2 seconds, then the band mapping angle should be 1.5 x 2 = 3 deg.

The starting position of the simulation should also be considered. In the above example, if the starting position is 0 deg or 3 deg, then 3 deg would be a good band mapping angle. If the starting position is 1.5 deg, the mesh will not be mapped at the starting position, in which case 1.5 deg is the recommended setting.

Another approach for determining the band mapping angle is to start with the model. For example, assume that the stator has 90 slots in the full model, with the solution domain angle equaling 1/6 of the full 360 degree model or 15 slots for 60 mechanical degrees. If you would like to solve with each slot having 8 mesh segments, there would be 120 total mesh segments (15 slots x 8 segments per slot). In this case, the band mapping angle would be 60 (deg) /120 = 0.5 deg. To ensure rotation through one mesh segment per time step, assuming the rotational speed is 1200 rpm, the time step should be set as:

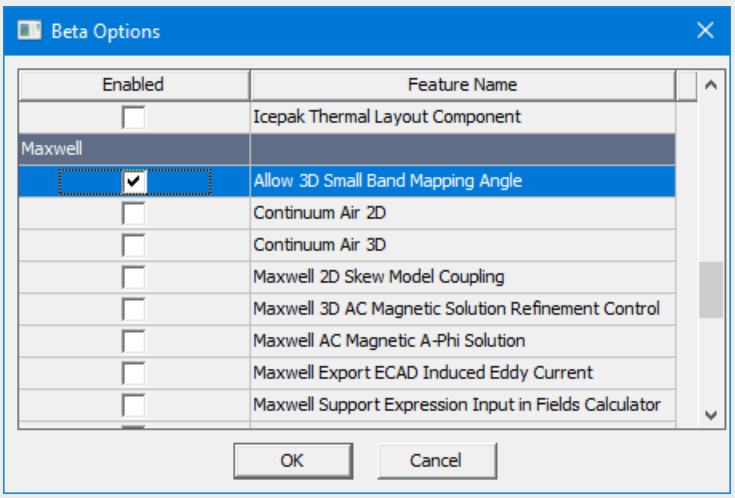
$$Timestep = \frac{1}{(speed_{rpm}/60seconds) \times TotalSegmentNumber \times (360°/DomainAngle°)}$$

$$Timestep = \frac{1}{(1200/60) \times 120 \times (360/60)} = \frac{1}{14400} seconds$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

For band mapping angles less than 0.5 degrees, clone meshing may improve with the **Allow 3D Small Band Mapping Angle** beta feature. This feature is accessed from **Tools** > **Options** > **General Options** > **Beta Options** button, which opens the following window:



TAU2D: Band Mapping Angle Mesh (Clone+Symmetry)

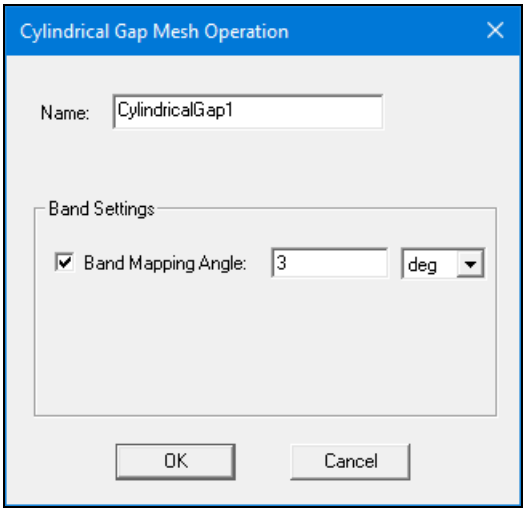
Band Mapping Angle is a new meshing option for transient simulation of motors in Maxwell 2D. When selected, the meshing process will ensure that each mesh segment on the outside of the band (the curve that separates the static and moving regions) has exactly the sweep angle that has been specified by the user. For example, for a quarter model with 90 degrees of matching boundaries, specifying a band mapping angle of 0.5 degree will result in a band outline having 180 mesh segments of equal length.

Because the meshing angle is determined by this setting, the band object must have a curved outline, using zero segments.

To use the 2D Band Mapping Angle function:

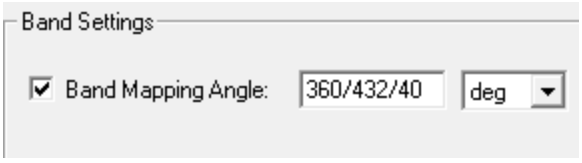
1. When you open a 2D transient project, select **Maxwell 2D** > **Mesh** > **Assign Mesh Operation** > **Cylindrical Gap Treatment**, which opens the **Cylindrical Gap Mesh Operation** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

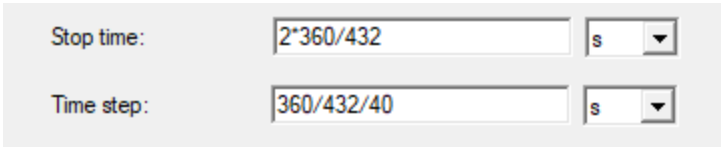


2. Select **Band Mapping Angle**, set the desired mapping angle, then click **OK**.

   Note that it is best to synchronize the transient simulation timestep with the band mapping angle. For example, suppose the following mapping angle is used to accurately model the cogging torque:



   If the motor's rotational velocity is 1 degree/second, then the following timestep should be used:

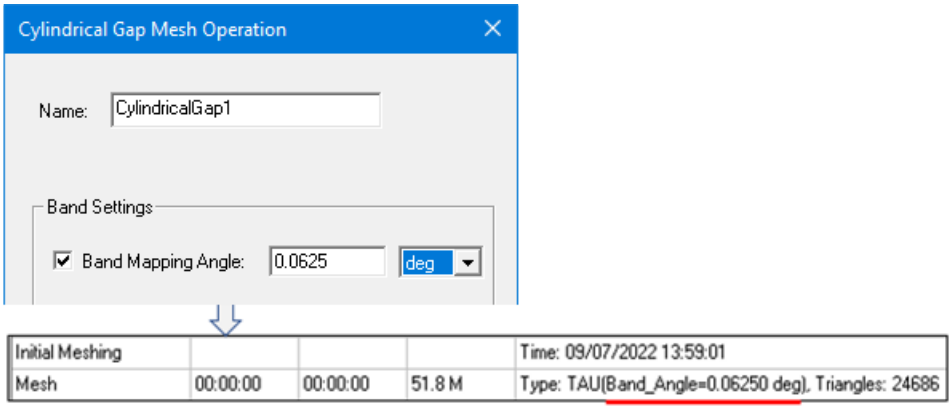


   This practice will help to remove numerical noise in the results.

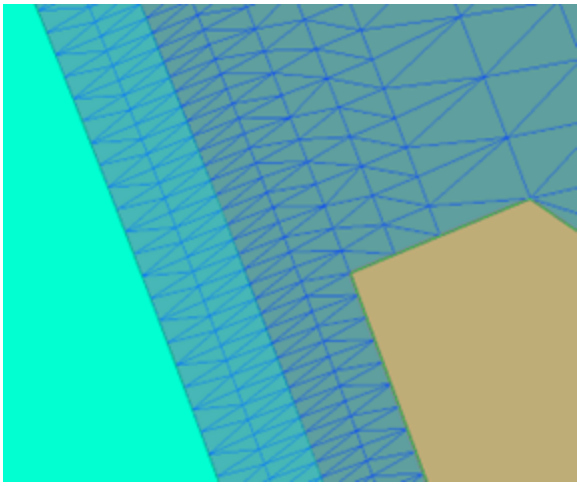
There are two ways to verify that the band mapping angle has been applied to a simulation:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

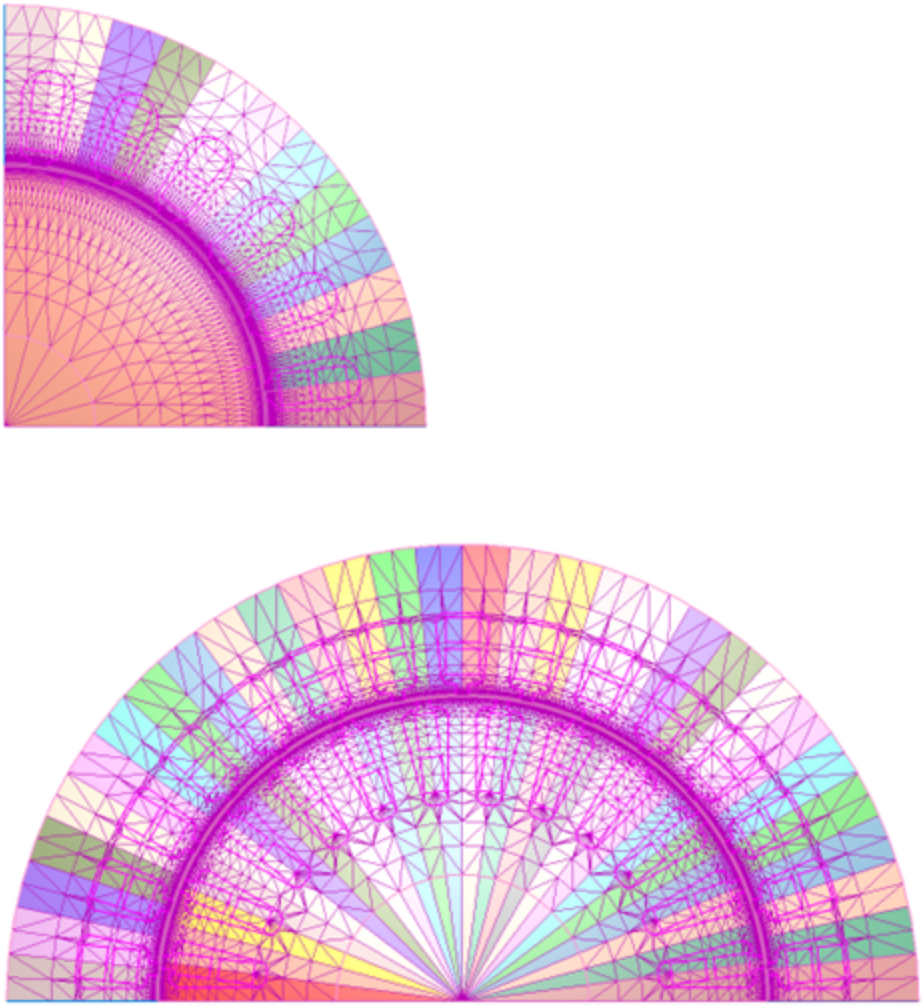
- The specified angle will appear under Initial Meshing, on the Mesh line after “TAU”:



- The effects of the mapping angle will be seen in mesh plots:
  - An additional layer will be created on both sides of the band.
  - The triangle orientation will be exactly the same on each side of band, to eliminate noise.
  - The layered structure will extend to the moving and static regions.
  - Mesh density will smoothly transition from fine to coarse with distance from the band.
  - Clone mesh if the region is repeatable.
  - Mirror mesh if the region is symmetrical.
  - Sweep layer mesh if the region is not repeatable.



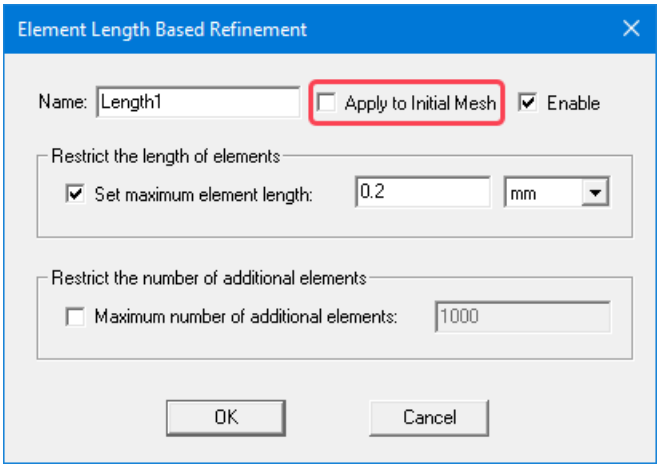
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



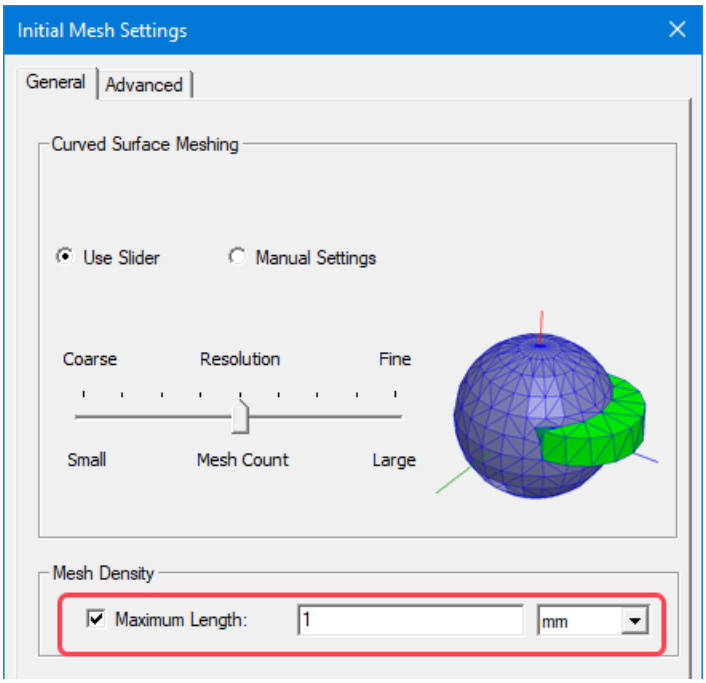
Notes

- This feature is intended to replace the use of wire bodies to guide meshing in the band. However, if wire bodies are included, the mesh will respect them along with the band mapping angle.
- Length-based mesh operations applied to an initial mesh (on objects, faces, or edges) are not yet supported. These mesh operations will be ignored by the mesher if a band mapping angle mesh is requested. However, these mesh operations can be used if **Apply to Initial Mesh** is **not** activated in the Element Length Based Refinement dialog box:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In addition, the global **Maximum Length** setting in the Initial Mesh Setting window can be used in conjunction with the Band Mapping Angle function to control mesh size.



- Eccentricity is not supported.
- Segmented circular edge is not supported.
- This feature is limited to Maxwell 2D transient models with a Cylindrical Gap mesh operation.
- The Band Mapping Angle mesh operation does not currently support models with translational motion.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Layers in Band Mesh *(Maxwell 3D Magnetostatic and Transient)*

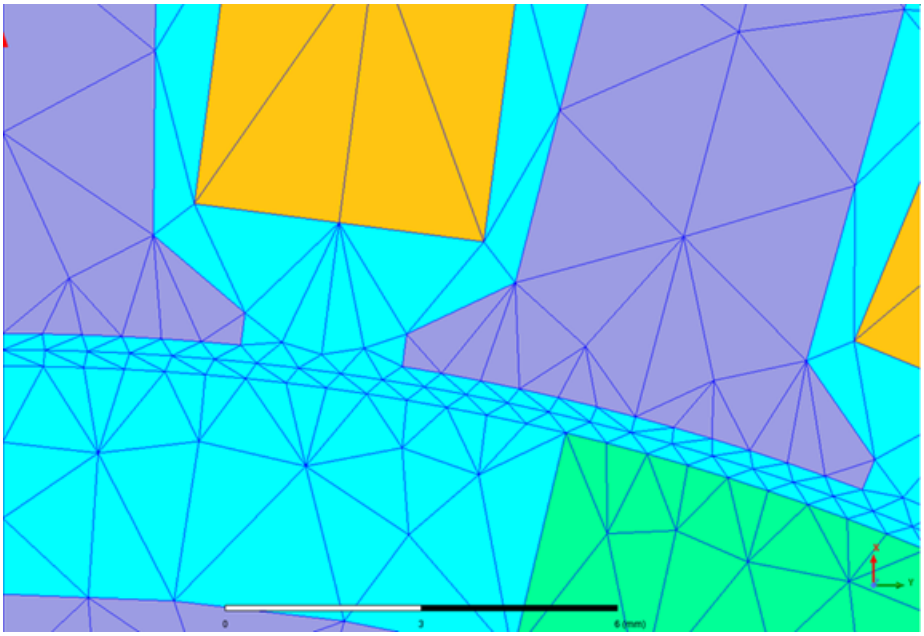
Maxwell rotating machine models need very precise cloned mesh to get accurate estimates of cogging torque. Thus, the mesh near the interface of the static side and moving side, *i.e.*, band surface, is crucial to the accuracy of this simulation result.

Clone mesh is an initial mesh process and currently provides one layer mesh between band and stator and one layer mesh between band and rotor. However, one layer may not provide sufficient accuracy for some designs. For such cases, multiple layers in band for clone mesh can be used to improve accuracy. By assigning this mesh operation, you can control the number of layers near the band on *both* the static and the moving side without losing the repeatability of the clone mesh.

> **Note:** This feature is applicable only to Maxwell 3D magnetostatic and transient solution types.

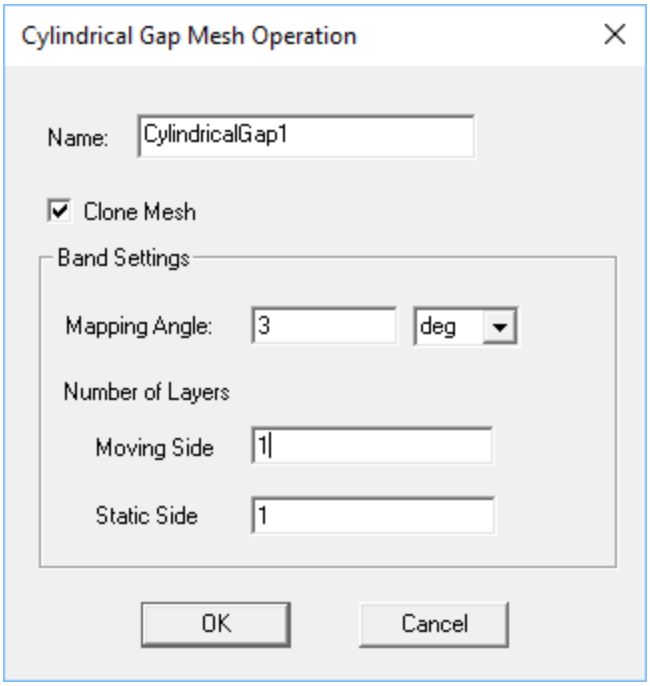
Example

Without multiple layers in the band mesh, when the model is cloned, the static side has clone geometry for which clone mesh is provided. The moving side does not have clone geometry so sweeping mesh is provided for it. This simple mesh type is shown below.

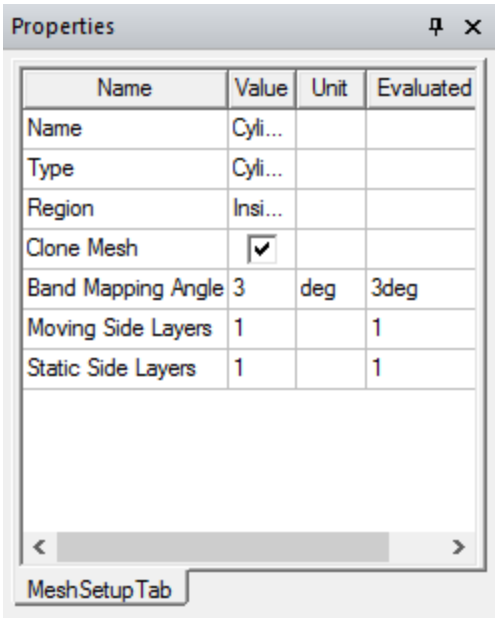


To improve accuracy of results, multiple layers can be specified for a **Clone Mesh** in the **Band Settings** section of the **Cylindrical Gap Mesh Operation** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

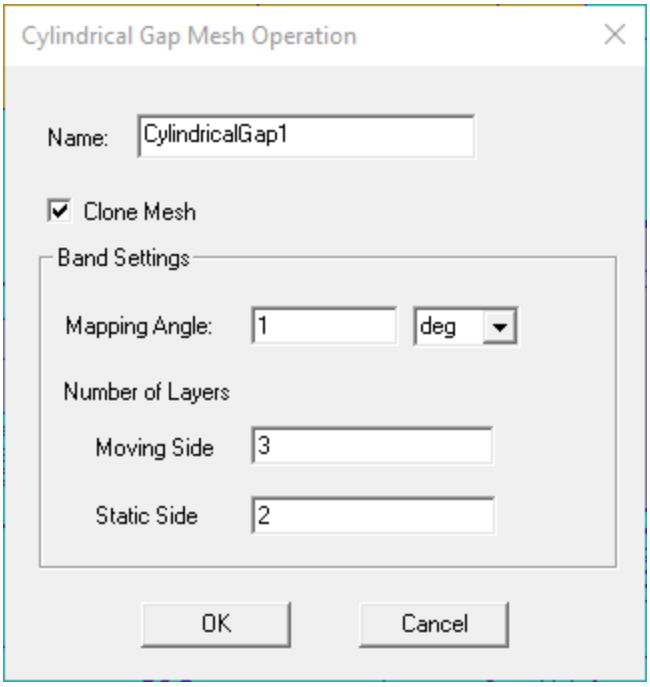


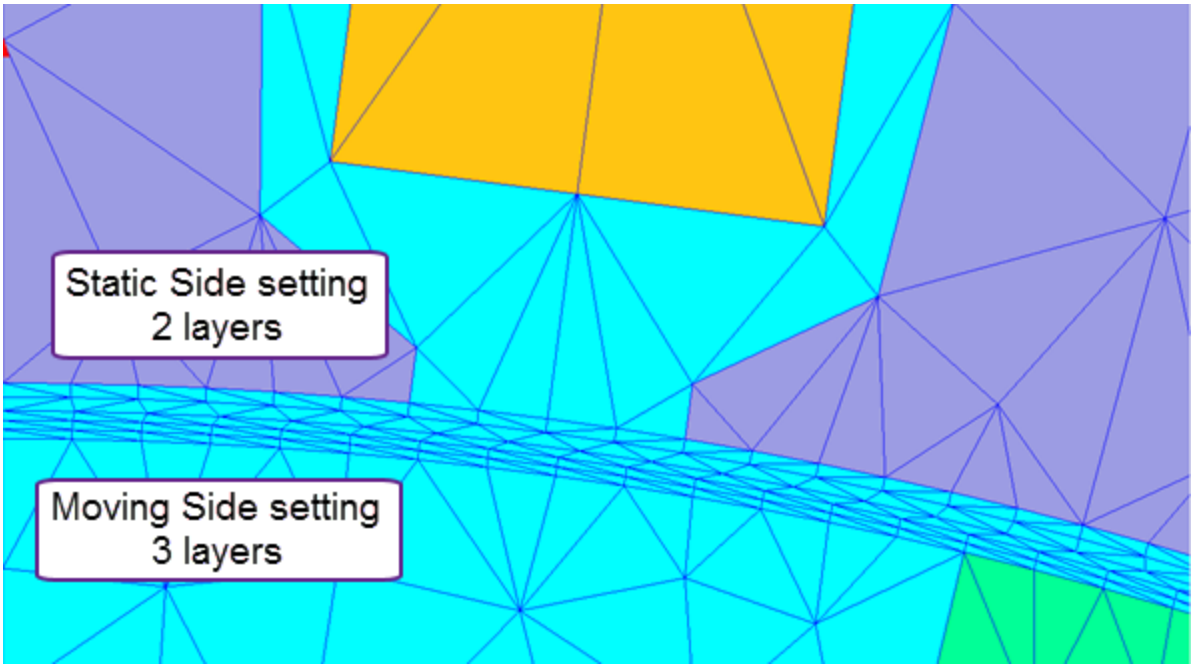
The layers options are also available in the **Properties** window's **Mesh Setup** tab when the cylindrical gap mesh operation is selected in the Project Manager tree.



By setting the **Number of Layers** for the **Moving Side** and/or the **Static Side** to integer values greater than 1, the number of band layer meshes will be increased accordingly.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



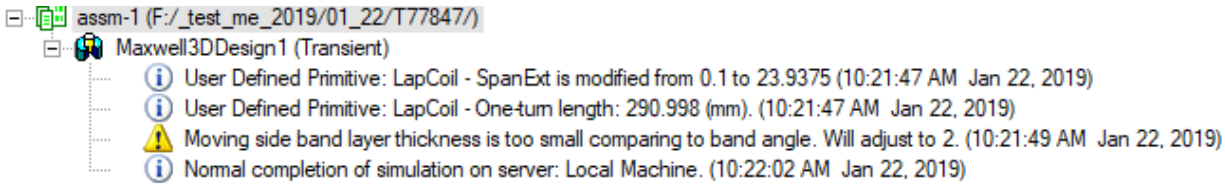


Considerations and Limitations

- If you specified a number of layers that would result in a lowering of mesh quality, clone mesh will adjust the number of layers, and give a warning message similar to the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

following:



- If the model it not suitable for clone mesh, or if clone mesh fails to mesh it, this setting will not be respected in TAU regular 3D mesh. A warning message will be given by mesh in this situation.
- Postprocessing seeding refinement will respect the layers in the mesh. Adding any postprocessing mesh operation will not destroy the layers; however, the clone mesh might be destroyed.

Clone Mesh Density

For Maxwell 3D rotational transient designs and Mechanical–Thermal designs of rotating machines, this meshing operation allows you to specify the clone mesh density while keeping the identical mesh in identical sub-regions. Different from Length-Based Mesh Refinement, this Clone Mesh Density mesh operation will only be applied to clone mesh. Only when clone mesh is enabled under cylindrical mesh operation, mesh will respect this mesh operation. Mesh density can be controlled while identical mesh is maintained.

To control the clone mesh density:

1. Select the object.
2. Click **Maxwell** > **Mesh Operations** > **Assign** > **Inside Selection** > **Clone Mesh Density**.

   The **Clone Mesh Density** dialog box appears.
3. Type a name for the mesh operation in the **Name** box, or accept the default name.
4. To specify the radial direction density, enable the radial direction option. Type the maximum length of the element size required inside the object(s) in the **Maximum element length** box, and select the units. Maxwell refines the element edges inside the object(s) until they are equal to or less than this value. The default value is set to 10% of the maximum dimension of the bounding box of the model.
5. To specify the axial direction density, enable the axial direction option. Enter the **Number of layers** to be added. Maxwell refines the mesh in axial direction inside the object(s) until the number of mesh layers in the axial direction are equal to or more than this value.
6. Click **OK**. You need to specify mesh density in at least one direction to enable this mesh operation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

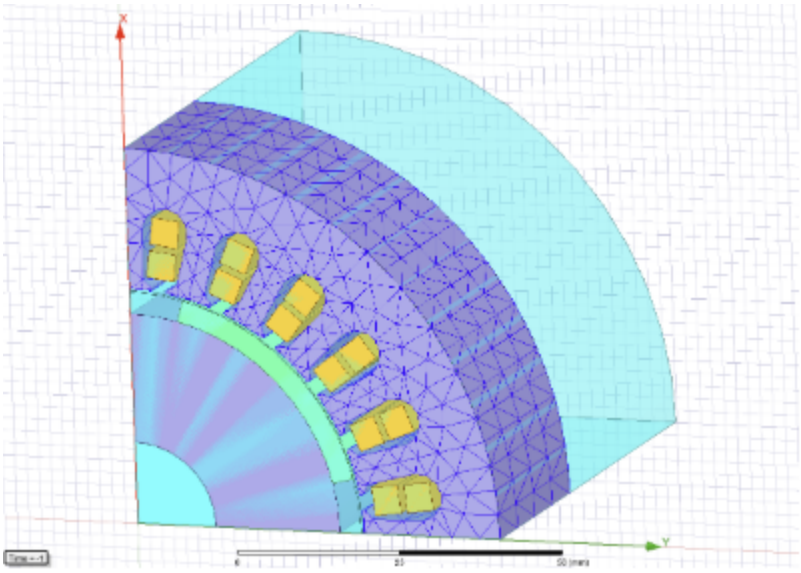
When the clone mesh is generated, the refinement criteria you specified are used. You can specify different maximum length of the element size on different objects. Note that identical density setting should be applied to identical objects to ensure the specified density to be applied on those objects.

You can also specify different axial direction number of layers on different objects. Note that the number of layers in axial direction are maintained separately inside and outside of the band. If objects share a same region in axial direction on the same side of the band, the mesh refinement will refines until the layer thickness equal or smaller than the smallest average layer thickness in this region.

The Clone Mesh Density can not be applied to band object. To control the element size on band, use the **Band Mapping Angle** setting.

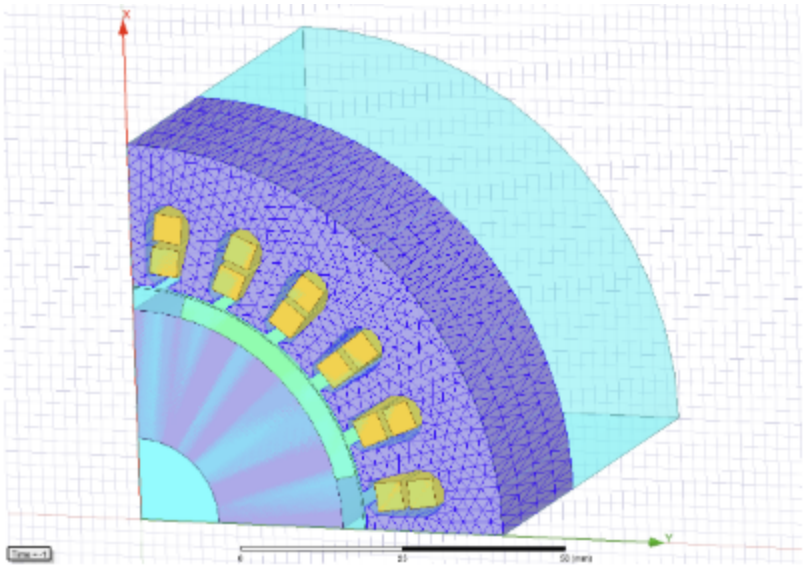
Clone Mesh Density can only be respected in clone mesh region. Mesh density in regions that are skewed or not clonable may not be controlled as precise as it is in clone region.

Original clone mesh without Clone Mesh Density mesh operation:

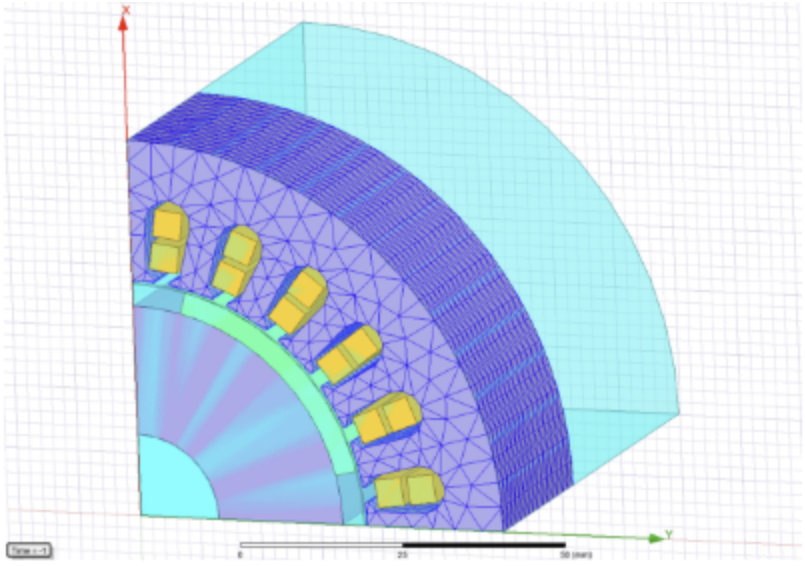


Clone mesh with Clone Mesh Density mesh operation on Stator. Radial direction mesh is refined according to maximum element length specification:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Clone mesh with Clone Mesh Density mesh operation on Stator . Axial direction mesh is refined according to number of layers specification:



Skin Depth Layers Setting for 2D

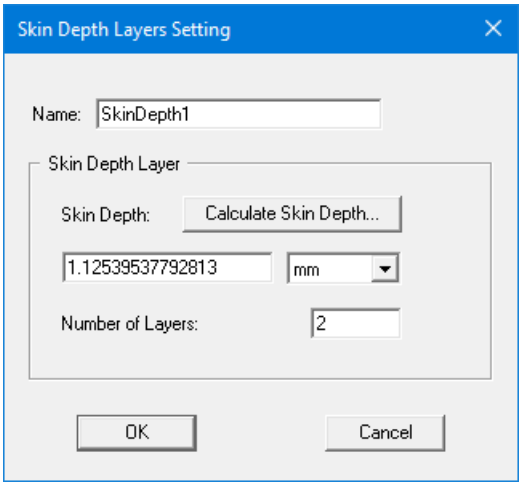
You can apply these settings only to the edges of sheet bodies in Maxwell 2D when the TAU mesher has been selected in the **Initial Mesh Settings** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Any changes you make using these settings will invalidate the initial mesh, as well as the current mesh and solutions, if any.

1. Select the edge of a sheet body, and click **Maxwell 2D** > **Mesh** > **Assign Mesh Operation** > **On Selection** > **Skin Depth Based...** to open the **Skin Depth Layers Setting** dialog box:

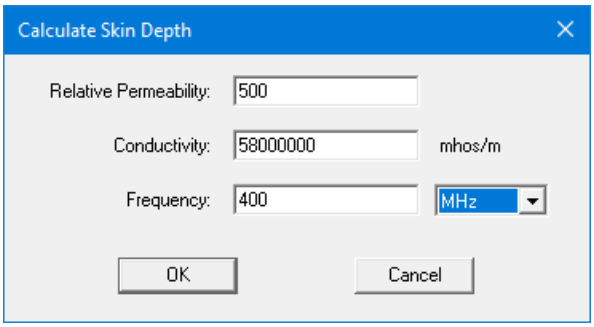


2. Type a name for the mesh operation in the **Name** text box or accept the default name.
3. Enter the skin depth in the **Skin Depth** text box.

   Alternatively, calculate the skin depth based on the object's material permeability and conductivity and the frequency at which the mesh will be refined:

   a. Click **Calculate Skin Depth**.

      The **Calculate Skin Depth** dialog box appears.



   b. Enter the material's **Relative Permeability** and **Conductivity**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Specify the **Frequency** at which to refine the mesh.
d. Click **OK**.

   Maxwell calculates the skin depth and enters its value in the **Skin Depth** text box.

4. In the **Number of Layers of Elements** text box, type the number of layers to add perpendicular to the object's surface.

Maxwell will add an equivalent number of mesh points to each layer. For example, if Maxwell added 10 points to satisfy the **Surface Triangle Length**, it will add 10 points to each layer.

> **Note:**
>
> Caution should be used with this mesh operation, as very thin layers may cause a reduction in mesh quality or unnecessarily cause the generation of a very large mesh. Furthermore, regions refined under this operation and its close neighbors do not participate in solution adaptive refinement. This is another reason to use this seeding operation with caution.

Modifying Surface Approximation Settings

Each solver applies surface approximation settings for selected objects when it generates the initial mesh. If you modify a solver's default settings after the initial mesh has been generated, they will not affect the mesh for that design variation. Surface approximation makes sense for curved surfaces, for which the mesh will not exactly reproduce the surface shape. It also can be used to restrict the aspect ratio of triangles on planar surfaces. For a given surface approximation slider setting, the approximation is more refined for IE and SBR+ solvers than for FEM solvers.

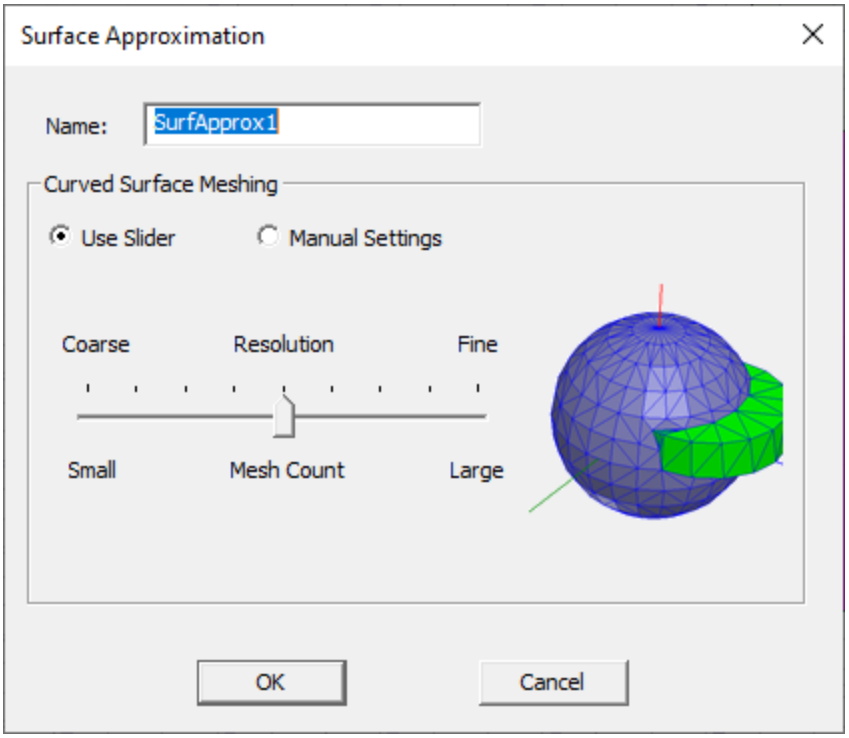
1. Select the faces for which you want to modify the surface approximation settings.

   Alternatively, select an object if you want to modify the surface approximation settings of every face on the object.

2. Click **Maxwell** > **Mesh** > **Assign Mesh Operation** > **Surface Approximation** or right click on **Mesh** in the Project Manager and select **Assign Mesh Operation** > **Surface Approximation**.

   The *Surface Approximation* dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

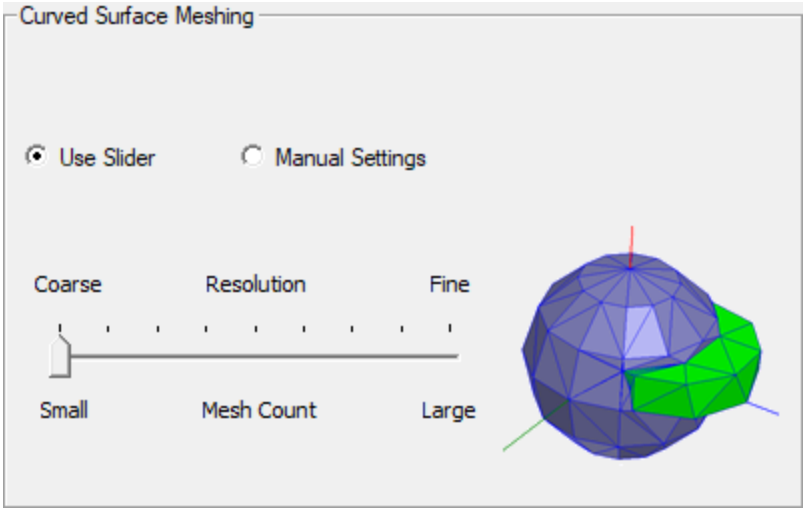


3. Type a name for the group of settings in the **Name** text box or accept the default name.
4. Under **Curved Surface Meshing**, you can select **Use Slider** or specify **Manual Settings**.

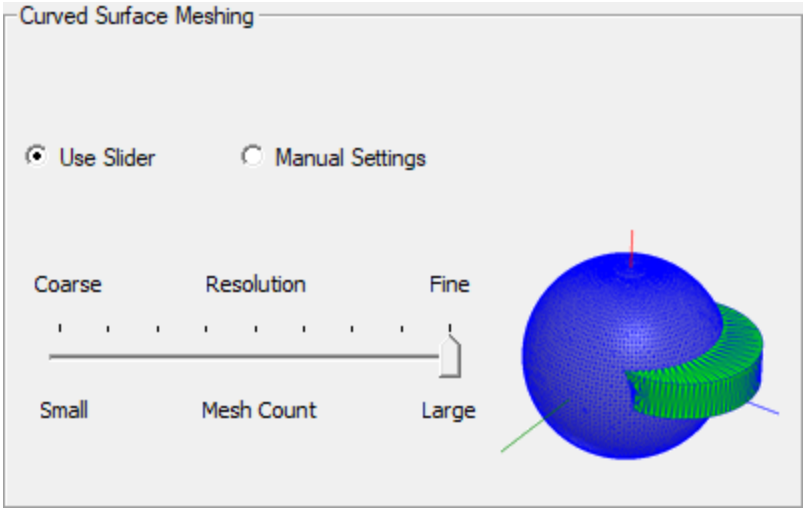
   The slider includes a visual representation of your choice, ranging from a ***Coarse*** *Resolution* (with a ***Small*** *Mesh Count*) through a nine position scale to a ***Fine*** *Resolution* (with a ***Large*** *Mesh Count*). Here, "*Mesh Count*" refers to the number of elements comprising the mesh.

   Coarse Resolution, Small Mesh Count:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

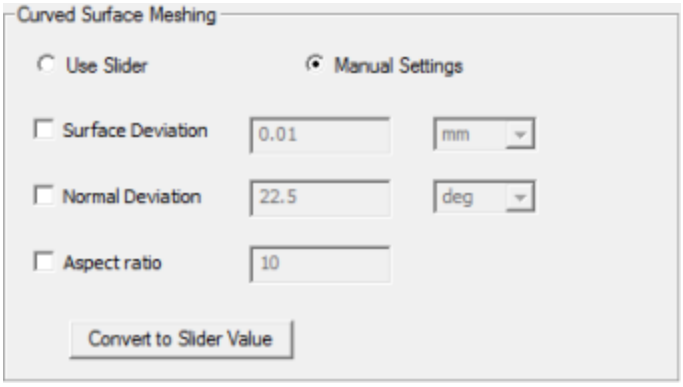


Fine Resolution, Large Mesh Count:



5. If you choose **Manual Settings**, the dialog box changes to show text fields.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> If you selected **Use dynamic surface resolution** in Initial Mesh Settings, use of **Manual Settings** is not permitted. You must use slider bar to specify one of three levels of surface representation: coarse (1-3), normal (4-6), and fine (7-9).

6. Select **Surface Deviation**. Use the drop-down menu to select a unit of measure and then enter the distance between the true surfaces of the selected faces and the meshed faces.
7. Select **Normal Deviation**. Use the drop-down menu to select a unit of measure and then enter the angular distance between the normal of the true surface and the corresponding mesh surface.
8. Select **Aspect Ratio** and then type a value in the text box. This value determines the shape of the triangles. The higher the value, the thinner the triangles. Values close to 1 will result in well-formed, wide triangles.
9. Click **OK**.

   The settings are applied to the initial mesh generated on the selected surface. The group of settings is listed in the project tree under **Mesh**.

   You can also specify Initial Mesh Settings to apply to all objects. However, if you apply separate surface approximation mesh operations to specific objects, the object settings take precedence over the general setting.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Selecting **Use dynamic surface resolution** in the Initial Mesh Settings specifies the best-practice mesh operations over the geometric models. The default mesh operations or user-defined mesh operations may or may not be replaced by optimized mesh operations with model analysis. In general, large curve faces, curve faces with small gaps, or skewed cables get smoother curvature representation while small curve faces, such as fillets and small curve objects, get relatively coarse triangulations to reduce overall mesh size.

This dynamic surface resolution mesh operation supports 3D volume mesh and surface mesh in all products.

Surface Approximation Settings

Object surfaces in Maxwell may be planar, cylindrical or conical, toroidal, spherical, or splines. The original model surfaces are called *true surfaces*. To create a finite element mesh, Maxwell first divides all true surfaces into triangles. These triangulated surfaces are called faceted surfaces because a series of straight line segments represents each curved or planar surface.

For planar surfaces, the triangles lie exactly on the model faces; there is no difference in the location or the normal of the true surface and the meshed surface. When an object's surface is non-planar, the faceted triangle faces lie a small distance from the object's true surface. This distance is called the *surface deviation*, and it is measured in the model's units. The surface deviation is greater near the triangle centers and less near the triangle vertices.

The normal of a curved surface is different depending on its location, but it is constant for each triangle. (In this context, "normal" is defined as a line perpendicular to the surface.) The angular difference between the normal of the curved surface and the corresponding mesh surface is called the *normal deviation* and is measured in degrees.

The *aspect ratio* of triangles used in planar surfaces is based on the ratio of circumscribed radius to the in-radius of the triangle. It is unity for an equilateral triangle and approaches infinity as the triangle becomes thinner.

You can modify the surface deviation, the maximum permitted normal deviation, and the maximum aspect ratio of triangles settings on one or more faces at a time in the **Surface Approximation** dialog box. (Click **Maxwell** > **Mesh Operations** > **Assign** > **Surface Approximation**.)

The surface approximation settings are applied to the initial mesh.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> For the initial mesh, all the vertices of the triangles lie on the true surfaces. During mesh refinement all the points are first added to the faceted surface for all products, not to the true surfaces. Then, for all Maxwell projects and for curvilinear meshing enabled HFSS projects , the points on the faceted surfaces will be moved to the curved surface locations when such movement does not degrade the quality of the mesh.

Guidelines for Modifying Surface Approximation Settings

If you intend to modify the surface approximation settings for an object face or faces, keep the following guidelines in mind:

- When necessary, override the default surface approximation settings to represent curved surfaces more accurately. More accurate representation will increase the mesh size and consume more CPU time and memory. The default settings are adequate for most circumstances.
- If you want to obtain a faster solution by using a cruder representation of curved surfaces, set the coarser setting for the whole object, not just a single face.
- It is difficult to satisfy aspect ratio demands if the aspect ratio value is set close to 1 because an arbitrary shape cannot be filled with only equilateral triangles. Therefore, setting the aspect ratio to 1 can lead to unreasonably large meshes. The aspect ratio is limited to 4 for planar objects and 1.2 for curved objects.

Applying Curvilinear Elements

For **HFSS**, **Maxwell**, and **Mechanical** designs, Curvilinear elements provide an accurate representation of the CAD model geometry by improving the conformity of element nodes to curved CAD surfaces. You can apply curvilinear elements as a Mesh Operation on selected curved model faces see procedure below or as a global option in the Initial Mesh Settings dialog box.

For **Maxwell** designs:

When curvilinear elements are applied globally or to specific CAD faces, the meshing process attempts to pull corner nodes of elements added during mesh refinement to the curved CAD faces. The following figures explain the effects with and without curvilinear elements applied:

- With the **Apply Curvilinear Elements** option *not* selected:

  Corner nodes of an initial coarse mesh are located on curved CAD surfaces. However, as the mesh is refined, the corner nodes of any additional elements remain along the plane of the initial mesh's element faces. Therefore, the number of facets along a curved face (in

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the circumferential direction) does not increase, and the facets remain coarse. This effect is shown in the first mesh plot below. Note the red-circled nodes. These nodes were added as a result of mesh refinement, and they were not pulled to the curved CAD face:

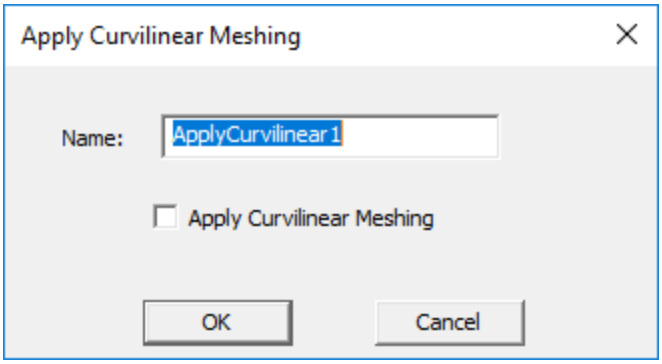
- With the **Apply Curvilinear Elements** option *selected*:

  The meshing process will attempt to pull all corner nodes of additional elements added during mesh refinement to the curved CAD faces. Therefore, the facets along curved faces are finer, and the conformity of the mesh to the curved CAD faces is improved. The effect is as shown in the following, second mesh plot:

How to Apply Curvilinear Elements to Specific Curved Faces:

1. Select the object or face to which you want to apply curvilinear elements.
2. Select **Mesh** in the Project Manager, right-click for the short cut menu and select **Assign Mesh Operation** > **Apply Curvilinear Meshing**.

   The **Apply Curvilinear Meshing** dialog box appears.



3. Either accept the default name or provide one of your own.
4. Select **Apply Curvilinear Meshing**.
5. Click **OK**.

The defined Mesh Operation appears in the Project Manager.

Modifying the Model Resolution

(Setting Model Resolution applies to Maxwell 3D Designs only)

The model resolution parameter is used by the mesh generator to distinguish large features from small features in the model. This setting controls how large a feature must be to be resolved by the mesher. For example, if you set the model resolution length to 20 mm, any model features smaller than 20 mm are not represented in the mesh. Neither the model nor the model files are changed. The resolution only controls how the mesh for the model is represented.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

While removing small details, if the mesh generator finds that the representation of the model is not accurate enough, it returns an error condition. The mesh generator then starts with the most accurate representation and prunes away the details smaller than the model resolution length. It returns an error if the specified model resolution length forces the final representation to deviate too greatly from the model.

You can set Model Resolution on one or more objects to remove unnecessary details from the mesh representation. This can be used to reduce the mesh complexity of the selected objects.

A cautionary note is needed concerning the use of model resolution. It can sometimes make model faces fail to be represented in the mesh, which can effectively remove some of your boundary conditions. This can only happen if faces are closer together than the model resolution distance you specify.

To modify the model resolution:

1. Select the object or objects on which to specify a Model Resolution operation.
2. Click **Maxwell 3D** > **Mesh** > **Assign Mesh Operation** > **Model Resolution**.

   The **Model Resolution Mesh Operation** dialog box appears.

   Alternatively, you can display the same dialog using these steps:

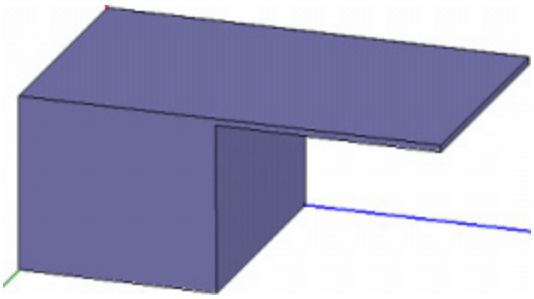
   a. Right-click on **Mesh** in the Project Tree to display the shortcut menu.
   b. Click on **Assign Mesh Operation** > **Model Resolution** on the shortcut menu.

   The **Model Resolution Mesh Operation** dialog box contains text fields for the mesh operation Name and radio buttons with choices for the following

   - **Auto Simplify Using Effective Thickness** – The mesher calculates the resolution length based on each object's effective thickness. One mesh operation can be assigned to many objects, and each will be simplified based on its own dimensions. Use the **Auto Simplify** selection:
     - To remove many details while retaining an object's overall shape and size
     - For objects of generally uniform thickness
     - To assign one mesh operation to many objects
   - **Use Model Resolution length** – Enables fields for you to specify the resolution value and units. Use this selection for:
     - Tighter control of mesh accuracy.
     - Objects of non-uniform thickness. For example, the thin section of the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

object shown below might be lost with **Auto Simplify**.



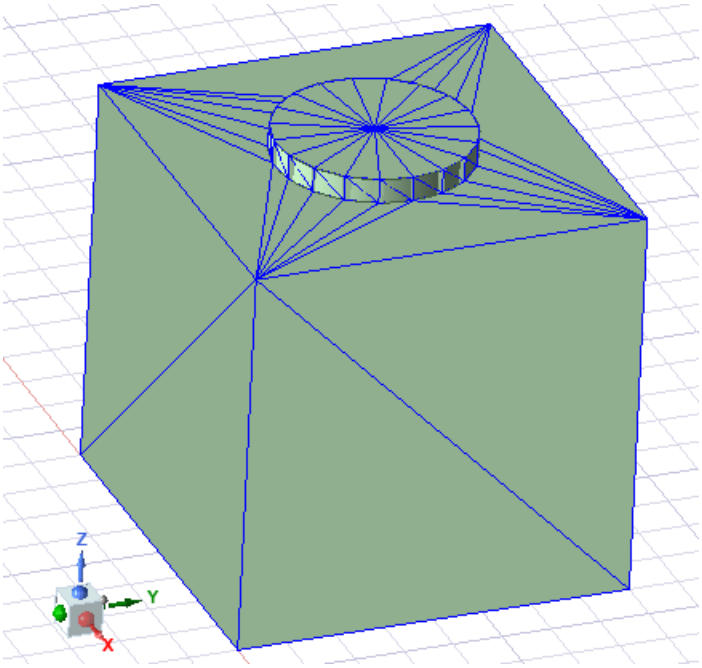
3. After defining the operation, click **OK**.

   This adds the named Model Resolution operation under the **Mesh** icon in the Project Manager tree.

> **Note:** Setting Model Resolution will invalidate any existing solutions. When two objects in contact have different model resolution lengths, the smaller length will apply for the common regions.

Use Model Resolution Length

To illustrate how the model resolution length can be used, a meshed model of a box that has a small cylindrical feature on the top surface is shown in the following figure.



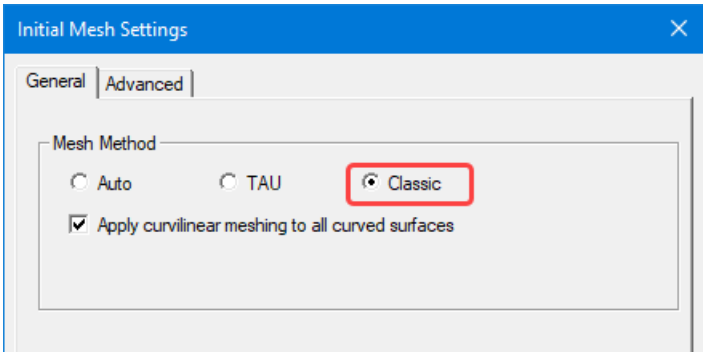
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you want to exclude the cylinder from meshing, specify an appropriate value for **Model Resolution Length** such that it is smaller than the cylinder diameter, but bigger than the cylinder height.

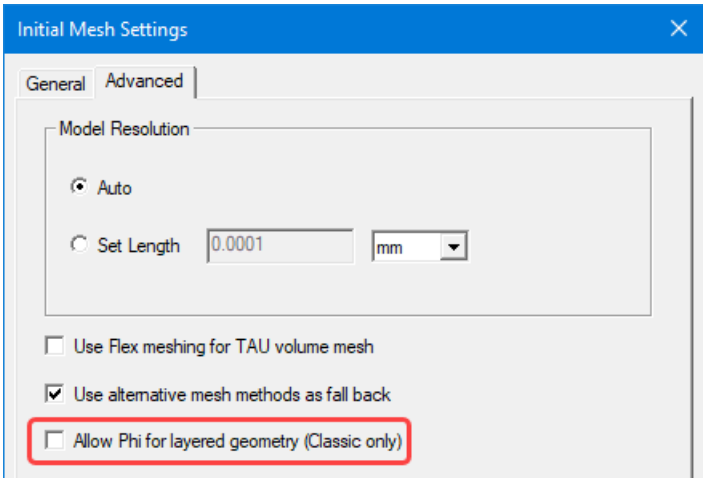
> **Warning:** The Model Resolution Length function is not supported for the Phi mesher or fully supported for the TAU mesher.

To assign **Model Resolution**, follow these steps:

1. Press **Ctrl+A** to select the whole model.
2. Right-click **Mesh** and select **Initial Mesh Settings**.
3. From the **General** tab, select **Classic**.



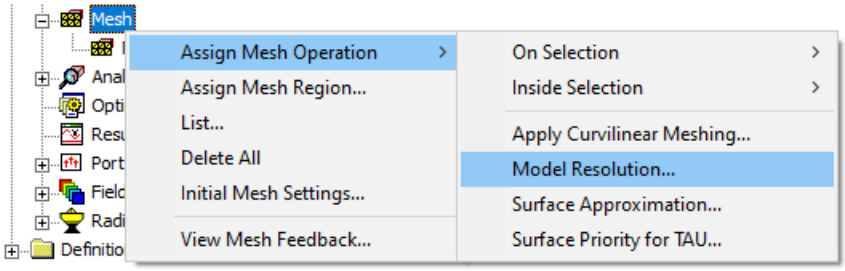
4. Click the **Advanced** tab, and make sure that **Allow Phi for layered geometry (Classic only)** is **unchecked**.



5. If you know the desired Model Resolution Length, you can set it in the **Advanced** tab of the Initial Mesh Settings window before generating the mesh.

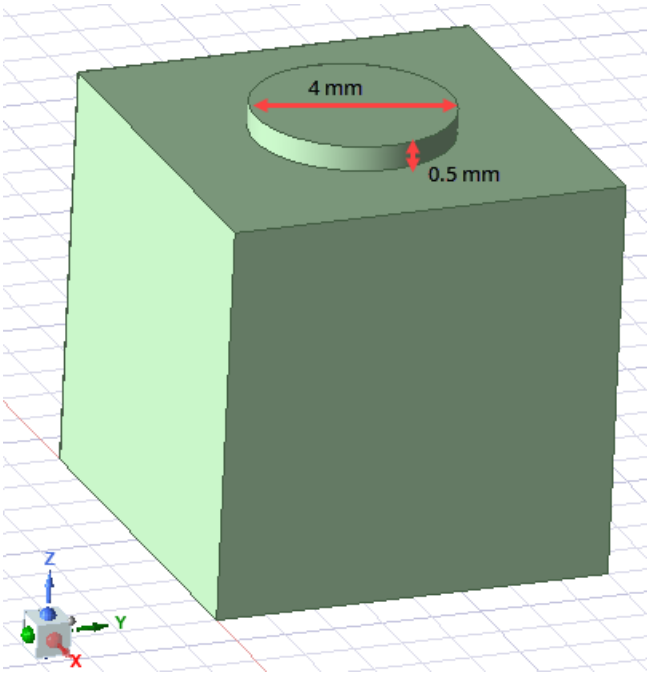
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

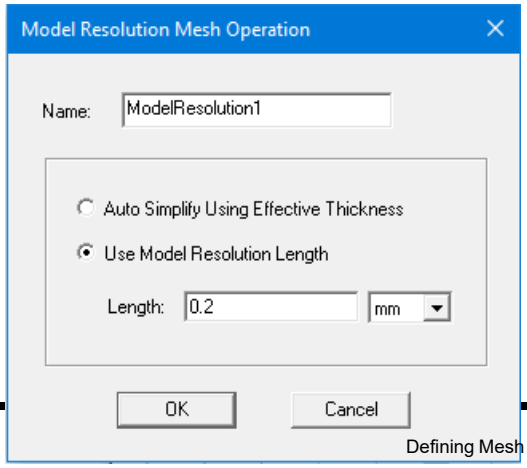
6. If you have already generated the mesh and need to modify the Model Resolution Length, right-click **Mesh** and select **Assign Mesh Operation** > **Model Resolution**.



The **Model Resolution Mesh Operation** dialog box appears.

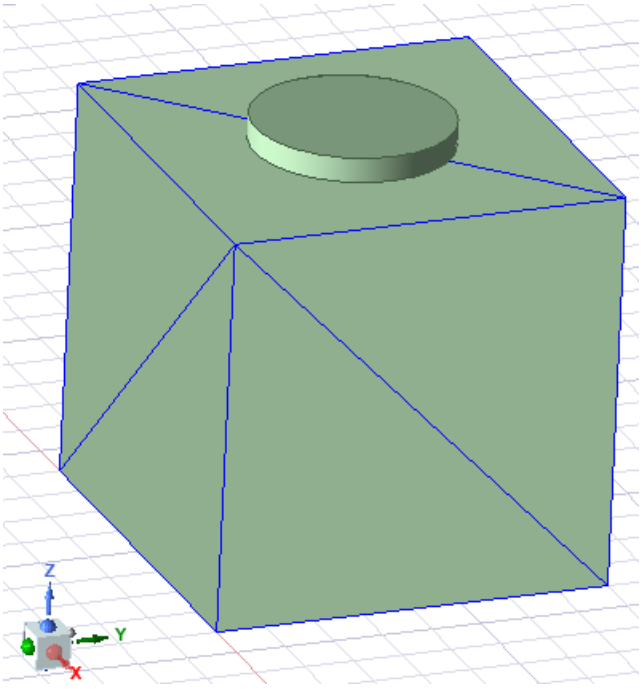
7. Set the value of the length smaller than the cylinder diameter, but bigger than the cylinder height, and click **OK**. In this case, we set it to 0.2 mm.





Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The meshed model excluding the cylinder looks as shown in the following figure.



Reverting to the Initial Mesh

The initial mesh is the mesh that is generated the first time a design variation is solved. It includes surface approximation settings, but does not include lambda refinement or defined mesh operations.

If you have modified the design setup, and do not want to use the existing current mesh, revert to the initial mesh prior to solving.

- On the **Maxwell** > **Analysis Setup**> **Revert to Initial Mesh**.

Reverting to the initial mesh is useful when you want to evaluate how a different solution frequency affects the mesh generated during an adaptive analysis. You lose all solution data for a solve setup and all of its sweeps when you revert to the initial mesh for that setup. You can do this for all solve setups at once by selecting the command through Analysis in the menu system or project tree, or for a specific solve setup via its right mouse click menu in the project tree.

Generating Mesh without Solving

If you want to refine the mesh on a face or volume but do not want to generate a solution, define mesh operations. Then do one of the following:

- Select [**Maxwell**] > **Analysis Setup** > **Generate Mesh**.
- Right-click the **Analysis** or **Setup** icon in the **Project** window and select **Generate Mesh**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The same solve machine rules that apply to solving any other setup also apply here. The mesh operation will be sent to the default solve machine. If you specified "Prompt for analysis machine when launching analysis" under **Tools** > **Options** > **General Options** > **Analysis Options**, the **Server Setup** dialog box appears and you can specify a solve machine.

- If a current mesh has been generated, the solver will refine it using defined mesh operations.
- If a current mesh has not been generated, the solver will apply the mesh operations to the initial mesh.
- If an initial mesh has not been generated, the solver will generate it and apply the mesh operations to the initial mesh.
- If the defined mesh operations have been applied to the selected face or object, the current mesh will not be altered.

> **Tip:**
>
> Define a new mesh operation rather than modify an existing mesh operation. The solver will not re-apply a modified mesh operation.

> **Note:** Mesh operations at the target design of a mu (permeability) link are ignored.

Applying mesh operations without solving enables you to experiment with mesh refinement in specific problem regions without losing design solutions. You cannot undo the applied mesh operations, but you can discard them by closing the project without saving them or by reverting to the initial mesh.

TAU Flex Meshing

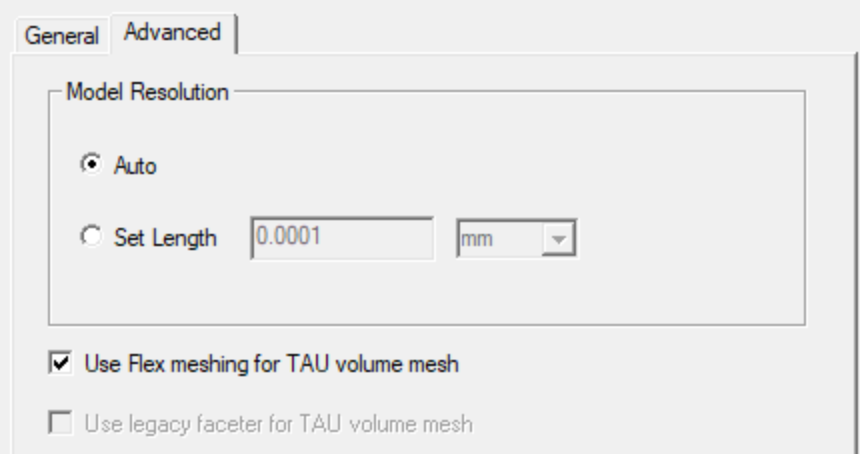
TAU Flex is designed for guaranteed mesh success, and may also reduce meshing time. It provides a mesh even for complex and “dirty” geometry model without any preliminary geometric healing or repair.

TAU Flex is a multi-domain, priority mesher which provides more efficient and reliable meshing. The priority mesh generation is driven by solver requirements for materials (metal/non-metal), boundary conditions (ports, radiation,...), conduction paths, and so forth. The constraint-tolerant meshing method gives the TAU Flex mesher flexibility to excuse mesh difficulties in lower-priority regions so that the resulting mesh can still provide acceptable simulation results.

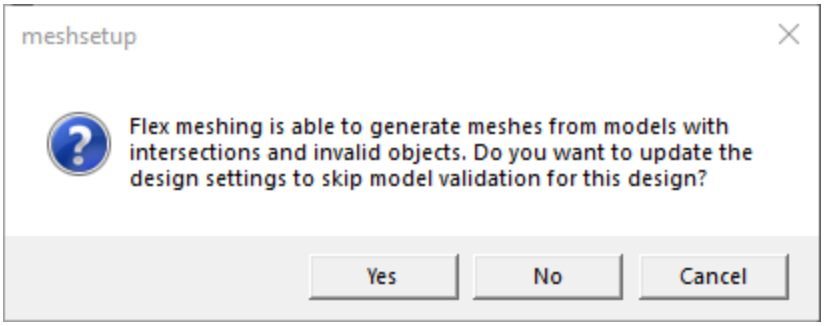
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Steps for Using TAU Flex

1. Navigate to the **Initial Mesh Settings** window.
2. Click the **Advanced** tab and select **Use Flex meshing for TAU volume mesh**.

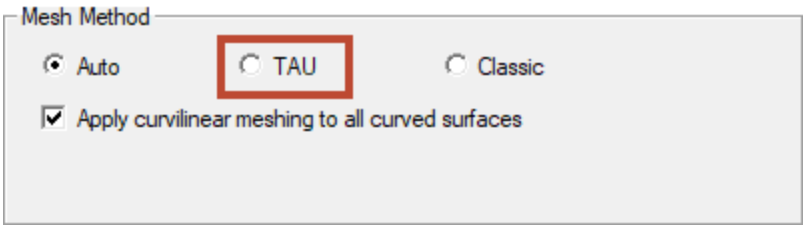


Selecting this option opens a dialog box. Select **Yes** to continue.



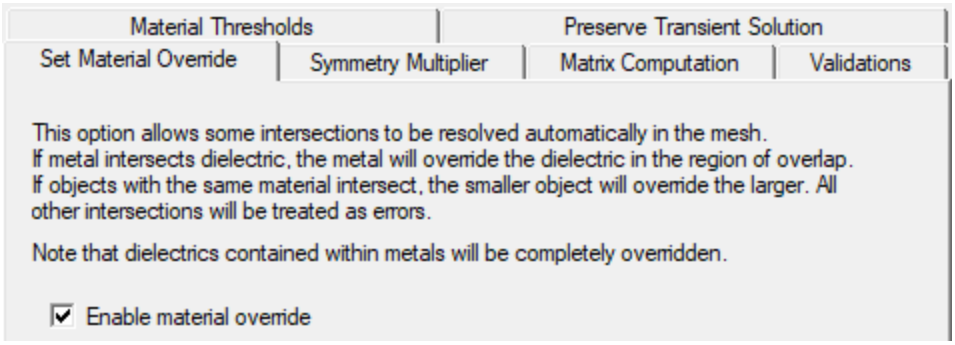
To get the full benefits of TAU Flex, especially for complex or "dirty" models, follow these recommended steps:

1. On the **General** tab of the **Initial Mesh Settings** window, select **TAU**.



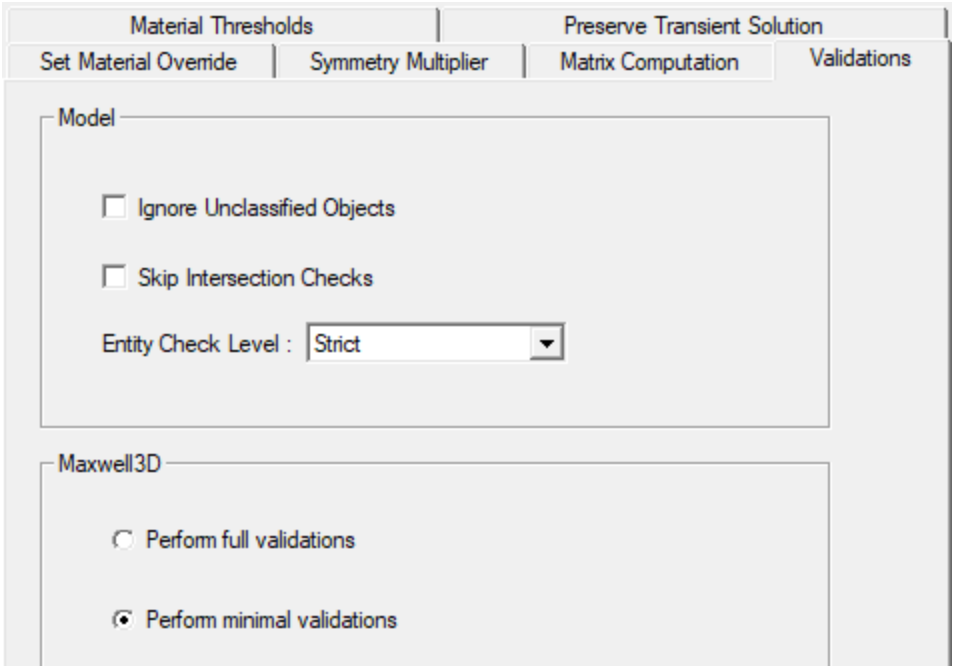
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **Maxwell** > **Design Settings** to open the **Design Settings** window.
3. On the Set Material Override tab, select **Enable material override**.



This activates TAU's priority meshing method based on metal/non-metal materials.

4. Still in the Design Settings window, click the **Validations** tab and select **Perform minimal validations**.



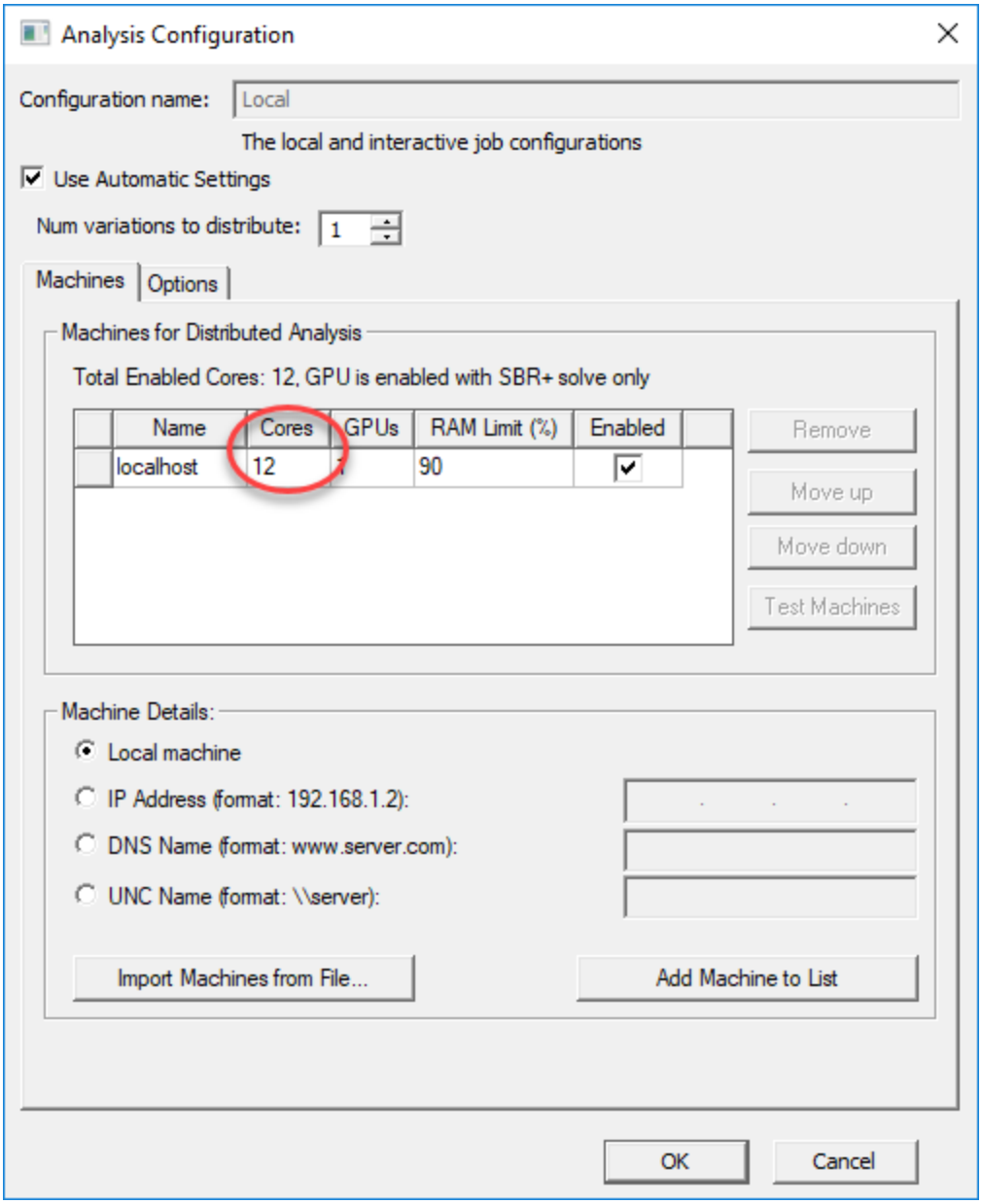
This reduces preprocessing time for complex models, and allows simulation of "dirty" geometry for which TAU Flex can provide a mesh.

5. Because TAU Flex is a parallel mesher for many processes, you should specify multiple cores in **HPC and Analysis Options** to speed up the meshing process. Access the Analysis Configuration window one of two ways:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select the Simulation tab on the ribbon and click **Analysis Config**.
- Use **Tools** > **Options** > **HPC and Analysis Options** or select the **Simulation** tab on the ribbon and click **HPC Options**. Then double-click a configuration.

The **Analysis Configuration** window appears.

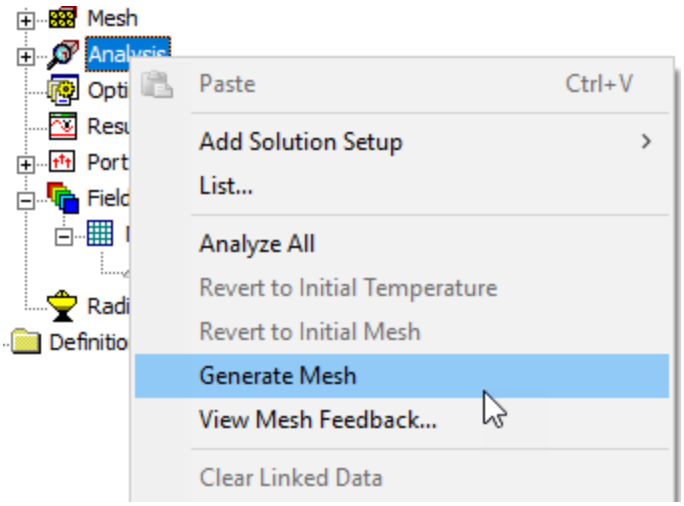


Specify the number of cores to use.

6. In most cases, TAU Flex can provide mesh as accurate as TAU or Classic, but not always. Therefore, we recommend that you **Generate Mesh** without solving, and that you review

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the mesh before proceeding the simulation.



TAU Flex Warning Messages

In cases where TAU Flex has difficulty making a strict mesh on some objects, mesh warnings are given for those objects. You should review the mesh on those objects before proceeding to the simulation. You can exercise discretion in deciding whether a given mesh issue needs to be addressed (for example, critical antenna geometry) or can be ignored (for example, a small "leak" in a dielectric substrate far from the source).

There are three types of warnings:

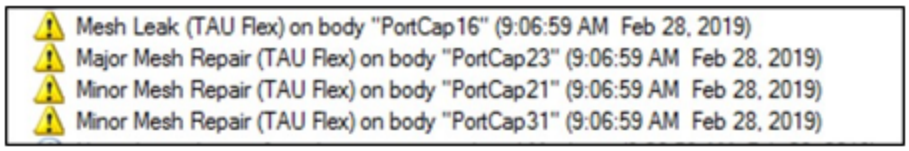
- **"Mesh Leak"** – *The most severe warning*. A body does not have any mesh element, or it has mesh elements with a significant mismatch of surface area between the mesh and geometry – larger than 10% of planar area, or 20% of curved area.
- **"Major Mesh Repair"** – A body has major surface area mismatch between the mesh and geometry – larger than 5% of planar area, or 10% of curved area.
- **"Minor Mesh Repair"** – *The least severe warning*. A body has minor surface area mismatch between the mesh and geometry – less than 5% for planar area, or 10% for curved area.

Reviewing TAU Flex Mesh Warnings

TAU offers three ways to view mesh warnings.

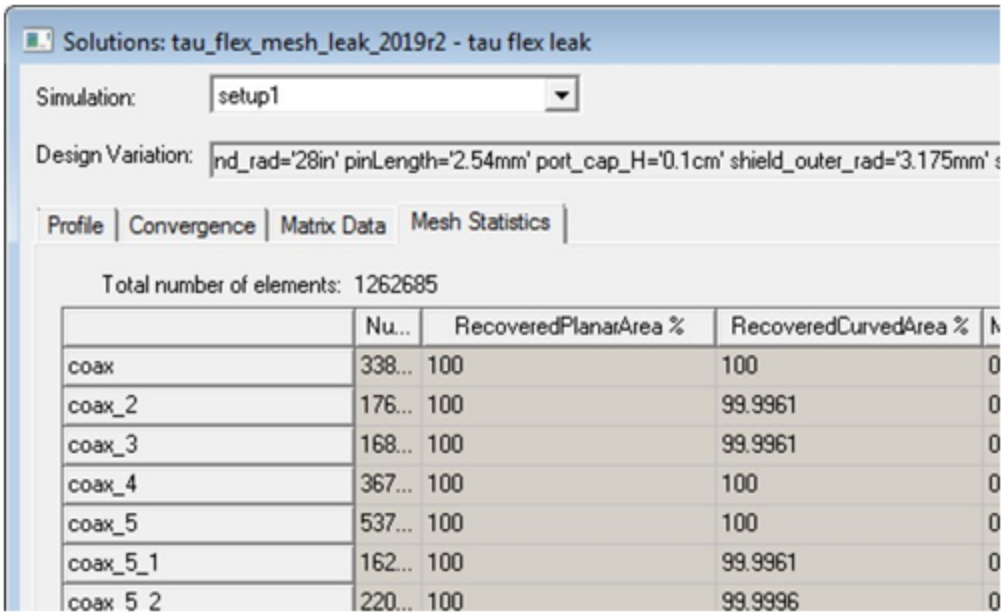
1. **Message Manager** – Immediately after the mesh generation is done, if a tolerant mesh is given, warnings display in the Message Manager. If the window is hidden, click **Show Messages**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. **Mesh Statistics** – On the **Results** tab of the ribbon, click **Solution Data** to open a **Solutions** window.

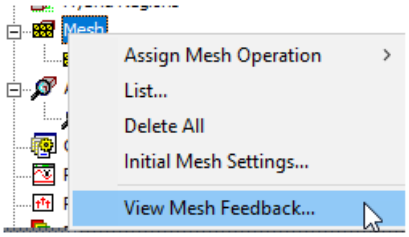
   On the **Mesh Statistics** tab of the **Solutions** window, if a tolerant mesh is generated, the surface area difference between the mesh and geometric model at body level is listed as "RecoveredPlanarArea" and "RecoveredCurveArea". 100% means that the body has a strict mesh. You can click the surface area tab to sort the body list and see the tolerant mesh bodies first.

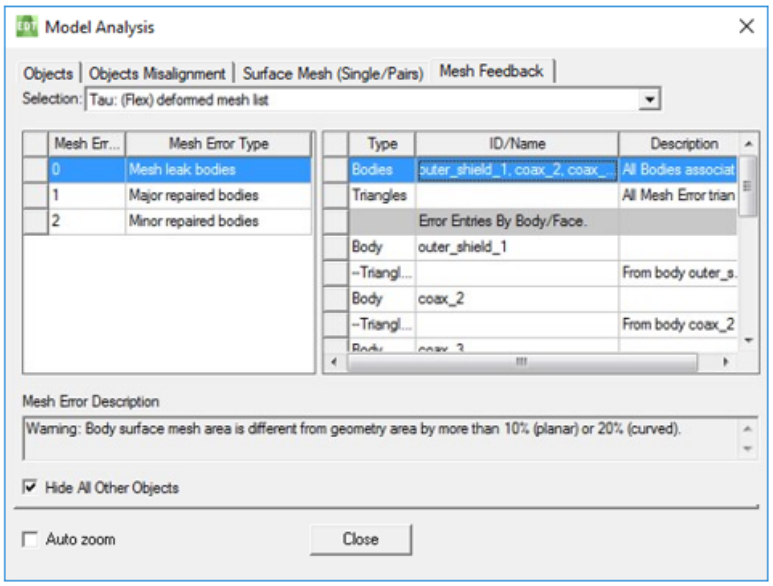


	Nu...	RecoveredPlanarArea %	RecoveredCurvedArea %	
coax	338...	100	100	0
coax_2	176...	100	99.9961	0
coax_3	168...	100	99.9961	0
coax_4	367...	100	100	0
coax_5	537...	100	100	0
coax_5_1	162...	100	99.9961	0
coax_5_2	220...	100	99.9996	0

3. **Model Analysis** – To view the details of the tolerant mesh, click **Modeler** > **Model Analysis** > **Show Analysis Dialog** > **View Mesh Feedback** to launch the **Model Analysis** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Click on the top list to display objects that have tolerant mesh.

Click the "Triangles" to display the mesh on the tolerant objects.

Individual objects are listed under "Error Entities By Body/Face". You can select one or multiple bodies to view the mesh at a more detailed level.

Addressing TAU Flex Mesh Issues

In cases where TAU Flex has difficulty making a strict mesh on some objects, you can address the problem using one of the following techniques:

1. Redraw the geometry in question in the native modeler.
2. Heal the geometry, either with integrated healers or perhaps in Discovery.
3. If the object has curved or true surfaces, consider applying a localized surface mesh operation which creates a finer than default mesh.
4. Remove difficult objects that are not critical to the simulation.

Specifying Surface Priority for TAU Mesher

You can choose to specify the surface representation priority for the TAU mesher. For most designs, you can let the solver automatically choose which of two meshing approaches to take. The solver predicts which one gives the best results, balancing mesh reliability, speed, quality, size and design characteristics. In most cases, the solver uses TAU mesh, rather than the classic mesh. In general, it looks for specific features (for example, stacks of large planar parallel facets with small gaps) and situations where the initial TAU mesh is 4 times larger than the

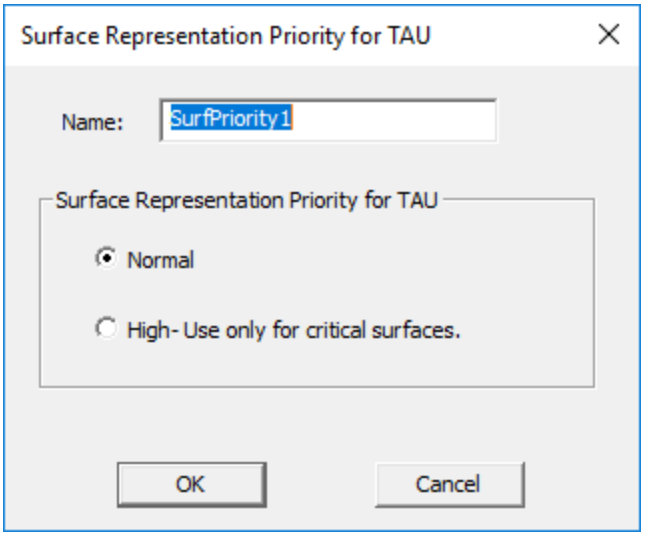
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Classic. Experienced users may choose to designate a high surface representation priority for the critical surfaces with the TAU mesh is used.

> **Important:**
>
> Use this mesh operation with caution. Use this option only if the model has tiny elements or critical boundary conditions since the solver performs several additional strict mesh verification steps on the selected objects.

1. Select the surface of interest.
2. If desired, specify a Name for the Mesh operation.
3. Right-click the **Mesh** icon in the Project tree to view the shortcut menu and select **Assign Mesh Operation** > **Surface Priority for Tau**.

   The **Surface Representation Priority for Tau** dialog box appears.



4. Select **High** for the critical surface.
5. Click **OK**.

   The Mesh operation appears in the Project Manager.

TAU 2D Mesh Generation

The Maxwell TAU (Triangular Adaptive Uniform) 2D mesher provides a fully automated initial mesh generation method that creates a uniform, high quality, solver-driven mesh. It is designed for solvers without adaptive passes, or demanding high quality mesh. Several meshing methods

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

are used by the TAU 2D mesher, such as skin depth layer mesh, clone mesh, rotational sweep mesh, and remeshing.

To benefit from these meshing technologies for Maxwell 2D designs, ensure that **TAU** is selected as the **Mesh Method** on the **Initial Mesh Settings** dialog **General** tab by right-clicking the Mesh icon in the Project Manager and selecting **Mesh** > **Initial Mesh Setting**, or by selecting **Maxwell** > **Mesh** > **Initial Mesh Settings...**. Note that **TAU** mesh is the default setting for new 2D designs.

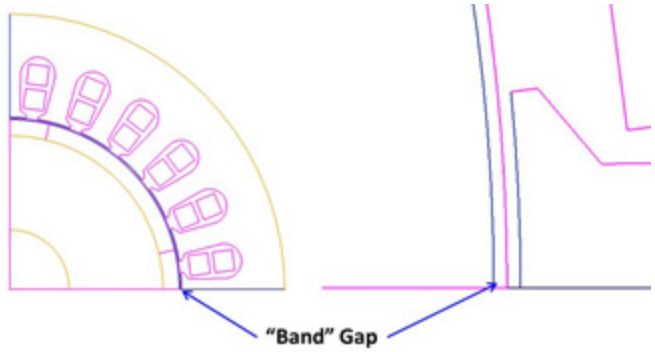
When TAU 2D mesh is selected, its meshing methods are applied to specific local regions. The local regional meshes are then assembled as the final step for the simulation.

The following sections describe TAU 2D mesh generation methods for rotational models, for non-rotational models; and the Skin Depth Layer Mesh Generation method, which can be applied to any model.

TAU 2D Mesh Generation for Rotational Models

If the mesher determines that a model is a rotational model, it uses a rotational mesh generation with either the clone mesh or rotational sweep mesh method.

To determine if the model is a rotational model, the curves around the origin are calculated. The smallest rotational gap is defined as the “Band” gap. The model shown below is an example of a rotational transient model:



Two major regions, a static region and a moving region, are classified by the Band gap.

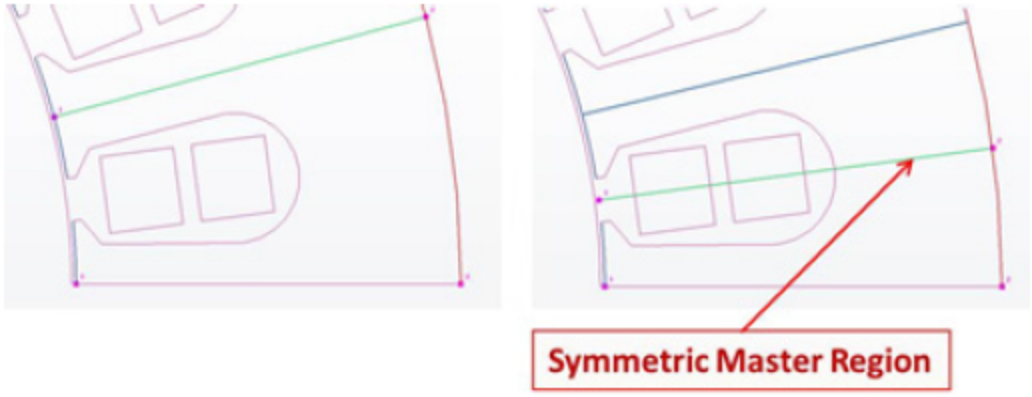
TAU 2D Clone Mesh Generation

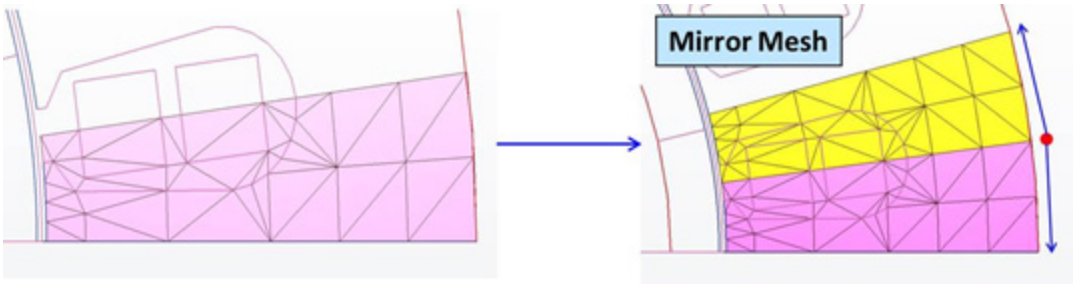
> **Note:** To use clone mesh, you must also setup Length-Based Mesh Refinement Inside Objects.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The static geometric model in the local rotational region is analyzed to determine whether it has duplicate subregions. If a model has duplicate subregions, the TAU clone meshing is automatically activated and will generate the mesh so that each subregion has the topologically identical mesh. The following example shows a static clone region with six identical subregions - a master and clone regions:

In the clone region, the master region is calculated and analyzed. If the master region has a symmetric feature, the symmetric mesh is generated; see "Maxwell 2D Symmetry Mesh" on page 12-65 below.

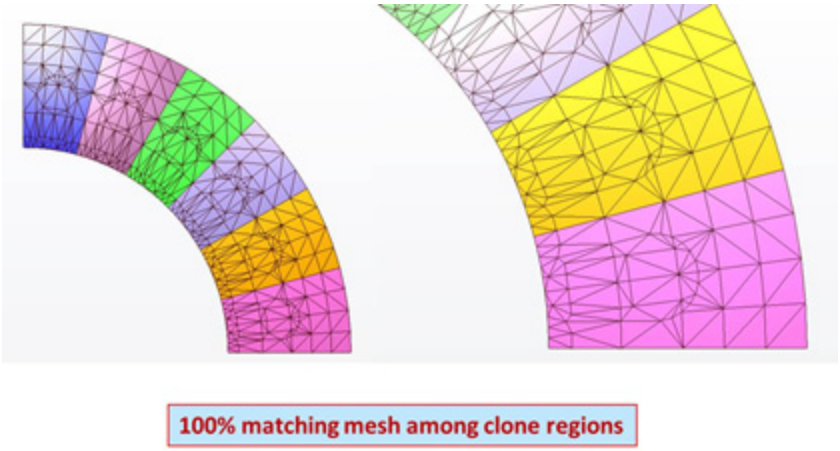




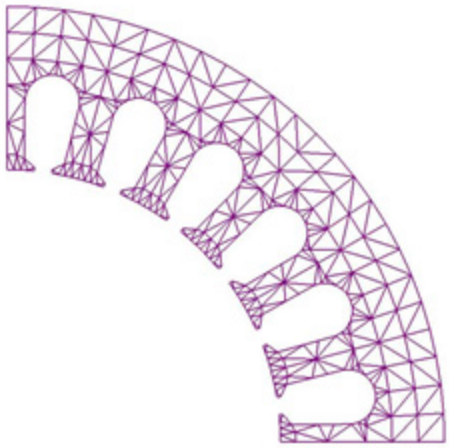
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

As shown above, a uniform mesh will be generated in one half of the master region first, then the mesh is mirrored about the symmetric central line to generate the symmetric mesh in the master region.

Because all the child regions have identical geometric features, the mesher uses copy-paste to replicate the master mesh at each of the child regions, rotating it to the appropriate angles.



This ensures the identical mesh in the different regions, especially for the teeth.



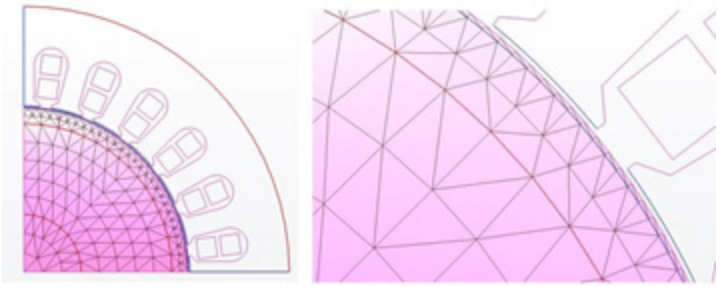
TAU 2D Rotational Sweep Mesh Generation

If the local region does *not* have identical geometric subregions and thus the clone mesh is *not* generated, a geometric analysis is performed to verify whether the region is rotational.

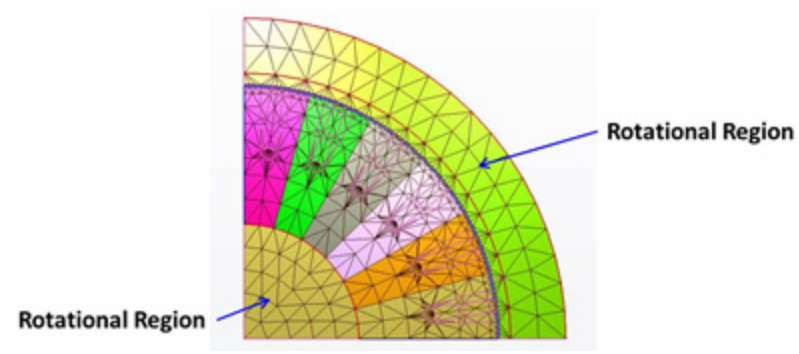
Around the rotational center, if a region has two or more rotational curves, this region can be defined as a “rotational” region. In this case, rotational sweep mesh generation is used

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

proceeding in the radial direction with layers while using uniform sweep elements in the rotational direction. This meshing method controls the aspect ratio in the radial direction, while maintaining the uniform sweep in the rotational direction.



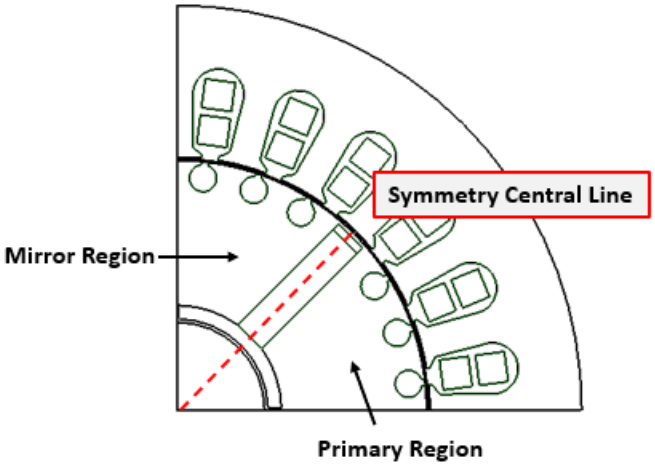
Rotational Sweeping Mesh



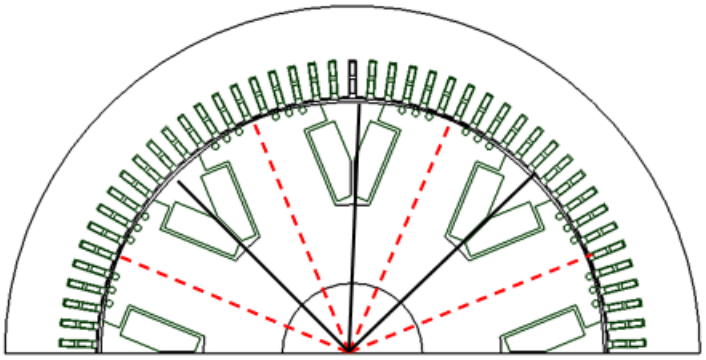
Maxwell 2D Symmetry Mesh

A symmetry pattern is characterized by the existence of a symmetry central line. The objects/outlines at one side of the symmetry central line can be constructed mirroring the objects/outlines at the other side of the symmetry line:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

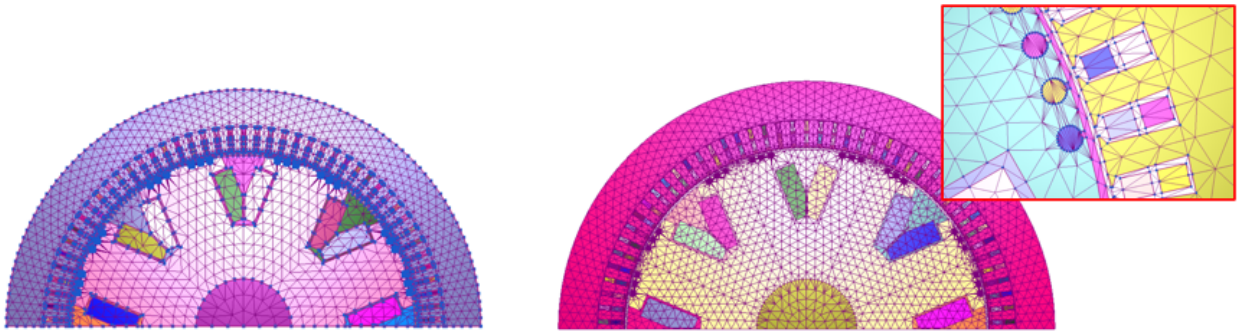


A specific geometry can be a combination of a mirror and a circular pattern. For example, in the transient simulation of motors, the design may contain one or multiple symmetry patterns, as shown below:



The TAU mesher will automatically detect such patterns regarding their inner symmetry and their circular periodicity. It will then create a symmetric and periodic mesh to reduce simulation noise. The resulting mesh can be plotted to verify its symmetry:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Non-Symmetry Mesh

Symmetry Mesh

Symmetry mesh can be combined with mesh density control when this mesh operation is applied to the initial mesh.

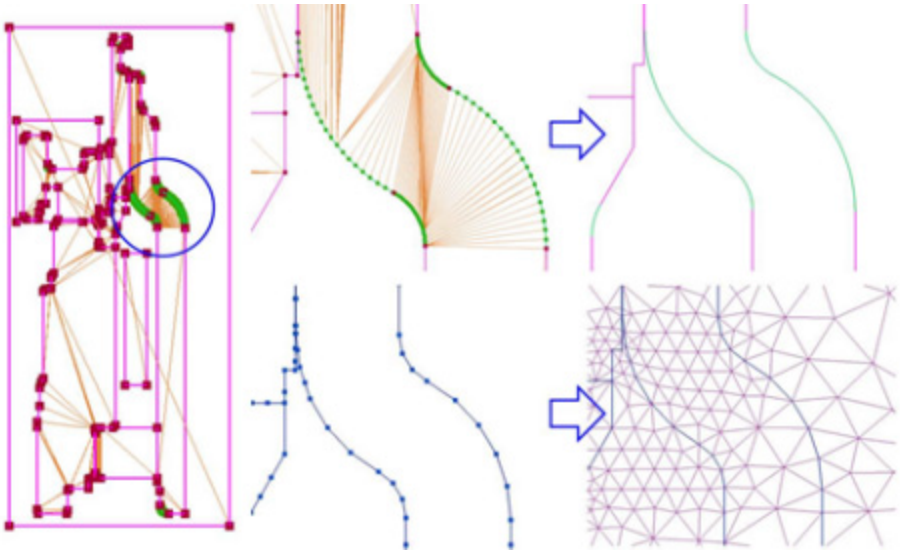
Note the following:

1. To generate a symmetry mesh, the band of motion must be defined using a curved, *i.e.*, non-segmented, object such as an unsegmented circle with one outer edge.
2. In addition to (1), no segmented outlines should be defined around the band (*e.g.*, inner region object). If required, these objects should also be defined by unsegmented, curved geometry.
3. Symmetry mesh is currently not compatible with a band mapping angle mesh.

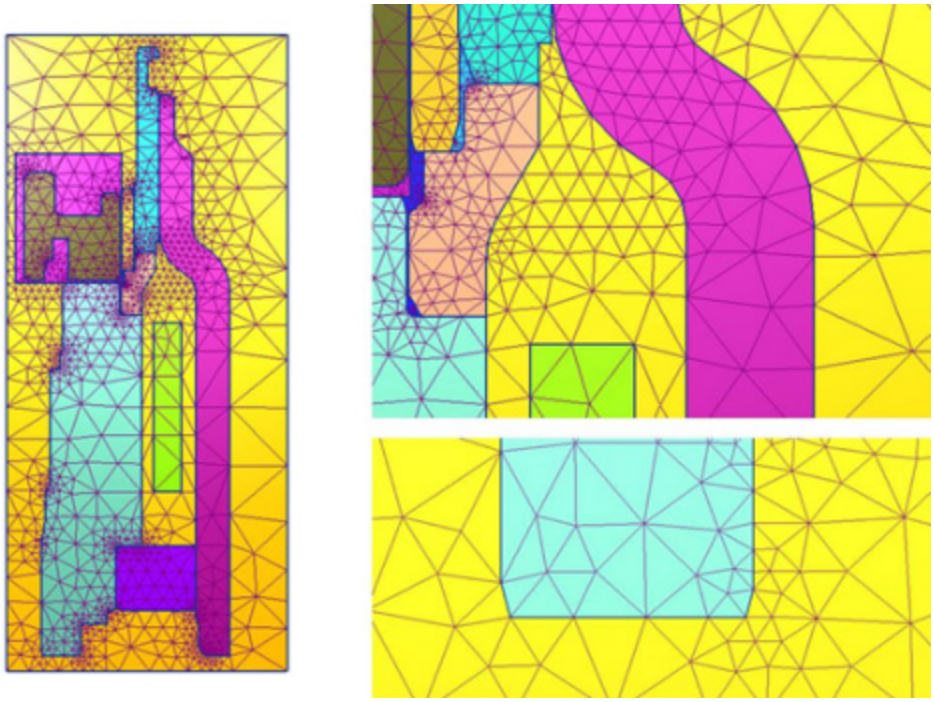
TAU 2D Regular Mesh Generation (Non-Rotational Models)

If the model is not identified as a rotational model, the TAU 2D regular mesher is used to generate the mesh. TAU 2D mesher does not use Parasolid union as the Classic mesher does. Instead it reads the objects individually as-is. Remesh technology is used to get the best representation of the curves with minimum mesh points.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The TAU 2D regular mesher uses a top-down meshing algorithm that starts the mesh from a uniform base mesh, then adapts the mesh in the necessary regions, and does the boundary recovery later. Mesh quality is controlled and maintained during the entire meshing process. To maintain a uniform mesh, 1:2 splitting method is used for the transitions from small elements to large elements.



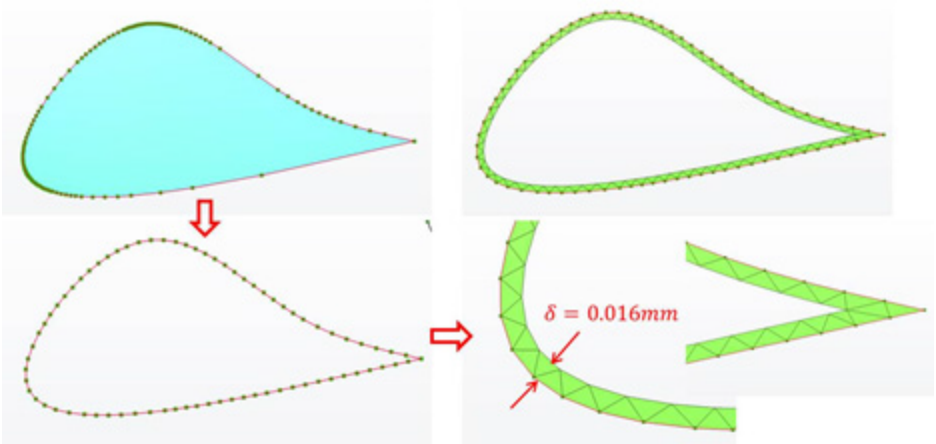
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In most cases, users do not need to add any additional seed refinement mesh operations. The mesh can be directly used to do the transient simulation. For solvers with adaptive passes, a few passes can be saved for the convergence although the initial mesh is larger since it is high quality and uniform.

TAU 2D Skin Depth Layer Mesh Generation

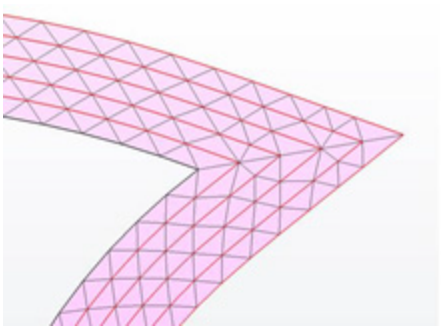
For this mesh generation method, the number of layers and the thickness can be specified by selecting one or more body edges. They will be exactly respected in the TAU mesh unless the layer thickness is too large, or layers from other directions overlap, intersect, or leave a narrow gap.

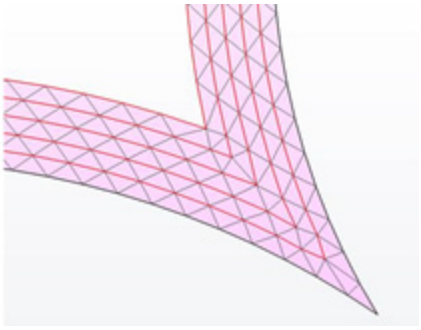
The layer mesh starts from the specified edges, and advances toward the direction of the body region layer-by-layer. If two bodies contact each other, the edges from these two bodies at the contact region may coincide, *appearing* as one edge even though there actually are two or more edges. Therefore, be careful when picking the edges in multiple body contact regions. In such cases, it is recommended that you hide all other bodies and only display the desired body, then make the edge selections. Remesh technology is then used to generate the mesh on the edges based on the thickness of the first layer to maintain the mesh quality.



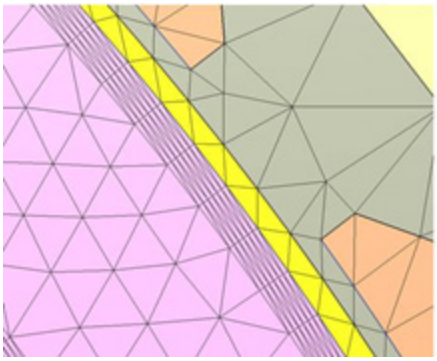
The aspect ratio in the layer mesh is defined by the element length in the tangential and normal directions. The ratio will be in the range of 1 to 40 based on the smallest thickness in the layers. Different layer mesh methods are using depending on the ratios. If the ratio is smaller than 4.0, the middle points of the edges are used to advance layers forward as shown below.

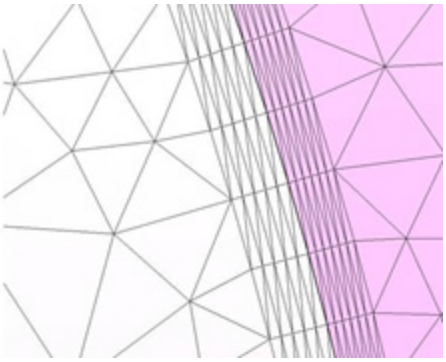
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





If the ratio is more than 4.0, edge points are used to advance layers.

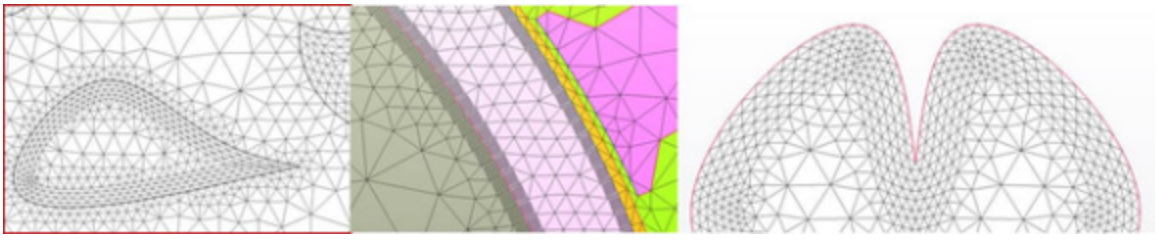




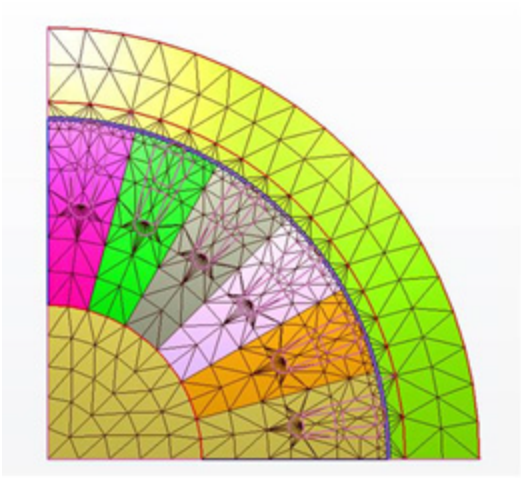
Messages are displayed if the thickness is too small or too large.

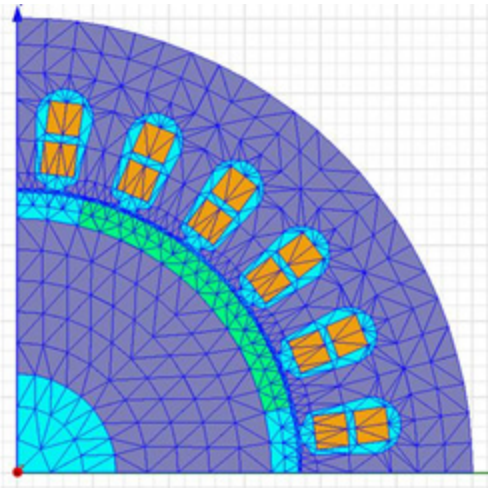
TAU 2D Mesh Assembly

To get a conformal mesh for the entire model, the final step assembles the local region meshes generated by the clone mesh, rotational sweep mesh, and skin depth layer mesh methods with the mesh generated by the TAU 2D regular meshing method.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Related Topics

TAU 2D Mesh Generation for Rotational Models

TAU 2D Regular Mesh Generation (Non-Rotational Models)

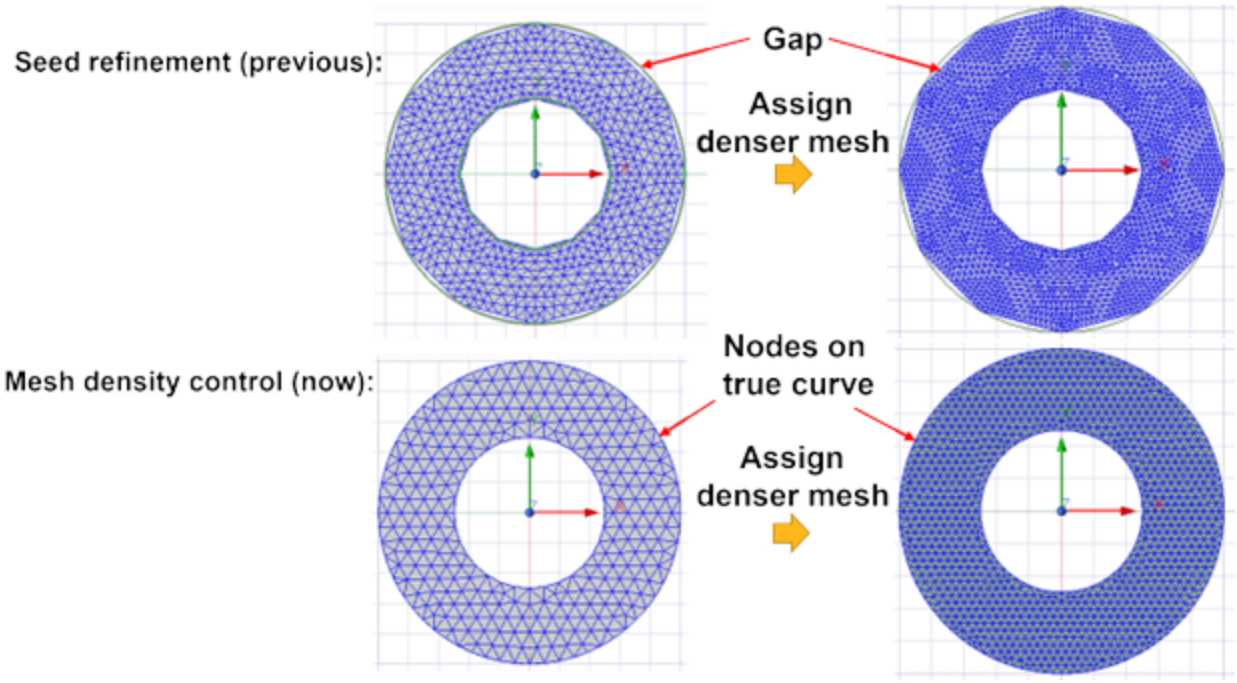
TAU 2D Skin Depth Layer Mesh Generation

TAU2D: Mesh Density Control Over Selected Objects

Mesh density control is a mesh feature used to control the mesh length on selected objects in the phase of initial mesh generation. When you request a desired mesh length on selected parts of the model, the mesh length over the selected part will not be larger than the requested length.

Previously, mesh density was controlled by seed refinement. After the initial mesh is generated, the size of the triangles over the selected body will be refined to meet the specified requirement. However, seed refinement will destroy the cloned mesh pattern generated by the initial mesh. Now, mesh density can be controlled in the phase of initial mesh generation. Combined with clone mesh, mesh density control guarantees that all clone regions have clone mesh pattern. If mesh density is assigned over a body in the primary region, all the clone bodies associated with the selected primary body will have the same density of mesh. It also provides a better geometric approximation of curves.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Mesh density can be controlled at three levels:

- model level
- body level
- edge level

If model level mesh density is specified, mesh over the entire model will be finer than the requested length. If body or edge level mesh density is specified, only the mesh over the selected bodies or edges will be finer than the requested length.

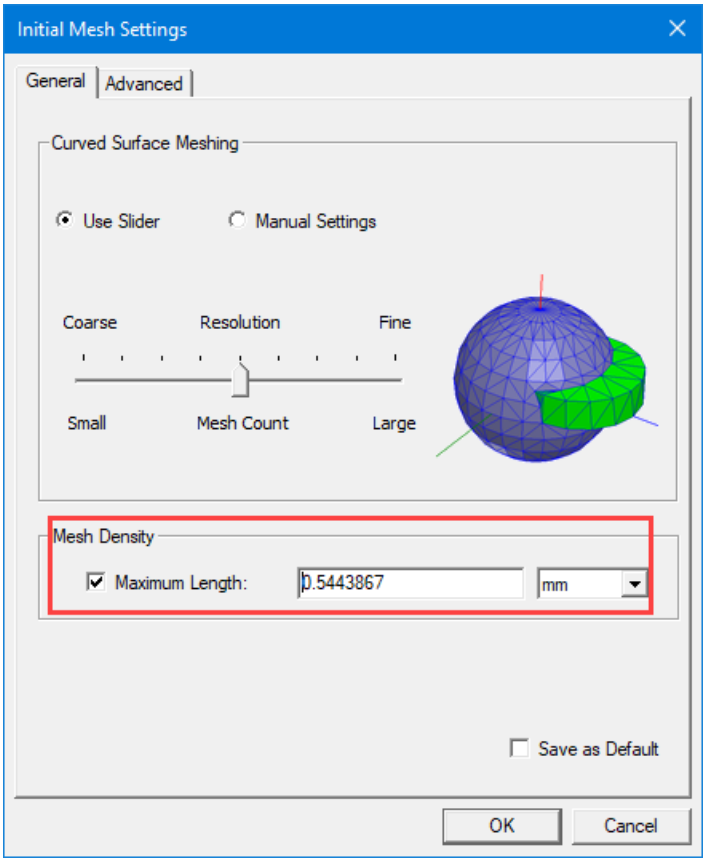
Different mesh densities can be specified over different parts of the model. If multiple mesh densities are requested on the same object, the mesh over the object will be finer than the minimum of all declared lengths. For example, if you request 1 mm of desired mesh length over the entire model and 2 mm of mesh length over a selected object, the generated mesh length over the selected object will be finer than 1 mm. On the other hand, if the user requests 0.5 mm over the object, the generated mesh length over the selected object will be finer than 0.5 mm.

Setting up model level mesh density

1. Select **Maxwell** > **Mesh** > **Initial Mesh Settings...** or in the Project tree, right-click on **Mesh**, and select **Initial Mesh Settings** from the shortcut menu. Alternatively, on the **Simulation** tab of the ribbon, select **Mesh Settings**.

   The **Initial Mesh Settings** dialog box appears with the **General** tab selected.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



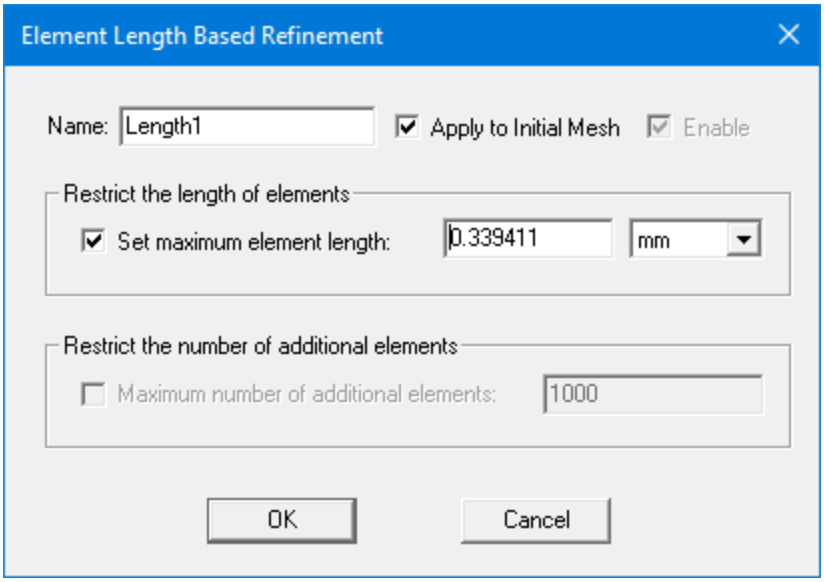
2. In the **Mesh Density** section, enable the **Maximum Length** check-box, and enter the desired value and unit, which must be greater than zero.
3. Click **OK** to close the dialog box.

Setting up body level mesh density on selected objects

1. When you open a 2D project, select the body (or multiple bodies) on which you want to apply the mesh density operation.
2. Right-click on the selected object(s) and select **Assign Mesh Operation** > **Inside Selection** > **Length Based**.

   The **Element Length Based Refinement** dialog box opens.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Enable the **Apply to Initial Mesh** check box, then enter the desired value, which must be greater than zero, in the **Set maximum element length** box, then click **OK** to close the dialog box.

> **Note:** If **Apply to Initial Mesh** is checked, input for **Maximum number of additional elements** will be ignored and grayed out.

Setting up mesh density on selected edges

1. When you open a 2D project, first make sure that **Select Mode** is set to **Edges** by right-clicking on the model, and choosing **Selection Mode** > **Edges**.

   Select the edge (or multiple edges) on which you want to apply the mesh density operation.

2. Right-click on the selected edge(s) and select **Assign Mesh Operation** > **On Selection** > **Length Based**.

   The **Element Length Based Refinement** dialog box opens.

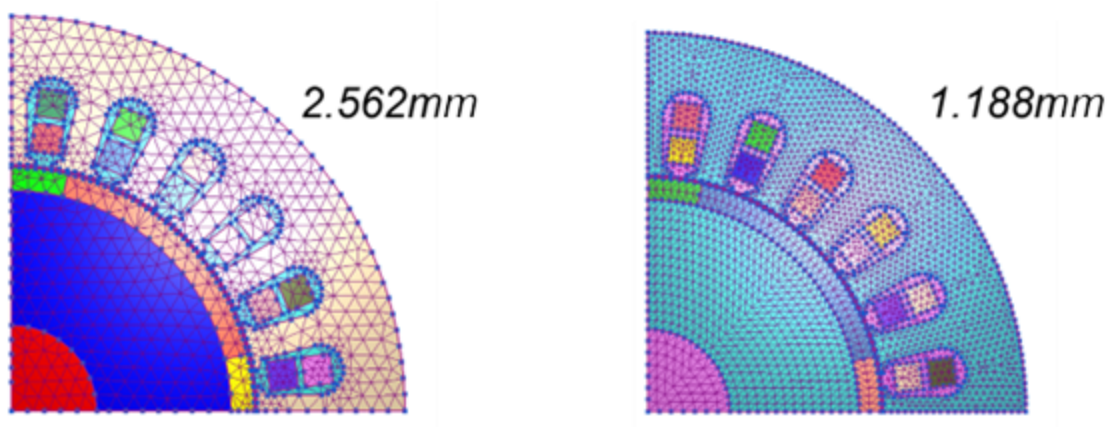
3. Enable the **Apply to Initial Mesh** check box, then enter the desired value, which must be greater than zero, in the **Set maximum element length** box, then click **OK** to close the dialog box.

> **Note:** If **Apply to Initial Mesh** is checked, input for **Maximum number of additional elements** will be ignored and grayed out.

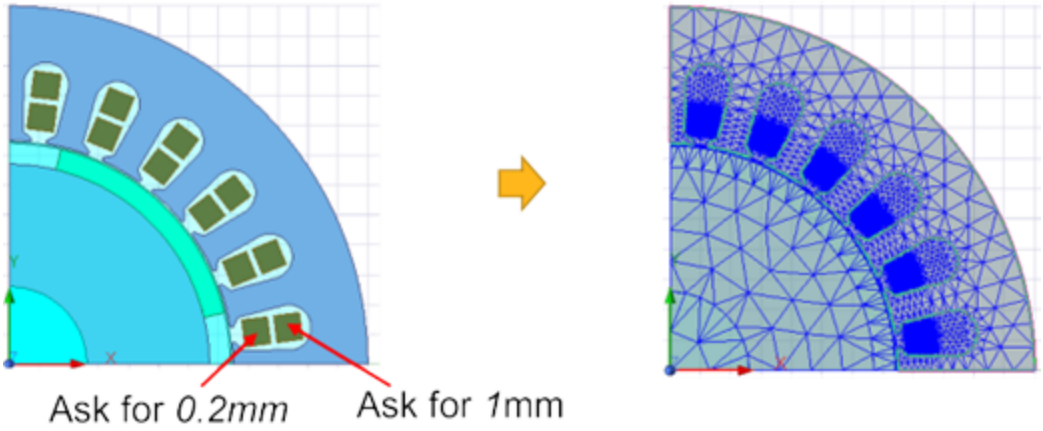
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The effects of the mesh density control can be seen in mesh plots:

Model Level

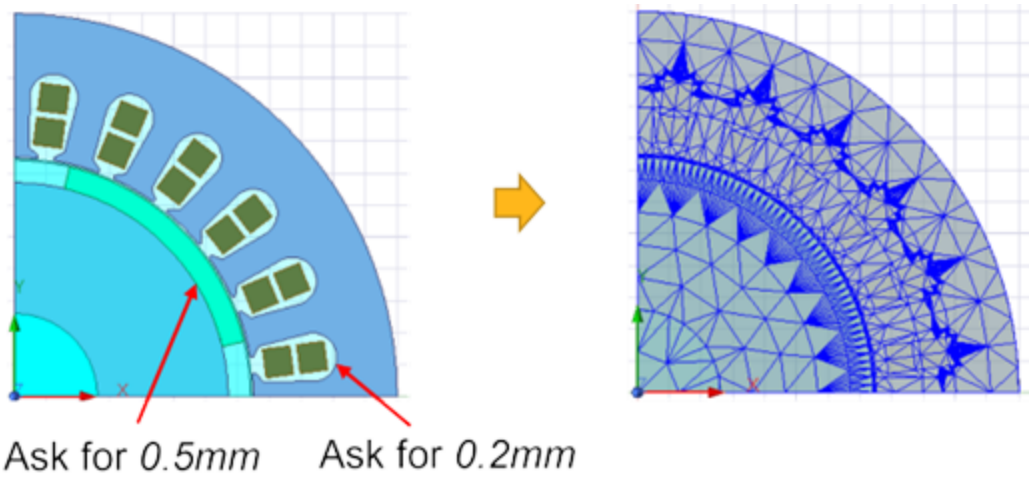


Body Level



Edge Level

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



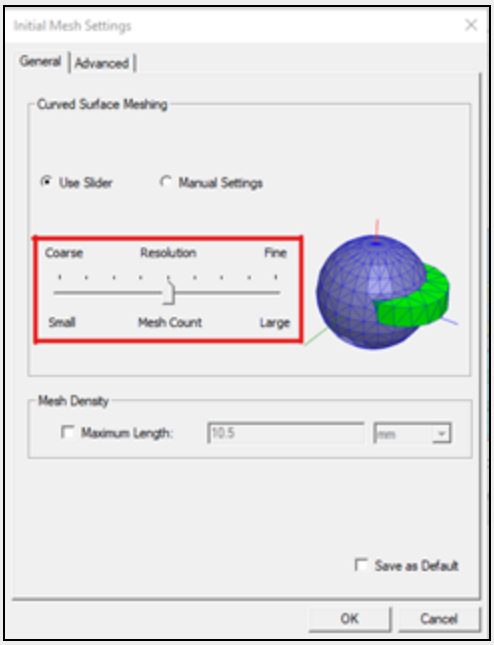
Note: For clone mesh, you should select objects in the primary region to get clone mesh pattern.

Note: For clone mesh, if the region is not repeatable, mesh density is not working yet in the current release. You can assign seed refinement to get the dense mesh as a workaround.

Note: If band mapping angle is enabled, only apply *model* level mesh density control. Neither *body* nor *edge* level mesh density control should be assigned as these may break the cogging torque mesh pattern.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: The **Mesh Count** slide bar must be set to 5 in the **Initial Mesh Settings** dialog box if you assign mesh density to spline curves.



Layer Mesh Generation for 3D Rotational Models

Layer mesh generation for 3D rotational models can only be applied to non-skewed bodies in Maxwell rotational models in which the CylindricalGap mesh operation is used.

- Only body-level selection is allowed.
- Rotational layer mesh can only be generated on non-skewed objects for current release.
- User can select multiple bodies, and assign a layer mesh operation to all of them.
- Rotational layer mesh is only generated for bodies near the band.
- For selected bodies, no layer mesh will be generated in the axial direction, or for matching boundaries.

There are three types of layer meshes for 3D rotational models.

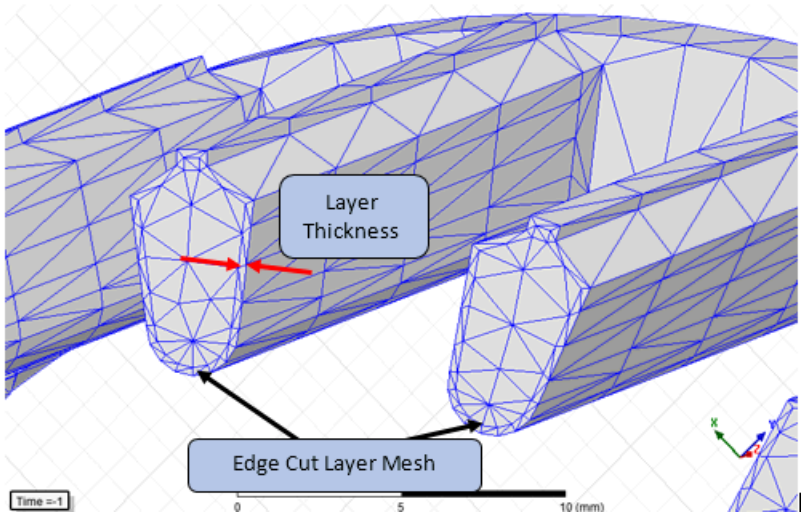
- Edge Cut Layer Mesh
- Rotational Layer Mesh
- Edge Cut + Rotational Layer Mesh

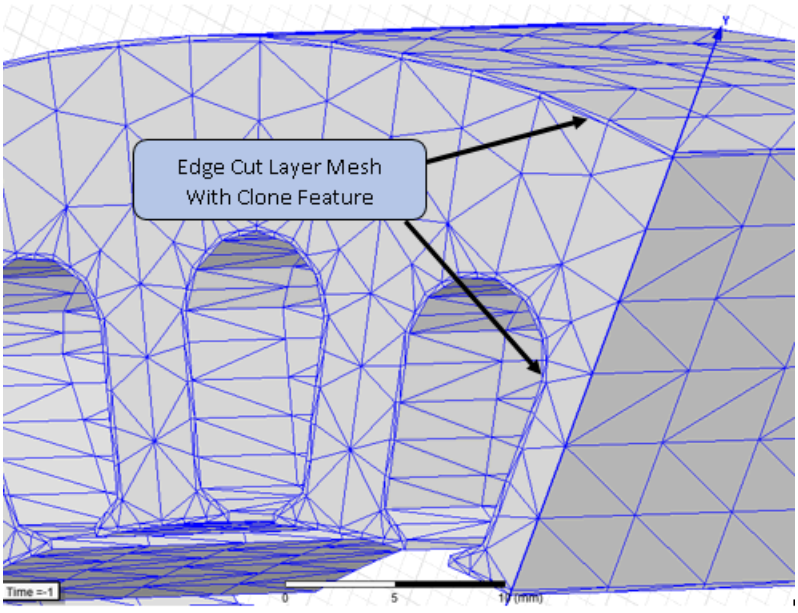
These mesh operations will work only if the clone mesh works.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Edge Cut Layer Mesh Generation

Edge Cut Layer Mesh creates one layer of mesh on the bodies with the layer thickness defined by user input. Layer mesh will not be created at the top or bottom of the axial direction and matching boundary faces. If the object has clone feature, the layer mesh will be coupled with the clone mesh.

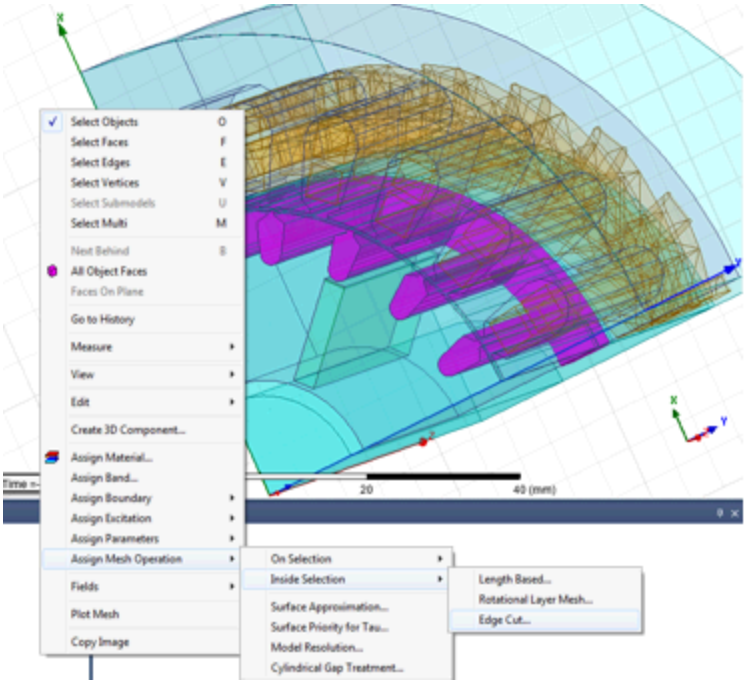




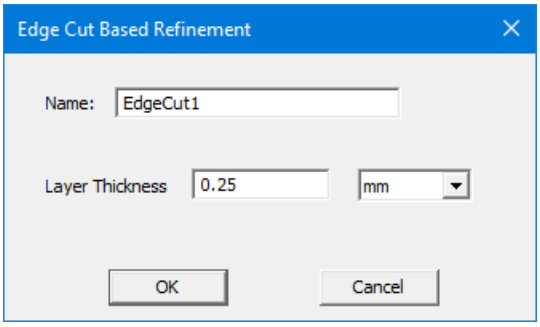
To apply Edge Cut Layer Mesh:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the desired bodies near the band.
2. Right-click to open the context menu, and select **Assign Mesh Operation** > **Inside Selection** > **Edge Cut Based**.



3. In the **Edge Cut Based Refinement** dialog box, you can rename the mesh operation, and specify the layer thickness. The recommended layer thickness is the band gap distance.

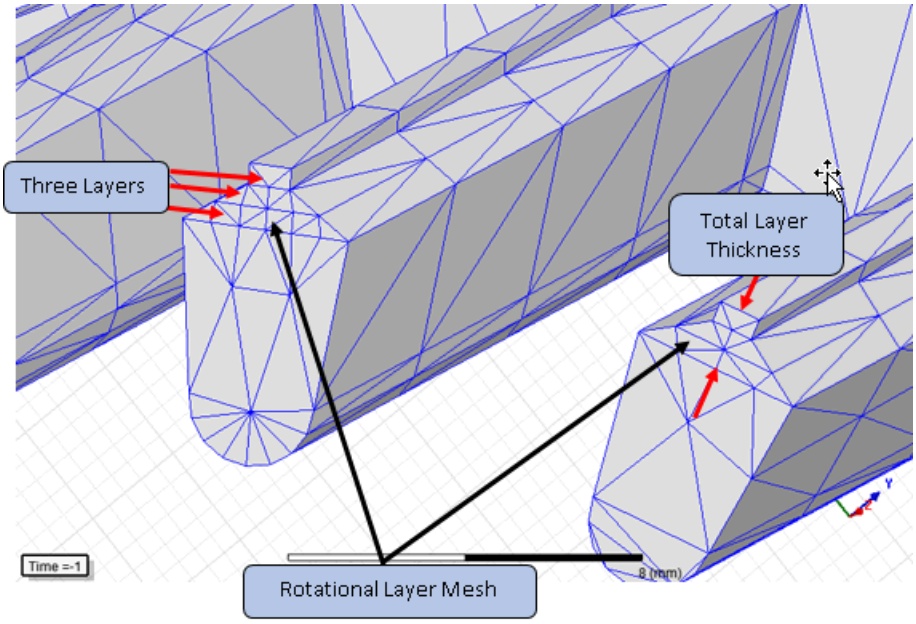


4. Click **OK**.

Rotational Layer Mesh Generation

Rotational Layer Mesh creates up to three layers in the radial direction starting from the faces most adjacent to the “Band”. You can define the **Number of Layers** (up to three) and **Total Layer Thickness**.

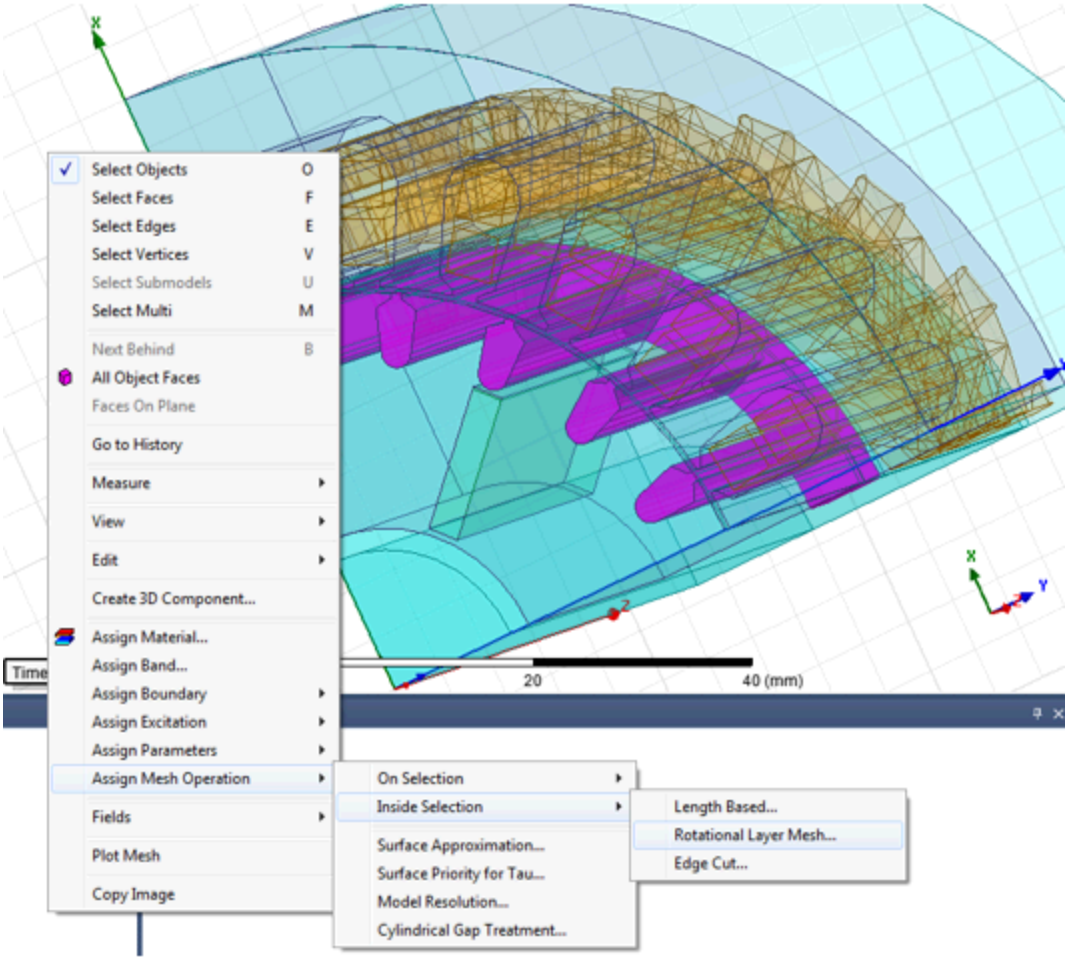
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



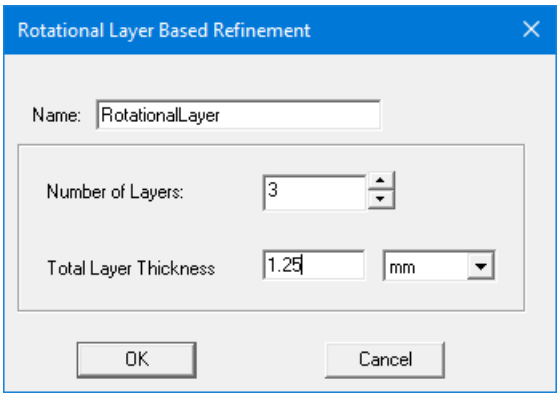
To apply Rotational Layer Mesh:

1. Select the desired bodies near the band.
2. Right-click to open the context menu, and select **Assign Mesh Operation** > **Inside Selection** > **Rotational Layer Mesh**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. In the **Rotational Layer Based Refinement** dialog box, you can rename the mesh operation, specify the number of layers (maximum of 3), and the total thickness of all the layers combined.

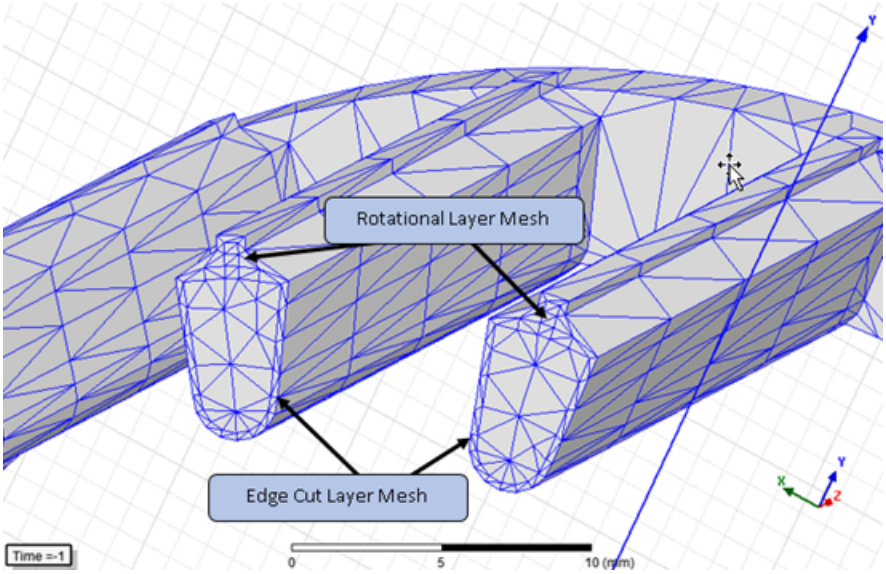


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **OK**.

Edge Cut + Rotational Layer Mesh Generation

These settings are a combination of Edge Cut Layer Mesh and Rotational Layer Mesh settings, and as such has the properties of both. One layer of edge cut mesh will be generated first, then extra multiple layers will be added by the rotational layer mesh settings.



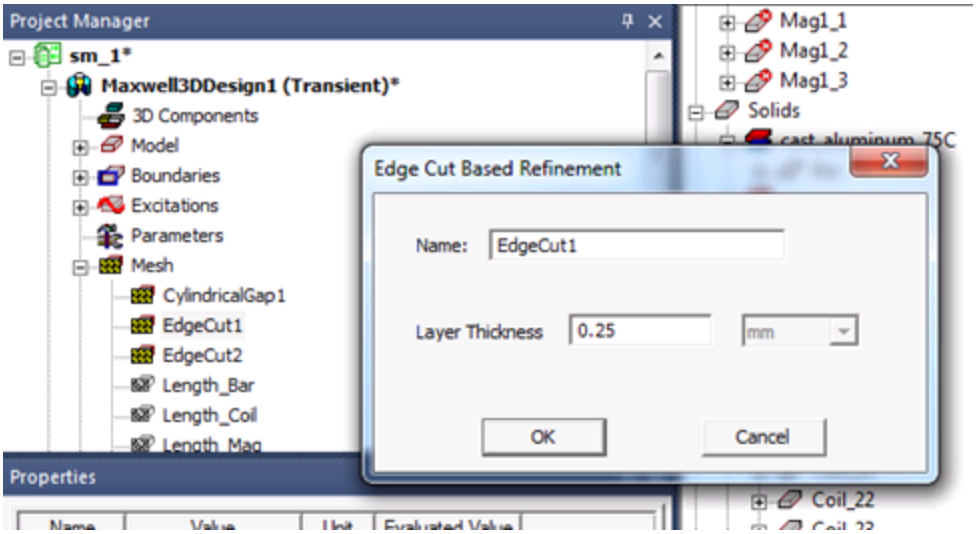
To apply Edge Cut + Rotational Layer Mesh:

1. Select the desired bodies near the band.
2. Right-click to open the context menu, and select **Assign Mesh Operation** > **Inside Selection** > **Edge Cut**.
3. In the **Edge Cut Based Refinement** dialog box, you can rename the mesh operation, and specify the layer thickness. The recommended layer thickness is the band gap distance.
4. Reselect the same bodies.
5. Right-click to open the context menu, and select **Assign Mesh Operation** > **Inside Selection** > **Rotational Layer Mesh**.
6. In the **Rotational Layer Based Refinement** dialog box, you can rename the mesh operation, specify the number of layers (maximum of 3), and the total thickness of all the layers combined.
7. Click **OK**.

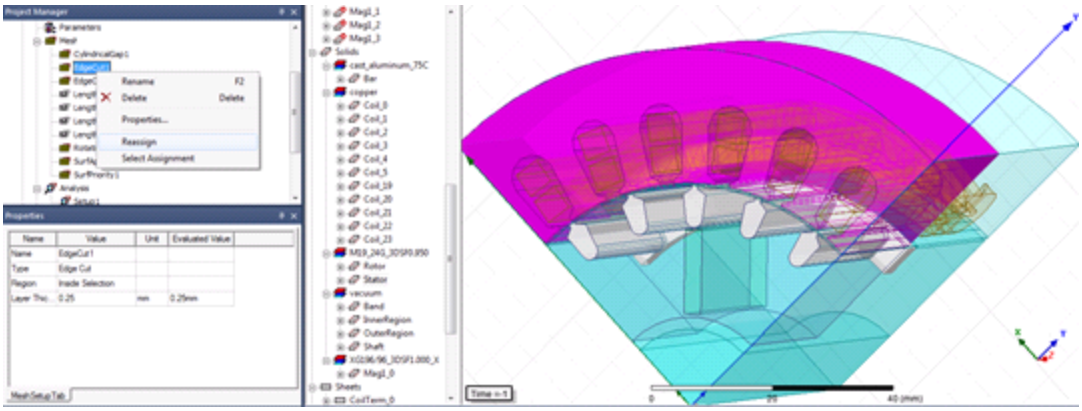
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Changing Properties and Reassigning Mesh Operations

To alter the mesh settings, find the entry under **Mesh** in the Project Manager tree. Double-click to bring up the dialog box. Change the layer thickness or number of layers as necessary.



To reassign which bodies the mesh operations are applied to, first select all the bodies you would like to assign the bodies to. Then right-click the mesh operation in the Project Manager and select **Reassign**. The mesh operation will now be reassigned to the selected objects.



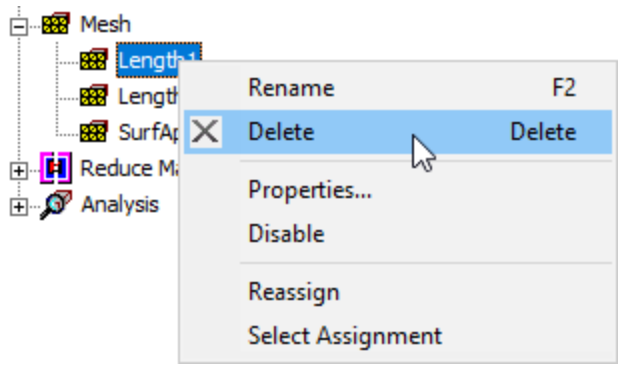
Deleting Previously Assigned Mesh Refinements

You can delete an individual mesh refinement, or delete all previously assigned mesh refinements.

To delete a single refinement:

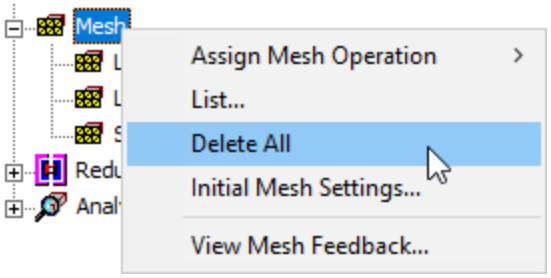
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- In the Project Manager, right-click the mesh refinement and click **Delete**, or press the **Delete** key.



To delete all previously assigned mesh refinement data:

- Click **Maxwell 3D**, **Maxwell 2D**, or **RMxprt** > **Mesh** > **Delete All**.
- In the Project Manager, right-click **Mesh** and select **Delete All**.



> **Note:**
>
> This will not alter any existing meshes that have already made use of one or more mesh refinement operations. However, deleted mesh operations will not be used again after reverting to the initial mesh.

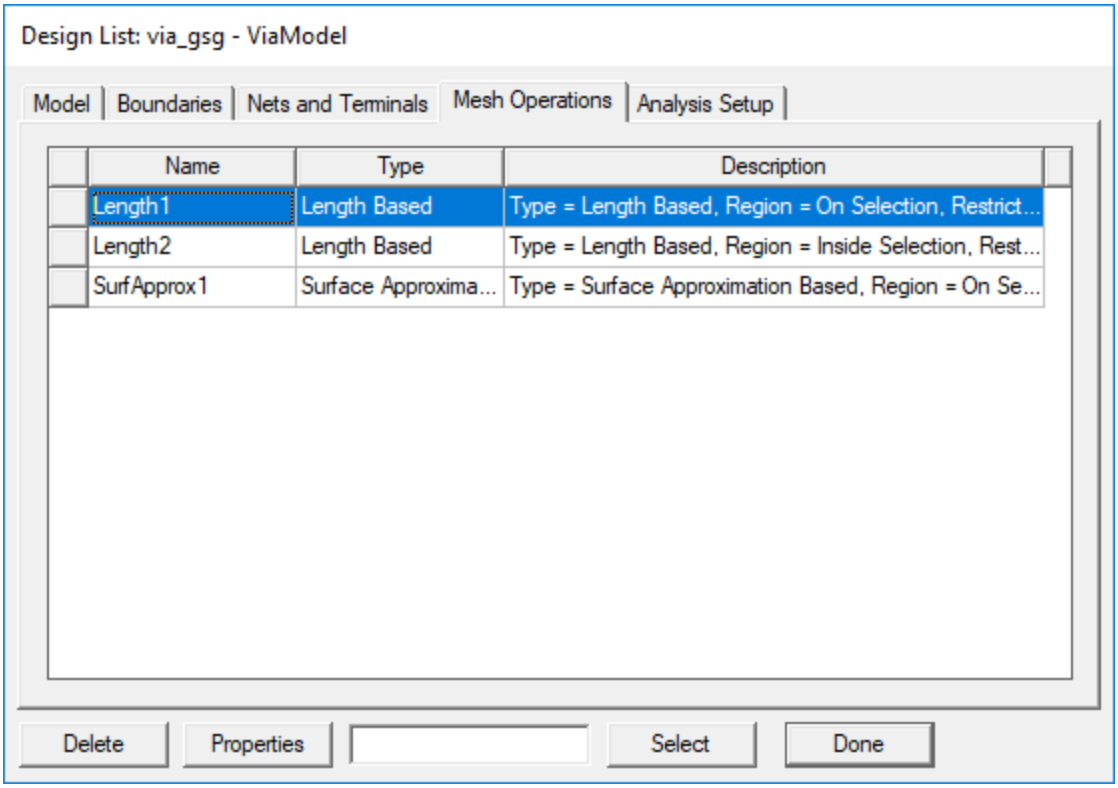
Viewing List of Mesh Operations

To list all mesh operations for the project:

1. In the Project Tree, right-click **Mesh** and select **List**.

   The **Design List** window appears, on the **Mesh Operations** tab.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

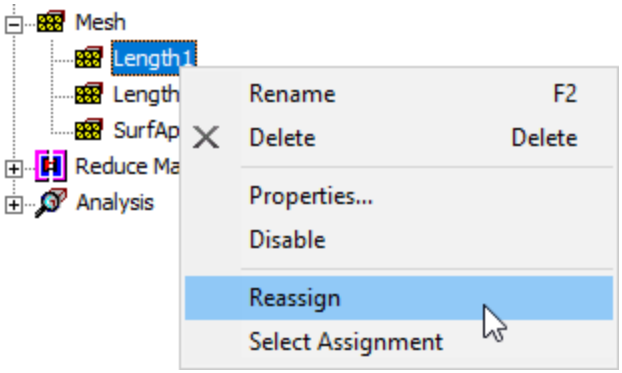


2. Click **Done** to close the window.

Reassigning a Mesh Operation

To reassign a previously assigned mesh setup:

1. Select the object(s) to which you want to reassign the mesh operation.
2. In the Project Tree, under **Mesh**, right-click the mesh refinement you had previously defined and select **Reassign**.



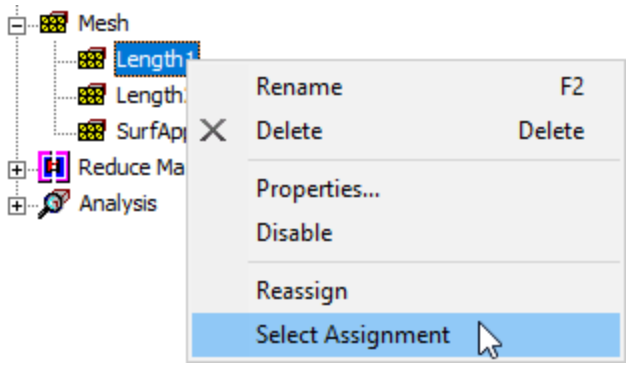
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The mesh operation is reassigned to the currently selected objects.

> **Note:**
>
> This pertains to mesh setup operations only. If you want to see information about a mesh that has been generated, see Viewing Mesh Statistics.

To view the objects to which a mesh operation is assigned:

1. In the Project Tree, select the mesh operation.
2. Right-click the previously completed mesh refinement and select **Select Assignment**.

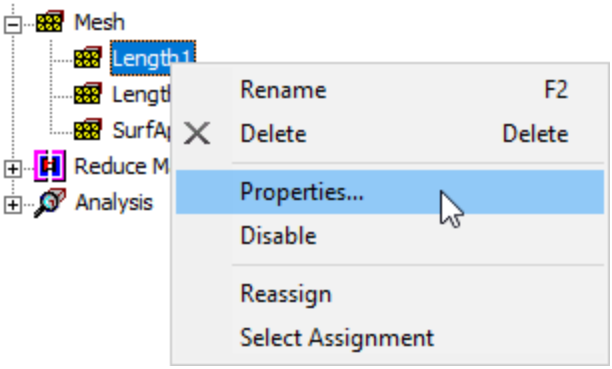


The objects to which the operation is assigned are selected in the modeling workspace.

Viewing Mesh Properties

To view mesh properties:

- In the Project Tree under **Mesh**, right-click a previously completed mesh refinement and select **Properties**.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A dialog box opens with properties specific to that mesh (length-based, surface approximation, etc.). You can edit the mesh settings as appropriate.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13 - Definition Libraries

This topic describes how to configure and manage **Electronics Desktop** definition libraries.

The typical Electronics Desktop user requires these library tools only rarely. The Desktop automates the creation of many kinds of components. This topic covers only the operations needed to create and maintain components like EM design components.

Related Topics

Working with Definition Libraries

Using the Library Editor

Using the Material Editor

Using the Script Editor

Using the Component Editor

Using the Padstack Editor

Using the Symbol Editor

Using the Footprint Editor

Vendor Library Components

Encrypted Libraries

Working with Definition Libraries

In Electronics Desktop, a component is a object placed in a design to be solved. Built-in Ansys components contain netlist and footprint information representing the native circuit and EM solver models. Other components can reference external models using frequency-dependent network data or other model formats.

The definition of a built-in component contains parameter data to pass to the solvers. The component definition also references separately stored data for the schematic symbol, fabrication material, layout footprint, and layer stackup definitions. These component dependencies are stored as objects in their own definition libraries.

System, User, and Personal Library Directories

The stock library files that ship with Electronics Desktop are stored under the **syslib** directory. These libraries are intended to be read-only and should not be modified.

In addition to the system libraries, Electronics Desktop recognizes two user-configurable library structures, called the **User Library** and the **Personal Library**. These are used to add foundry

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

support, user-defined models, and any custom or proprietary sets of components or simulation models. Customarily, **userlib** is a network repository for proprietary or corporate definitions available to all seats in an enterprise. **personalLib** contains project and circuit-specific libraries as needed by individual designs.

A root library directory is set up at installation. If none is specified, the default is the root Electronics Desktop directory.

Changing the Locations of Libraries

You can relocate the base library directory to a new valid library location. To specify a new library directory:

1. Click **Tools** > **Options** > **General Options**.
2. In the **Directories** group in the **Project Options** tab, type the new folder location in the **Library Directory** group box, or click **Browse ...** to specify it.
3. Click **OK**.

To return the library directory specification to its default value, click **Reset Library Directory**.

The personal library (**PersonalLib**) folder is located at **<Project Directory>\PersonalLib**, where **<Project Directory>** is the location you specified for your Electronics Desktop projects. To specify a new location for the **PersonalLib** folder, you must specify a new project directory location. To do this:

1. From the **Tools** menu, select **Options** > **General Options...**.
2. In the **Directories** group in the **Project Options** tab, type the new folder location in the **Project Directory** group box, or click **Browse ...** to specify it.
3. Click **OK**.

Library and Project Definitions

In Electronics Desktop, library component definitions are intended to be accessed once per component — when you place the first instance of a component onto a schematic. After it is placed in a schematic, the definition for the component transfers on the library to the project file. Editing and updating a component definition is then controlled on the *project definition*, which is listed in the **Components** folder in the **Project Manager** window. In other words, you do not edit component definitions in the library, but in the project. If you want your edits to be reflected in the library definition (for use in another design) you must export the edited components to a component library.

Once a definition is used, it is transferred to the current project, and remains in the project unless it is explicitly removed. To see the definitions included in a project, expand its **Definition** folders and subfolders in the **Project Manager** window:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modifying component and dependency definitions in libraries, or installing libraries with modified component and dependency definitions, does not automatically update those definitions in projects that contain them. To update project definitions from library definitions, see Updating Project Definitions from Library Definitions.

Updating Project Definitions from Library Definitions

Once a library component is placed into a design, the parent component definition resides in the project, and the link to the library is broken. To update a project component definition with a definition from a library file, select the project you want to update in the **Project Manager** window, then click **Project Tools** > **Update Definitions** on the **Tools** menu.

The **Update Definitions** window opens, listing any library definitions that have been edited since they are added to the project, and their original library locations. Use the window to select the item to update, and click **Update** to finish.

If you wish to remove project definitions that are unused, choose the **Project Tools** > **Remove Unused Definitions** command on the **Tools** menu which opens the Unused Definitions window.

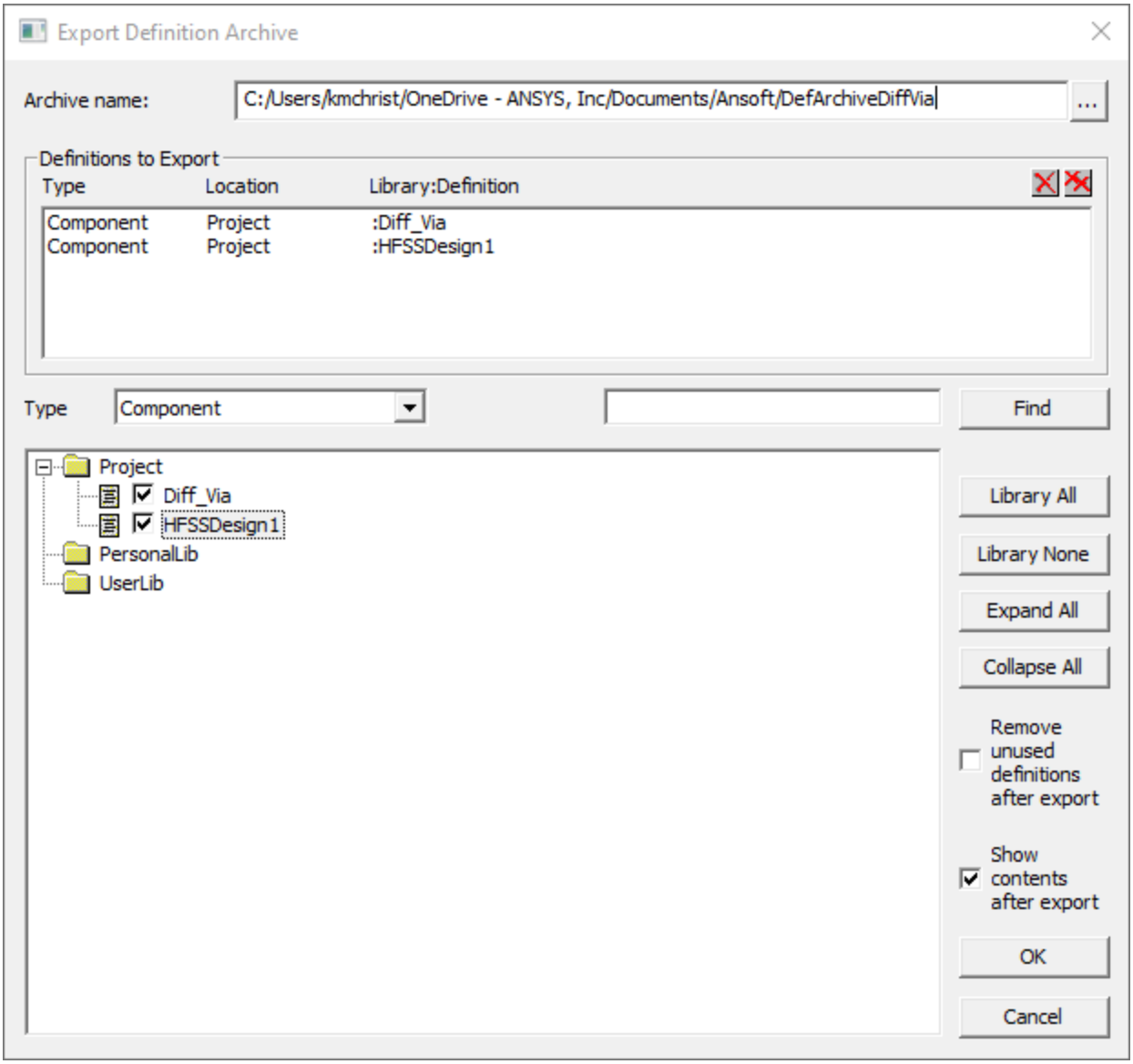
Exporting and Importing Definition Archives

The Import and Export Definition Archive commands allow you to select individual component, model, or other library definitions – or even entire libraries – to package and send to another user. Referenced definitions, data files, and DLLs are included automatically. All selected definitions (either those in an entire library or those selected separately) is written to one set of libraries (e.g., a component library and associated symbol and model libraries).

Exporting Definition Archives

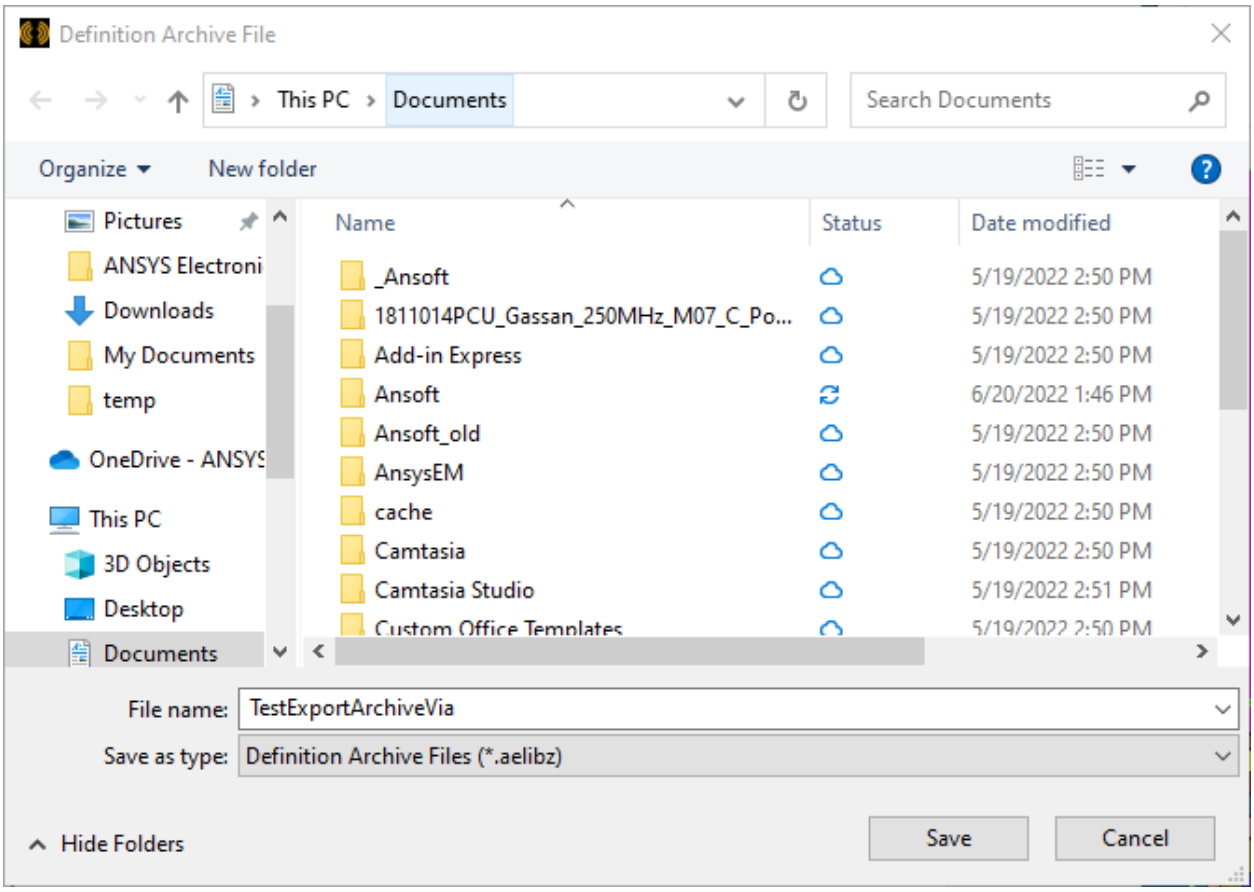
Select **Tools** > **Library Tools** > **Export Definition Archive** to open the **Export Definition Archive** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- To export a definition archive, first choose an archive name, either by typing the full filename in to the **Archive name** edit field, or by using the **[...]** button. This button opens a browser window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- The archive name must have a Definition Archive Files **.aelibz** extension, which is supplied by the window.

Note: An **.aelibz** archive is not readable with utilities such as WinZip®.

- The definitions selected for export are listed in the **Definitions to Export** listbox. Click the and buttons to delete selected entries, or all entries, respectively.
- The **Type** dropdown menu on the left, under the list of **Definitions to Export** controls which libraries are shown in the tree under it. Choices include: **Component**, **Symbol**, **Model**, **Package**, **Material**, and **Script**. In the tree, each leaf element (definition) has an associated check box. Checking the box adds the definition to the export list above. Clearing the box removes the definition on the export list. With a library or library element node highlighted, a user can select all definitions in that library for export by clicking **Library All**. Similarly, the user can unselect all definitions in the selected library by clicking **Library None**. **Expand All** and **Collapse All** expand or collapse the entire tree.
- Typing a string into the field next to the **Find** button allows the user to scroll to the first definition whose name matches the string. Searches are not case-sensitive. You can also use the “wildcard” asterisk character in search strings. For example: entering P* should

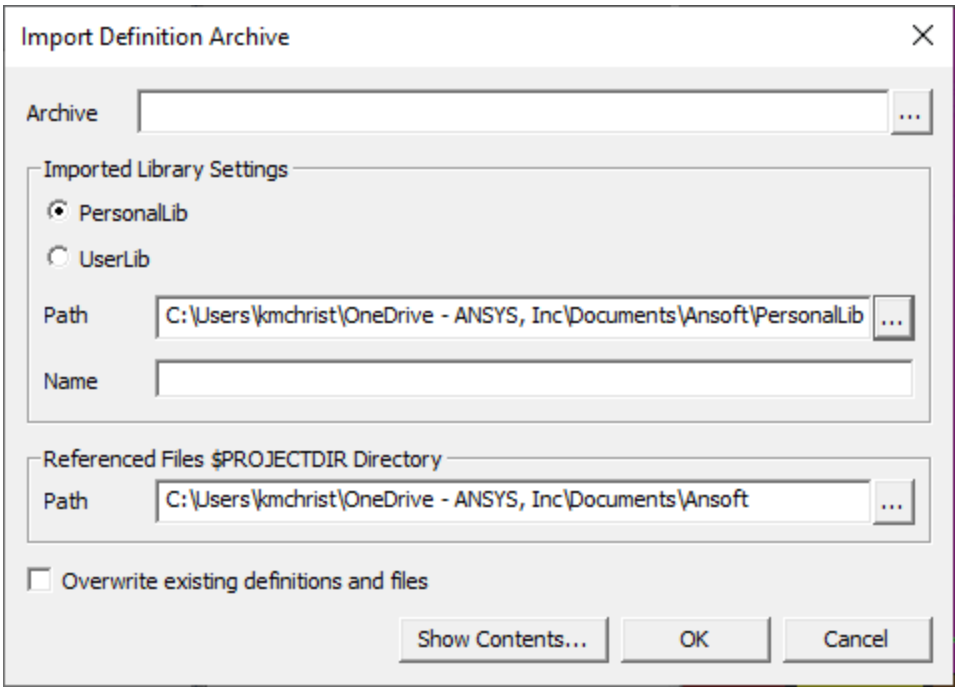
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

find, in succession, every component that starts with P (or p). Use multiple asterisks to further control matches (e.g., type *p* to find any definition with a 'p' in it, including as a first or last character; or *p*p* to find "pipe" and "pump", et cetera).

- The **Remove unused definitions after export** check box allows the user to clean up the current project after export is complete.
- When the **Show contents after export** box is checked, a **Contents** window shows a list of what was saved to the archive. This list may include multiple libraries (due to definition references) or external file references. Save the list to a file by clicking **Save , or** click **Done** when you are finished viewing.
- Click **OK** to complete the process. The selected definitions, libraries, and referenced files are zipped, and given a Definition Archive FIles extension, **.aelibz**. Users can then e-mail or otherwise transfer the archive file to others. The archive file is not encrypted, though models and packages contained in it may be.

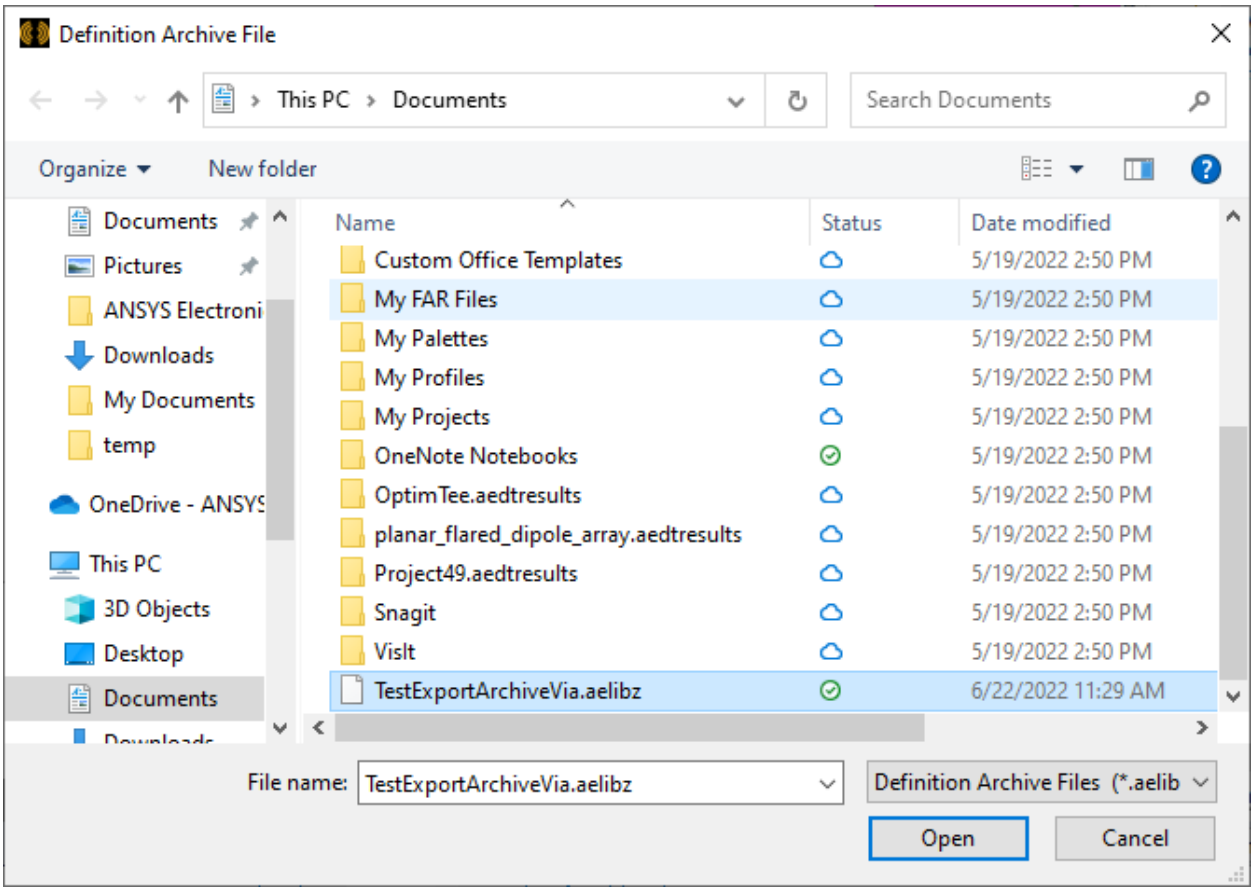
Importing Definition Archives

To import a definition archive, choose **Tools** > **Library Tools** > **Import Definition Archive** to open the **Import Definition Archive** window.



- First select a definition archive either by typing the path and definition archive file name in the **Archive** field, or click **[...]** to browse for the Definition Archive File created using the **Tools** > **Export Definition Archive** command.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- In the **Imported Library Settings** group box, click **PersonalLib** or **UserLib** to choose the basic location for the imported archive. If appropriate, the user can further choose a subdirectory **Path** under **PersonalLib** or **UserLib** as a location for the libraries. The user can also enter a **Name** for the imported libraries. The default name is the base name of the archive file.
- On import, data files is moved to appropriate **bin** and **data** directories. External **Referenced Files** that are not appropriate to **bin** and **data** library directories is referenced using **$PROJECTDIR**. The user can click the **[...]** button to choose the directory for these files. The default is the Project Directory specified in **Tools** > **Options** > **General Options**.
- The **Overwrite existing definitions and files** check box (unchecked by default) allows the user to replace current definitions and files with archive elements, permissions allowing.
- Click **Show contents** to view a window showing a list of what is in the archive. This list may include multiple libraries (due to definition references) or external file references. Save the list to a file by clicking **Save**, or click **Done** when you are finished viewing.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Managing Library Files

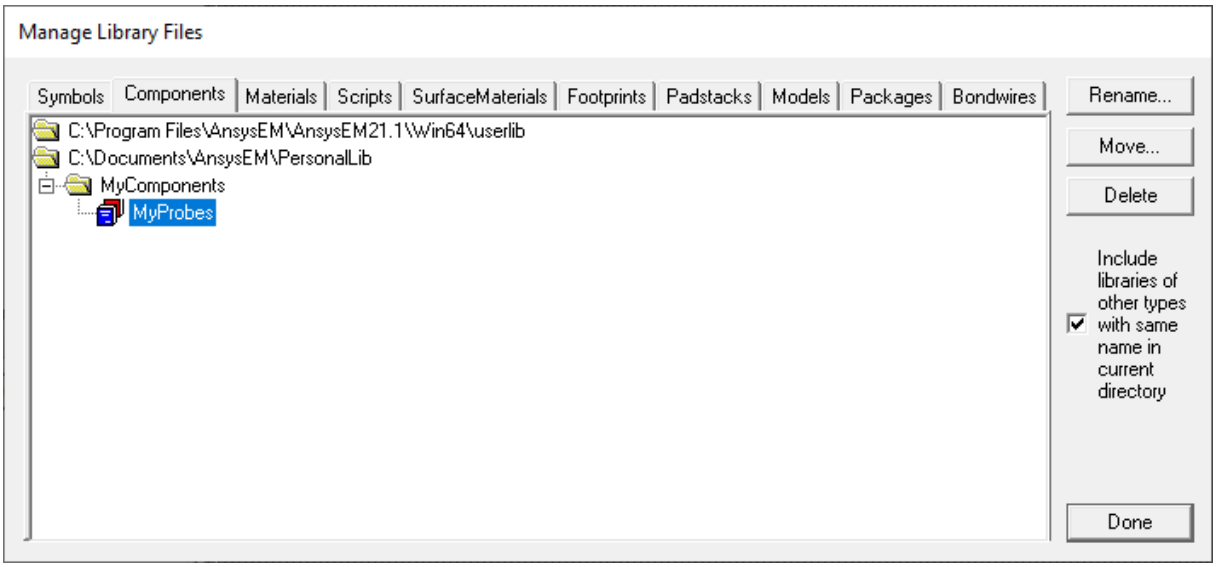
You can use **Manage Files** to change the names of libraries in the UserLib and PersonalLib directories; move libraries to another location in the directories specified for UserLib or PersonalLib, or delete libraries. You can also change the names of sub-directories of the UserLib and PersonalLib directories, move directories to another location in the directories specified for UserLib or PersonalLib, or delete directories.

> **Note:**
>
> - You cannot move libraries out of the UserLib or PersonalLib hierarchies.
> - To move a library or directory to a subfolder, that subfolder must already exist.

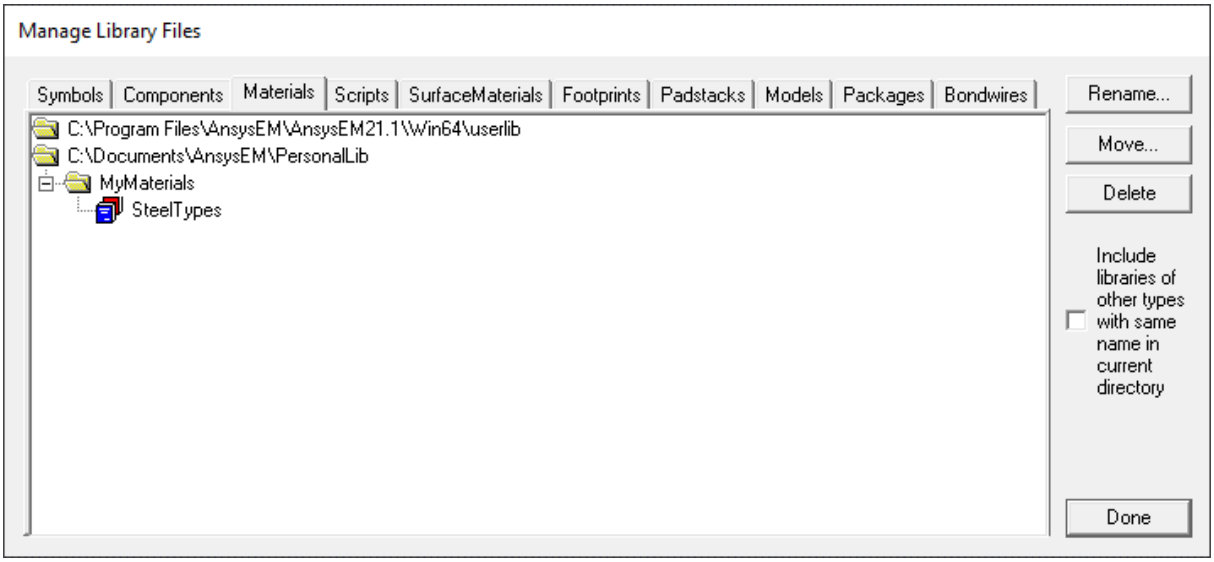
Components and other definitions in libraries directly affected by an action which See definitions changing library name or location has those references adjusted. Definitions in libraries indirectly affected– that is, those not having their names or locations modified by an action–with references to definitions in directly affected libraries are updated.

To manage library files:

1. Click **Tools** > **Library Tools** > **Manage Files**. The **Manage Library Files** window opens on the Components tab. The **Manage Library Files** window opens on the Components tab. There is a tab for each library type: Symbols, Components, Materials, Scripts, Surface Materials, Footprints, Padstacks, Models, Packages, and Bondwires.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click the tab you want.
3. Select an individual library or a directory on which you want to perform the action.
4. Click **Rename**, **Move**, and **Delete**, as necessary.
5. Select the **Include libraries of other types with same name in current directory** check box to cause the action to be performed on other types with the same name in the same directory. For example, if you select “MyProbes” on the Components tab, the action is extended to a symbol library called “MyProbes.aslb” in the same directory.
6. Click **Done** when finished managing libraries.

Library Search Precedence

When you place a new component in a design, Electronics Desktop searches in the current project for a definition with the corresponding name. If an appropriate definition exists in the project, that definition is used to satisfy the placement *regardless of whether additional instances, even more recent ones, may exist in external libraries.*

If Electronics Desktop cannot locate the required definitions in the project, it searches in personal (**PersonalLib**), user (**userlib**), and system (**syslib**) library files, *in that order*. Once a definition of the correct name has been located, the search ends, and that definition is used to satisfy the placement request.

Component Creation Sequence

Every component definition uses an associated bitmap (for the **Project Manager** window icon) and a schematic symbol. Many components also require footprints, which in turn depend on

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

material definitions and padstacks to characterize their physical presence in and electrical connections to a layout.

To save time in editing or creating new components, you should identify and modify or create its dependent elements in the following sequence as you define your component. All of the dependencies listed may not be necessary for every component you create or modify.

Materials

Material information defines the physical properties of a substance — such as substrate, metallization, or solder mask layer — for its inclusion in a *stackup*, the physical foundation of a manufacturable layout or model. Material definitions are stored in library files with the extension **.amat**. See Using the Material Editor for how to create and modify materials.

Padstacks

A padstack defines the physical structure and electrical connectivity of a *pad*, the region where a component pin connects to a copper trace. A padstack must be associated with each pin of a component. Padstack definitions are stored in library files with an extension of **.pslb**. See Using the Padstack Editor for how to create and modify padstacks.

Footprints

A footprint defines the planar space consumption, physical orientation, metal usage, and electrical connectivity of a component in a layout. Footprint definitions are stored in library files with the extension **.aflb**. See Using the Footprint Editor for how to create and modify footprints.

Symbols

A symbol is graphical representation of a component in a schematic. Symbol definitions are stored in library files with the extension **.aslb**. See Using the Symbol Editor for how to create and modify symbols.

Bitmaps

A bitmap is a small picture that is displayed when the ACLB containing the component is opened in the schematic editor. Bitmaps are stored in syslib\Bitmaps*.bmp. To create or modify a bitmap, use a bitmap editor such as MS Paint(r).

Components

The Electronics Desktop **Component Editor** supports the viewing, modification, and saving of information for component elements and graphical primitives. After creating or modifying a component, the information can be saved, used to render a component, or exported. Component definitions are stored in library files with an extension of **.asty**. See Using the Component Editor for how to create and modify components.

Scripts

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Electronics Desktop **Script Editor** supports the viewing, modification, and saving of script file information for footprint elements and graphical primitives. After modifying a script, the information can be saved, used to render a footprint, or exported. Script definitions are stored in library files with an extension of **.dsc**. See Using the Script Editor for how to create and modify scripts.

Using the Component Editor

The Electronics Desktop **Component Editor** supports the viewing, modification, and saving of component information for use in the development of circuit designs. You can create new components, edit existing components within a project file, and manage components within external libraries. After modifying a component, the information can be saved, used to define a design, or exported. Component definitions are stored in library files with an extension of **.aclb**. You can access component definitions using the **Component Editor**.

To start the **Component Editor**, open a project and select **Tools** > **Edit Libraries** > **Components**. When the Edit Libraries window opens, use its controls to open the Edit Component window, then create a new component, or select an existing one, and click **Edit** to open the **Component Editor**.

Creating and Editing Components

Use the Edit Component window to:

- Specify or change the component name, description, associated bitmap, and default property values, including those necessary for netlisting, co-simulation, and **Solver-on-Demand** operation, as appropriate
- Specify or respecify a graphical symbol to represent the component in the schematic editor
- Specify or change the component terminal properties
- Specify or respecify a footprint to represent the component in the layout editor

The changes you make when creating or editing a library component become part of the current project. The newly edited component is saved to the project, not back to the library. If the edited component is used in any schematics in the project, all default values of any modified parameters are updated immediately.

- To make a new or modified component available for use in other projects, save it to a component library (**.aclb**) file using the **Export** command in the Edit Libraries window.
- To update another project with your updated component(s), select its icon in the **Project Manager** window, then click **Project Tools** > **Update Definitions** on the **Tools** menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** If you are creating a new component, it is useful to first identify and, if necessary, create the necessary dependencies before defining your component. See Component Creation Process in the introduction for further information.

Related Topics

Editing an Existing Component

Creating a New Component

Editing an Existing Component

To edit an existing component, do either of the following:

- From the **Project Manager** window, expand the **Definitions** /**Components** subfolder for the project that contains the component you want to edit. Double-click the entry for the component you want to edit, or right-click the entry and select **Edit Component**.
- Open the **Tools** > **Edit Libraries** > **Components** window, then locate and select the component that you want to edit. Click **Edit Component**, or double-click the selected entry.

The Edit Component window opens, displaying the definition of the selected component for editing.

Creating a New Component

To create a new component, do either of the following:

- Select **Tools** > **Edit Libraries** > **Components**, and click **Add Component** to open the **Edit Component** window.
- Open the **Tools** > **Edit Libraries** > **Components** window, then locate and select an existing component on which to base your new component. Click **Clone Component**. and a renamed copy of the selected component is added to the list. Double-click the entry for the new component, or click **Edit Component** to open the **Edit Component** window.

The Edit Component Window

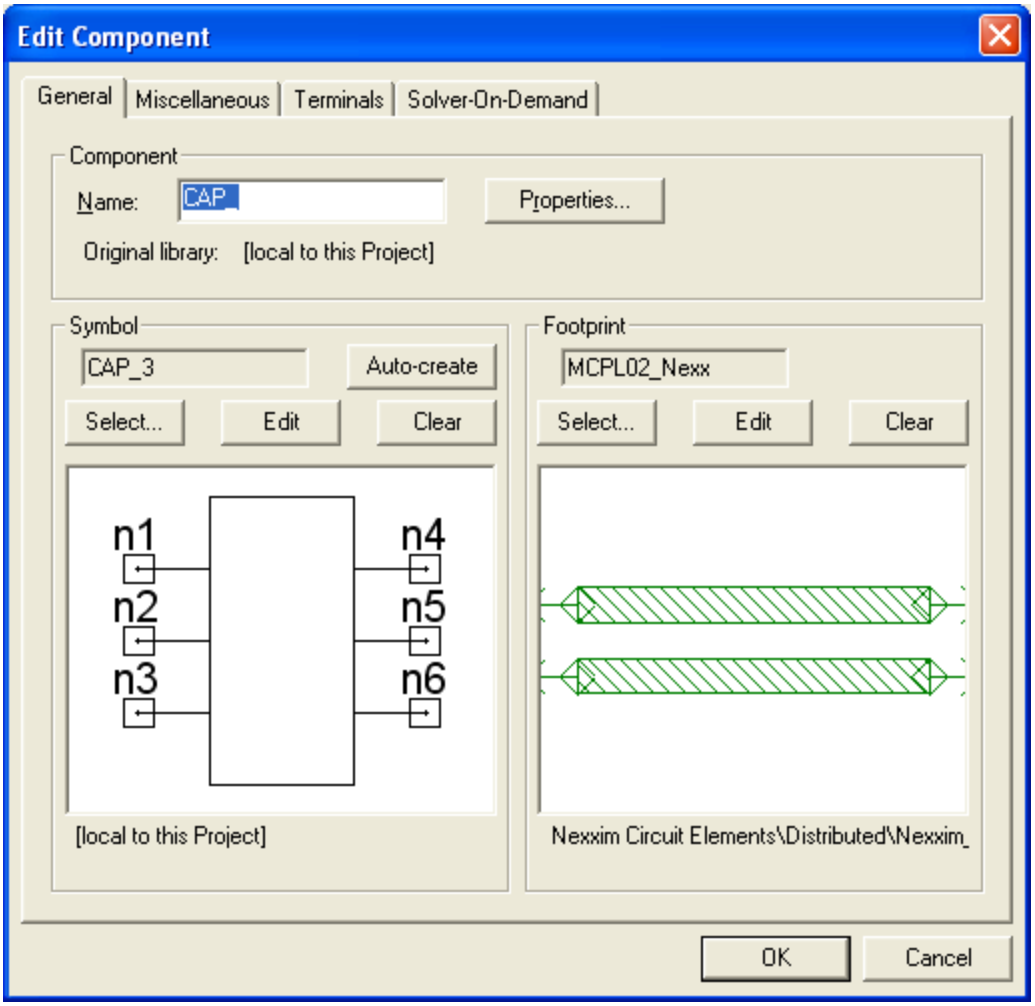
The **Edit Component** window is used to create new components, or edit existing components, in a project file. To show the **Edit Component** window, do either of the following:

- Double-click a component definition in the project tree, or double-click a component listed in the **Edit Library** window.
- Open a project and select **Tools** > **Edit Libraries** > **Components**. When the Edit Libraries window opens, use its controls to open the **Edit Component** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Components General Tab

Click the **General** tab to specify all parameter values for the component and assign symbols and footprints on their respective libraries. You can also start the symbol and footprint editors on the window, however, you must exit the component editor to complete the symbol or footprint edits. The symbol and footprint preview windows at the bottom of the window are automatically updated to reflect whatever changes you make to the symbols and footprints you modify.



Component Panel

The Component group box of the General Tab window displays the component Name. The Original Library name is displayed if found. To access and edit the component properties, click **Properties** next to the **Name** text window. In the Properties window, add, edit or remove default or local properties and values.

Symbol Panel

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Symbol group box of the General Tab window allows you to search by symbol name, and also displays a picture of the symbol. Choose any symbol which has an equal or greater number of pins than the component specified in the Component Name field. You can then choose to **Auto-create** the component.

The library of origin of the component is displayed beneath the component display in the **Symbol** area; if the component is **local to this Project**, that message is displayed. When you click **Select** (**...**), the Select Definition window opens and allows you to choose a definition for the symbol from a list that is displayed. Click **Edit** to modify the symbol. A confirmation message is displayed with a reminder that the Component and Definition editors is closed (if open) and that all changes are saved if you choose to continue. Click **Clear** to clear all selections and displays in the Symbol area.

Footprint Panel

The Footprint group box of the General Tab window allows you to search by footprint name, and is used to display a picture of the footprint. Choose any footprint which has an equal or greater number of pins than the component specified in the Component Name field.

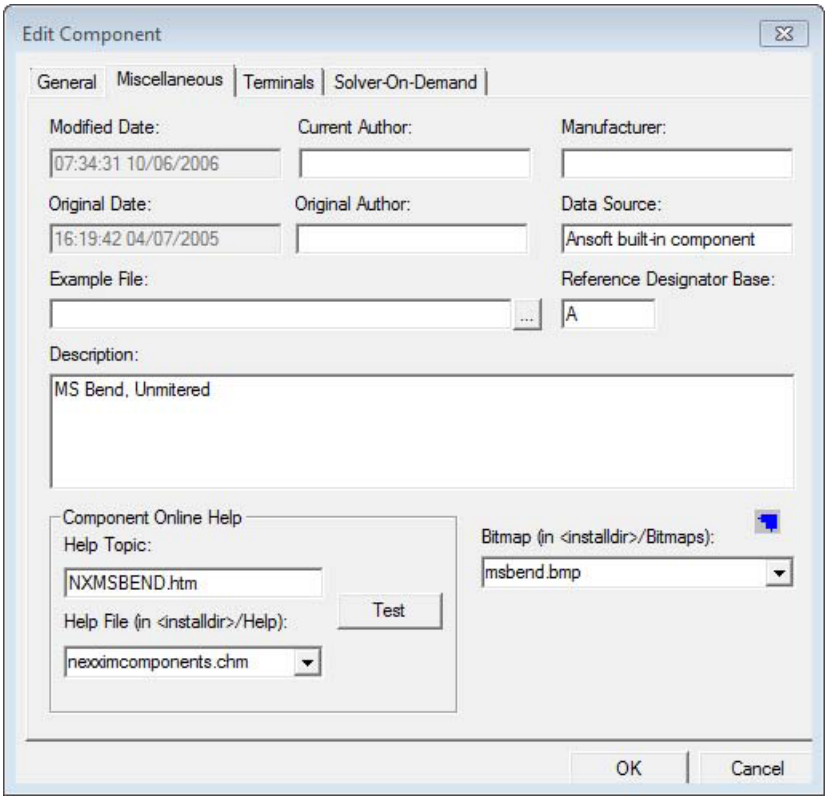
The library of origin of the footprint is displayed beneath the component display in the Footprint area; if the component is **local to this Project**, that message is displayed. When you click **Select**, (**...**) the Select Definition window opens and allows you to choose a definition for the footprint from a list that is displayed. Click **Edit** to modify the footprint. A confirmation message is displayed with a reminder that the Component and Definition editors is closed (if open) and that all changes are saved if choose to continue. Click **Clear** to clear all selections and displays in the Footprint area.

Click **OK** to save your changes and close the window, or click **Cancel** to close the window without saving any changes.

Components Miscellaneous Tab

Use the miscellaneous fields to enter component information such as the name, manufacturer, description, ref des, etc. You can also specify a bitmap for the **Project Tree** and an associated help description (if you use your own documentation). Help files must be located in **<Installation Directory>\Electronics Desktop\help**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The top two rows of the **Miscellaneous** tab window displays the **Modified Date** and **Original Date** of the component, and allows you to specify the following:

- Current Author
- Manufacturer
- Original Author
- Data Source

You may also specify an **Example File** directory for the component, browse for the directory if necessary (...) and specify its **Reference Designator**, or enter a text **Description** for the component. The default **Bitmap** of the component is displayed at lower-right, and you may use the adjacent drop-down menu to specify a different bitmap file.

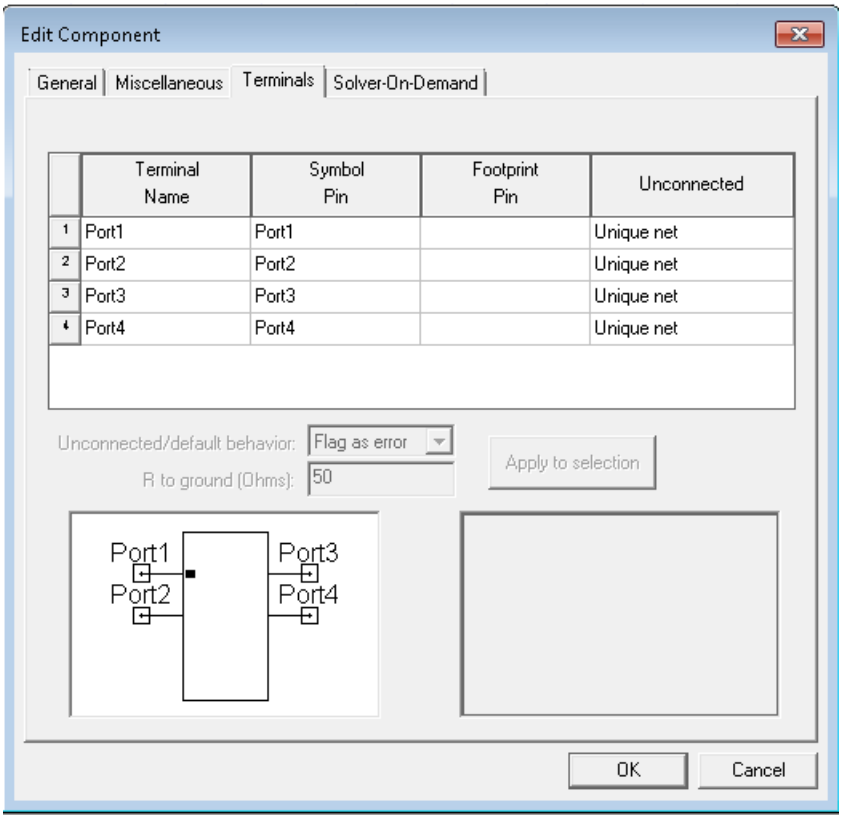
The **Component Help** area displays the default HTML help file associated with the component, and allows you to specify an alternative help file of your own. You may also **Test** the specified help file, which displays the component's HTML help page.

Click **OK** to save your changes and close the window, or click **Cancel** to close the window without saving any changes.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Components Terminals Tab

To line up component terminals in an envisioned order, Click the **Terminals** tab to associate each symbol pin with the proper terminal by first ordering the symbol pins, then selecting the corresponding symbol. This allows you to visually associate the symbol pins with the netlist properties for the simulation model. The symbol and footprint preview windows at the bottom of the window are automatically updated to reflect whatever changes you make to the symbols and footprints you modify.



The following controls are available:

In the **Terminals** display window, click one or more terminals directly, which then enables the following window options:

Unconnected/default behavior – Used to specify the response when a pin is not connected to a wire or port in a schematic. Choose between the following drop-down menu actions: **No action** (default), **Flag as error**, and **Grounded**. Click **Applytoselection** to apply your changes to the selected terminals.

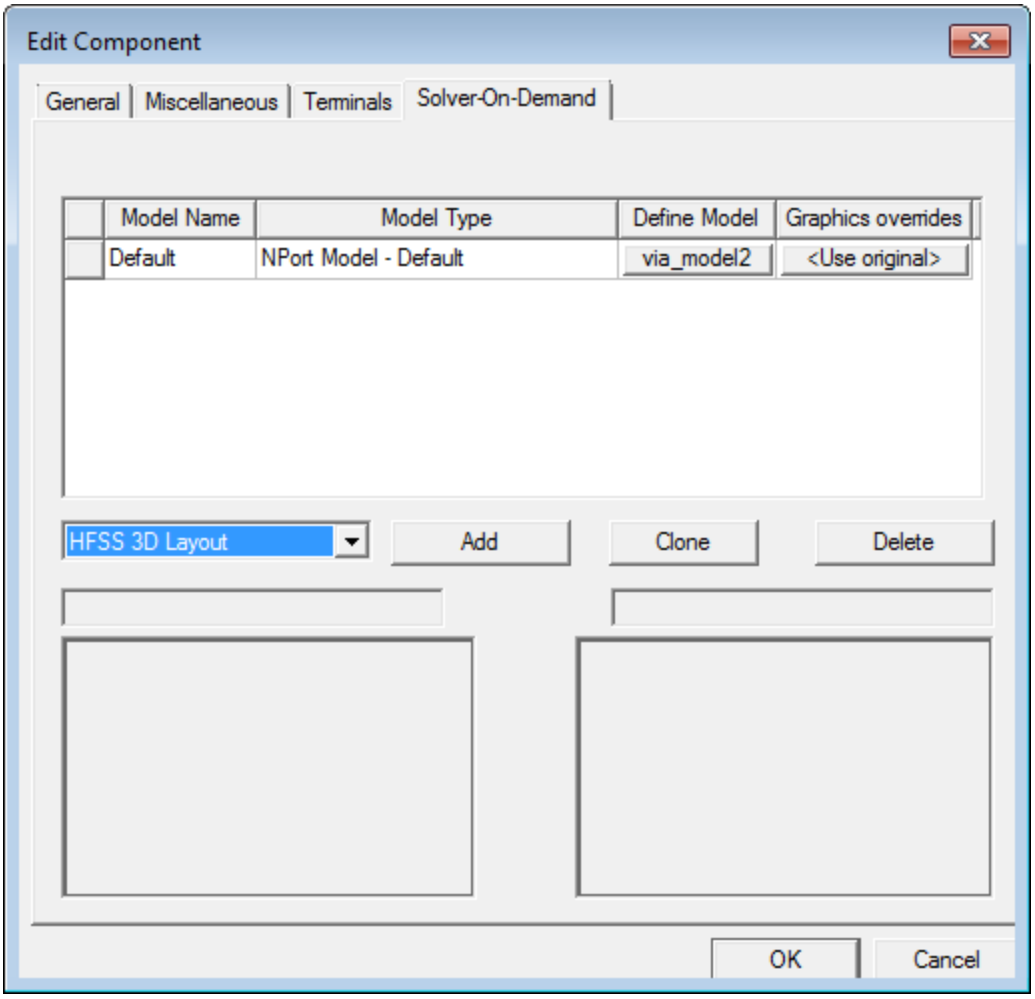
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

R to Ground – If you select **Grounded** for the **Unconnected/default** behavior, then specify the resistance to ground in the **R to Ground** entry window. Click **Applytoselection** to apply your changes to the selected terminals.

Click **OK** to save your changes and close the window, or click **Cancel** to close the window without saving any changes.

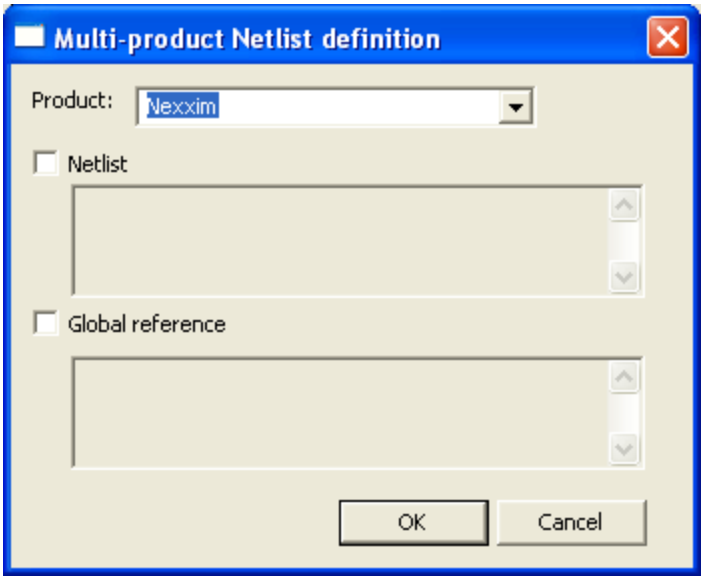
Components Solver On Demand Tab

Solver-on-Demand is a specialized routine to set up co-simulation using multiple products. When you first open the **Solver-on Demand** tab of the **Edit Component** window for a particular component, the only **Model Name** listed is the default multi-product type, **DefaultNetlist**.



DefaultNetlist is a default co-simulation definition type, provided automatically for each component, that allows you to define a different netlist for each Electronics Desktop product. Click the button in the **DefaultNetlist** row to open the **Multi-product Netlist definition** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Click the drop-down menu **Product** menu to choose which Electronics Desktop product netlist to define.
- Click **Netlist** to define a netlist.
- Click **Global reference** to define a global reference.
- Click **OK** to implement your changes, close the window, and return to the **Edit Component** window.

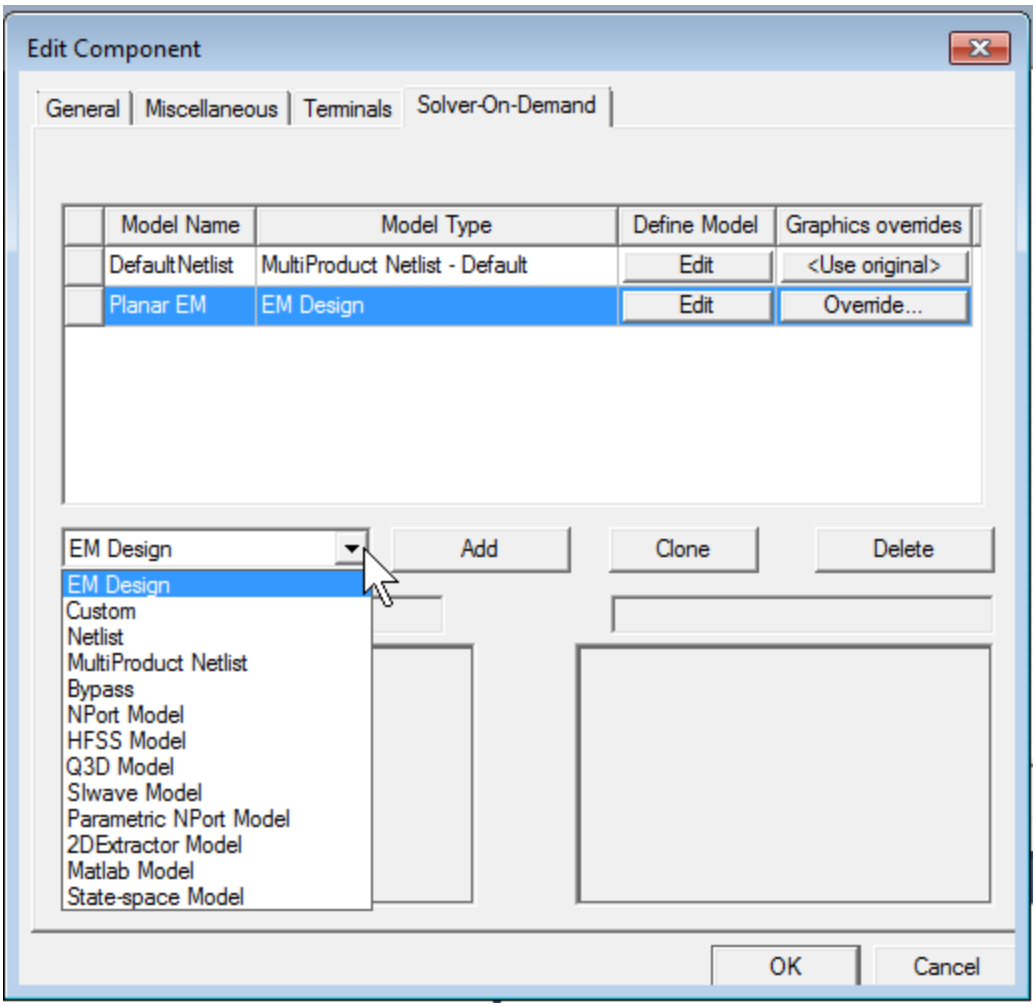
Multiple Co-simulation Models

Electronics Desktop allows for multiple representations of models that can be cosimulated. In the **Solver-on-Demand** tab of the **Edit Component** window, click in the drop-down menu to choose from a list of models to add to the list of **ModelNames**.

To add a model for Solver On Demand:

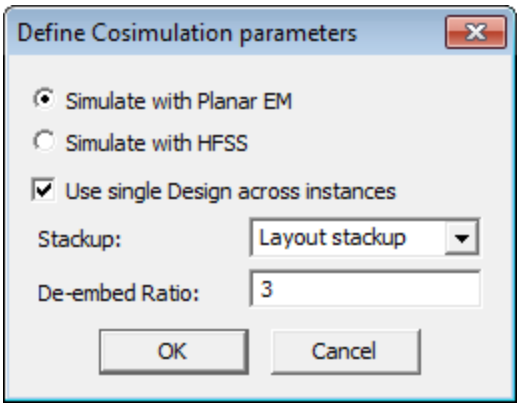
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Make a selection on the model type drop-down menu.



2. Click **Add**. A new model of the selected type is added to the list. The symbol and footprint preview windows at the bottom of the window are automatically updated to reflect whatever changes you make to a component's symbol/footprint definitions.
3. Click in the **ModelName** field for the new model and type the name you want to use for this model (or accept the default in the **ModelName** field). Note that when you highlight a **Model Name**, its symbol and footprint (if defined) are automatically previewed in the windows at the bottom of the window. These preview windows are automatically updated to reflect whatever changes you make to the component's symbol/footprint definitions.
4. Click **Edit** in the **Define Model** field (or with a **Custom** model name) to open a **Cosim** or **Netlist** definition window that is specific to the model type. Here is the window for the EM design model shown in the MSBEND component:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



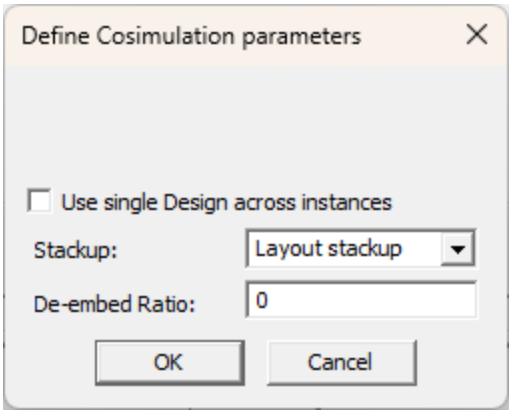
5. Repeat steps 1 through 3 to create all the models you wish to use for this type of component.
6. Click **OK** when you have defined all the models for this component type.

Repeat the above procedure to specify models for all components that is simulated using **Solver-on-Demand**.

For a complete description of the Solver On Demand tab and how to define model types, see Co-simulation.

HFSS 3D Layout as Solver On Demand

HFSS 3D Layout is the available field solves that simulates from a layout, substrate, or footprint defined in the component itself. From the **Solver-on-Demand** tab of the **Edit Component** window, select **HFSS 3D Layout** from the drop-down menu and click **Add** to open the co-simulation definition window.



- **Stackup** – Set to one of three values (Layout, Substrate, or Footprint) to determine which stackup definition are used to model the component when it is simulated by the selected solver. For more information see Stackup Definitions.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

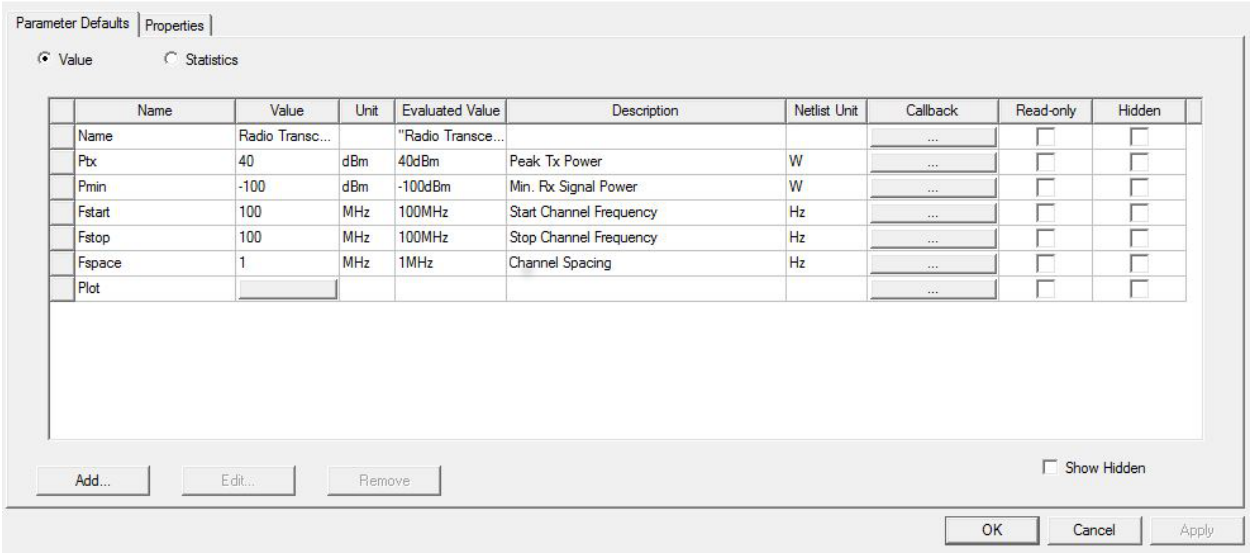
Solver-on-Demand with the HFSS 3D Layout tool requires the component possess a physical definition, in the form of a footprint. Many components in the Ansys Electronics Desktop® library have footprints defined so they can be simulated using HFSS 3D Layout as the **Solver-on-Demand**. When you define a component, it can be simulated using **Solver-on-Demand** as long as the component possesses a physical definition. For more information see Physical Definition.

- **De-embed Ratio** – Determines the manner in which excitation sign als are fed to the component via transmission lines during EM simulation. Short sections of transmission line are automatically added to the ports of the **Solver-on-Demand** component to allow discontinuities near the ports to decay in the EM simulation. The length of these transmission lines is equal to the De-embed Ratio times the width of the **Solver-on-Demand** port. That is, Length = De-embed Ratio * W (port connection). The solution data is automatically de-embedded back to the initial **Solver-on-Demand** component geometry to remove the effects of the additional transmission lines.
- **Use single Design across instances** enables the same design to be used for all instances of the component type, eliminating the need to specify the **Solver-on-Demand** model individually.

Click **OK** to save your changes and close the window, or click **Cancel** to close the window without saving any changes.

Edit Component Properties window

To open the **Properties** window for a selected component, double-click the component definition in the **Project Manager** window to open the **Edit Components** window. Then, in the **Edit Components** window, click **Properties** to open the window.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

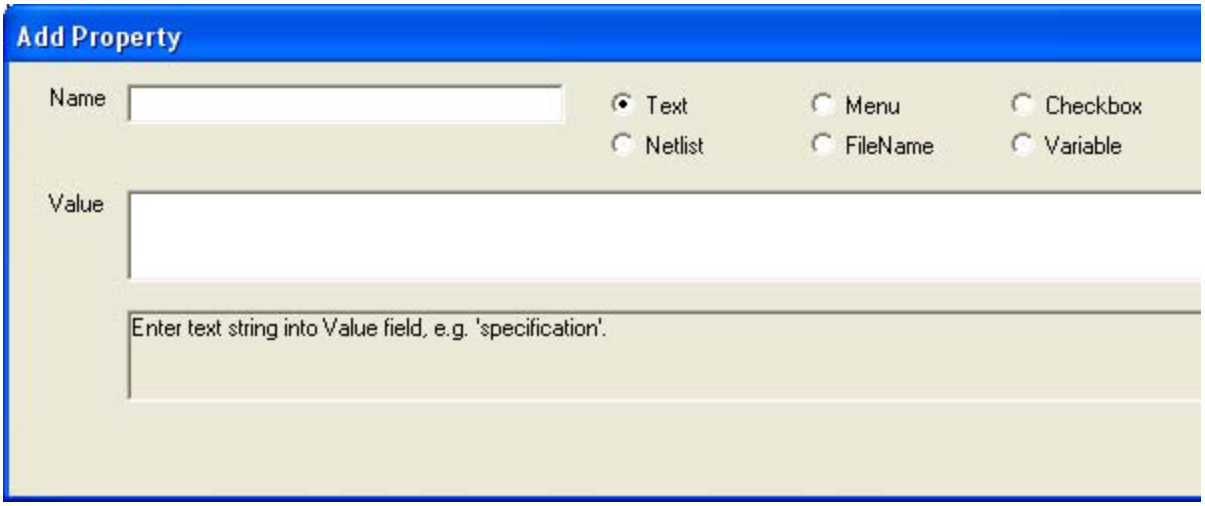
Parameter Defaults Tab

This tab displays and sets the default values for component parameters for general analysis (Value display) or statistical analysis (Statistics display).

When this option is selected, the **Parameter Defaults** tab lists, displays, and sets the default values for component parameters during general analysis.

Adding Parameters

Use **Add** to add a parameter to the definition. You are prompted to define the parameter name, value, and value type (text input, menu, check box, etc.). You may **not** add reserved system parameters to the definition.



Removing Properties

To remove a property on a component, select the property and click **Remove**. The property is deleted on the component definition (model) and from all instances in the design.

> **Warning:**
>
> Removing a property from a component from an Electronics Desktop component library may produce undesirable results when the component is simulated.

Related Topics

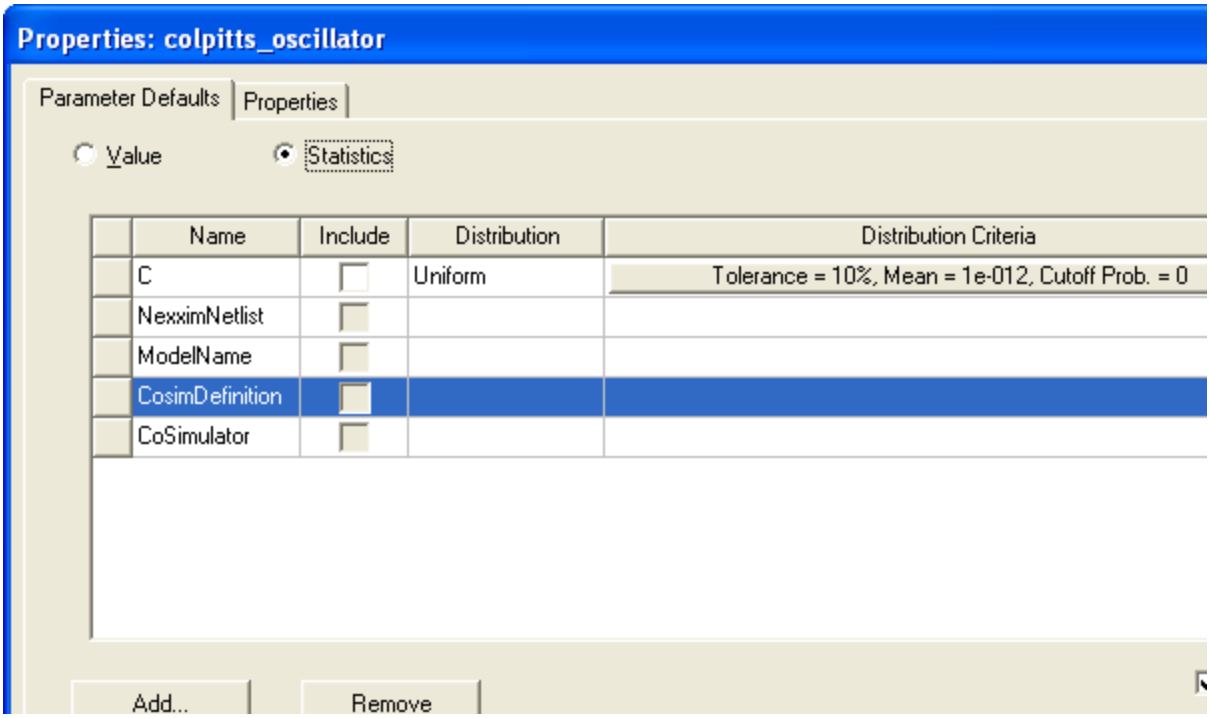
Parameter Statistics Display

Reserved Component Parameters

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameter Statistics Display

When this option is selected, the **Parameter Defaults** tab lists, displays, and sets the default statistics for component parameters during general analysis.



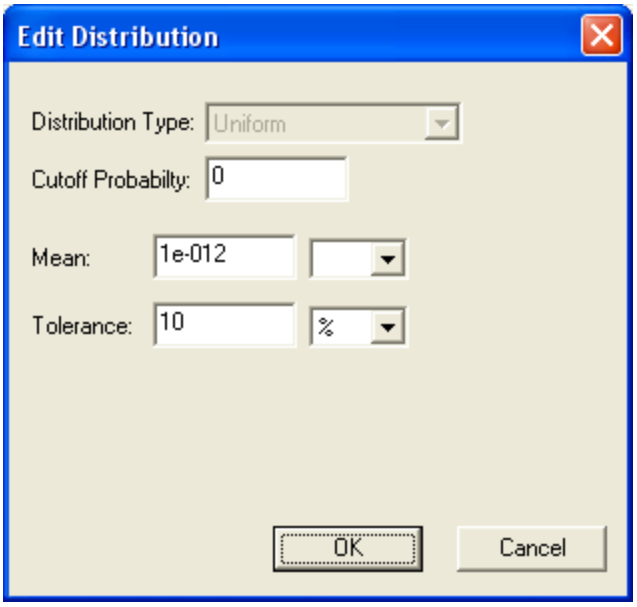
Name —Displays the names of component parameters.

Include —Controls whether or not the associated component parameter is varied during statistical analysis. Check **Include** for each parameter you want Electronics Desktop to vary during statistical analysis.

Distribution —Displays and controls whether the parameter value distribution is Uniform or Gaussian. To change the Distribution setting, click in the cell to open the options, then click the option you want.

Distribution Criteria —Click the display bar to open the **Edit Distribution** window. The **Distribution Type** drop-down menu displays whether the distribution is Uniform or Gaussian, and you can specify values for **Cutoff Probability**, **Mean**, and **Tolerance**. Units for **Mean** and **Tolerance** can be set using their adjacent dropdown menus.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Click **OK** to save your changes and close the window, or click **Cancel** to close the window without saving any changes.

Reserved Component Parameters

When using **Add** to add a parameter to the component definition in the **Properties** window, you are prompted to define the parameter name, value, and value type (text input, menu, check box, etc.). You may **not** add the following reserved system parameters to the definition:

"Id"

"Status"

"AnalysisSetup"

"AnalysisResults"

"PartName"

"Info"

"NPortData"

"Refbase"

"NumParts"

"CompName"

"Description"

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

"Manufacturer"

"Datasource"

"Date"

"Symbol"

"Footprint"

"PinCount"

"Refdes"

Component CosimDefinition Property

A netlist is a textual representation of a circuit. The Electronics Desktop simulators extract a netlist on the schematic and perform the simulation on the netlist. The Nexxim simulator uses the Nexxim netlist format. Nexsys components within a Nexxim design use a legacy Ansys netlist format for components.

When an analysis is requested for a modified design, or when **Browse Netlist** is invoked, Electronics Desktop writes the corresponding netlist to the project results folder. The netlist conveys to the simulation engine the connectivity, parameter values, analysis specifications, and other characteristics of the design(s) under analysis. To support this process, every Electronics Desktop component must include a valid definition for the **CosimDefinition** property.

To view the **CosimDefinition** for a component:

1. Open its **Properties** window.
2. Select the **Parameter Values** tab.
3. Click **Edit** for **CosimDefinition**.

The **Multi-product Netlist definition** window opens, displaying the netlist string. Use the window to edit the string if appropriate.

Related Topics

Netlist String Syntax

Netlist String Syntax

The general form for a netlist string is:

<Simulator>Netlist=[netlist_string]

where

<Simulator> is Nexxim | Q3D | etc.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The netlist string may contain the following netlist property syntax:

%<terminal index, 0-based>

Which specifies the name of the net connected to terminal index. A syntax error results if the terminal index is not a positive number, within range. Example:

%0, %1

%_<internal node index, 0-based>

Create a unique net for an internal node. Used when a component has nodes that are not associated with symbol terminals and are not tied to a global node. Syntax error if node index is not a positive number, within range. Example:

%_0, %_1

@<propname> | @(<propname>)

Value of property named propname. Syntax error if propname is empty or contains spaces. Example:

@R, @IDSS

?<propname>(<expr1>)[:(<expr2>)]

If property named propname exists, substitute expr1, else expr2. Also, expr1 and expr2 may contain additional substitutions. Syntax error if propname is empty or contains spaces. Example:

?IDSS(IDSS = @IDSS):(IDSS = 0.05)

~<propname>(<expr1>)[:(<expr2>)]

If property named propname doesn't exist, substitute expr1, else expr2. Syntax error if propname is empty or contains spaces. Example:

~SUB(SUB = MS)

?(<propname>==<value>)(<expr1>)[:(<expr2>)]

?(<propname>!=<value>)(<expr1>)[:(<expr2>)]

~(<propname>==<value>)(<expr1>)[:(<expr2>)]

~(<propname>!=<value>)(<expr1>)[:(<expr2>)]

Same as "?" and "~" above except the first term is evaluated for equal to (==) or not equal to (!=) and if true, substitute expr1 else expr2. Example:

?(sim==fullwave)(NSUM=@NSUM):(F0=@F0)

***<propname>(<expr1>)**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If property named propname has changed from default definition value, then substitute expr1. Syntax error if propname is empty or contains spaces. Example:

*IDSS(IDSS = @IDSS)

&(<expr>)[^(pname1,pname2, . . .)]

Add all properties that have changed from default definition value, except those in the optional exclusion list. In expr, “$” can be used to represent the property name, and “#” the property value. Example:

&($=#)^(Model)

netlists all properties that changed on their default value except the Model property.

\n

A new line marker to inform the netlister to insert a new line. Example:

R@ID %0 %_0 @R \n C@ID %_0 %1 @C

****

The backslash is used as an escape character. The character following the backslash is not processed but added to the netlist string as is. To add a single “\”, use “\\”. Example:

R:@ID %0 %1 R=\{\2*@R\}

Defined Variables

- $SYSLIB is expanded to **<InstallationDirectory>/syslib** upon netlisting, where **<Installation Directory>** is the directory into which Electronics Desktop was installed during setup.
- $USERLIB is expanded to **<InstallationDirectory>/userlib** upon netlisting.
- $PERSONALLIB is expanded to **<ProjectDirectory>/PersonalLib** upon netlisting, where **<ProjectDirectory>** is the location you specified for your Electronics Desktop projects.

Global Reference String in Electronics Desktop

A global reference property processes its value the same way as the netlist string and places the result in the top-level (global) part of the circuit file.

<Simulator>GlobalRef=[string]

where

<Simulator> is Nexxim | Q3D | etc.

Additional Examples

The following additional examples help to demonstrate netlist string syntax.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NexximNetlist= CPLE@ID %0 %1 %2 %3 ?Z(Z=@Z) ?E(E=@E) ?F(F=@F) *A(A=@A)

“Z”, “E”, and “F” use the **"?"** conditional because the component definition values assigned to these parameters are different from those in the engine and should always be netlisted if they are present even if the user has not changed them. The “A” parameter uses the **"*"** conditional because its component definition value matches that of the engine and it needs to be netlisted if the user changes the instance value.

NexximNetlist= R@ID %0 %1 @R ?TC1(TC1=@TC1 ?TC2(tc2=@TC2)) ?TJ(TJ=@TJ) ?TNOM(TNOM=@TNOM)

In this example, TC1 is netlisted only if TC1 exists, and TC2 is netlisted only if TC1 and TC2 exist.

NexximNetlist= MSBENDO@ID %0 %1 w=@Length sub=@Substrate

“w” and “sub” is netlisted.

NexximNetlist= MOSFET@ID %0 %1 %2 %3 MODEL=@Model &($=#)^(Model)

Model is netlisted. The “&($=#)^(Model)” argument designates that all parameters should be netlisted as “name=value”, except for “Model”.

NexximNetlist = D@ID %0 %1 1N914

NexximGlobalRef = .LIB $SYSLIB/Vendor/D.lib

which may netlist to:

D22 net_3 net_4 1N914

< rest of design netlists ...>

.LIB $SYSLIB/Vendor/D.lib

NexximNetlist= R@ID %0 %1 R=\{\2*@R\}

If ID is 1 and R is 100, the netlist is:

R1 net_0 net_1 R={2*100}

Using the Symbol Editor

A symbol is a set of information that defines the graphical representation and electrical connectivity of a component. Symbol definitions are stored in library files with an extension of **.aslb**.

You can access material definitions using the **Symbol Editor**. The Electronics Desktop **Symbol Editor** allows you to view, modify, and save symbol information for components and layout

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

primitives. After modifying a symbol, the information can either be saved, used to define a design, or exported.

When a symbol is used in a project, it is copied to the project. Thus new projects do not list any symbols in the Project Manager. If a change is made to the symbol, it is actually made to the project's copy of the symbol. So each project has its own set of symbols based on AEDT's symbol library. When you open an existing project, the Project Manager displays the symbols in use in the project.

To start the **Symbol Editor**, open a project and select **Tools** > **Edit Libraries** > **Symbols**. When the Edit Libraries window opens, use its controls to open the **Symbol Editor**.

Creating a New Symbol

1. Click **Tools** > **Edit Libraries** > **Symbols**.
2. From the **Symbols** tab of the **Edit Libraries** window, either:
   - Click **Add Symbol** to open the **Get Name** window.
   - Select an existing symbol to base your new symbol and click **Clone Symbol(s)** to open the **Get Name** window.
3. Type a name for the new symbol into the **Enter the name for this new Symbol** field and click **OK**.
4. Then double-click the entry for the new symbol or click **Edit Symbol** to open the symbol editor.
5. Use the symbol editor to:
   - Draw graphical primitives, such as rectangles, circles, and arcs, using options on the Draw menu.
   - Add pins for electrical connections using the Pin option on the **Draw** menu.
   - Modify pin properties in the Pin List dialog box.
   - Add labels. To include component properties or other variables in a label, use the Property Display Setup option on the Symbol menu. For unchanging text, use the text option on the Draw menu. Unusual labels are most likely represent a component property.
   - Update the current project with the new or revised symbol definition using the Update Project option on the **Symbol** menu

   You can also export a symbol to a symbol library (**.aslb**) file for use in other projects. For information on how to do this, see the Edit Libraries.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Editing an Existing Symbol

When a symbol is used in a project, it is copied to the project from a global library. Any changes made to the symbol are made to the project's copy of the symbol and only affect the symbol in the project and not in any other project. To edit an existing symbol, either:

- From the **Project Manager** window, open the project that contains the symbol you want and open **Definitions** > **Symbols**. Either double-click the symbol you want to edit or right-click the symbol and select **Edit Symbol**.
- Open **Tools** > **Edit Libraries** > **Symbols**. From the **Symbols** tab of the **Edit Libraries** window, either double-click the symbol that you want to edit or right-click the symbol and choose **Edit Symbol**.

The symbol editor opens with the selected symbol ready for editing.

Use the symbol editor to:

- Draw graphical primitives, such as rectangles, circles, and arcs, using options on the Draw menu.
- Add pins for electrical connections using the Pin option on the **Draw** menu.
- Modify pin properties in the Pin List dialog box.
- Add labels. To include component properties or other variables in a label, use the Property Display Setup option on the Symbol menu. For unchanging text, use the text option on the Draw menu. Unusual labels are most likely represent a component property.
- Update the current project with the new or revised symbol definition using the Update Project option on the **Symbol** menu

You can also export a symbol to a symbol library (**.aslb**) file for use in other projects. For information on how to do this, see the Edit Libraries.

Symbol Editor Operations

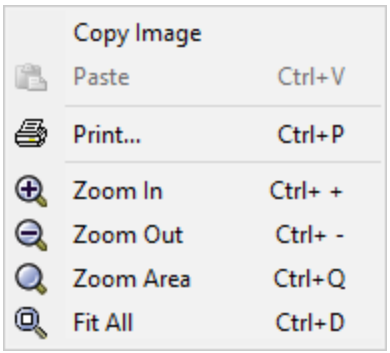
The symbol editor opens when you begin to create or modify a symbol. Like the **Schematic Editor**, the symbol editor is a graphical tool that keeps the placement of symbol elements uniform by snapping them to points on a grid. The window controls are the same as those in the schematic editor: press the arrow keys and **Page Up** and **Page Down** to pan, or **Ctrl** +**D** to fit the view.

With the symbol editor, add, modify, and manipulate graphical primitives, text labels, pins, and property displays. You can also adjust the resolution, color, and visibility of the editor grid. When you're done, update the current project with any changes you've made to the current symbol. Options related to these operations are available on the Symbol menu and the Symbol Draw menu.

Symbol Editor Shortcut Menus

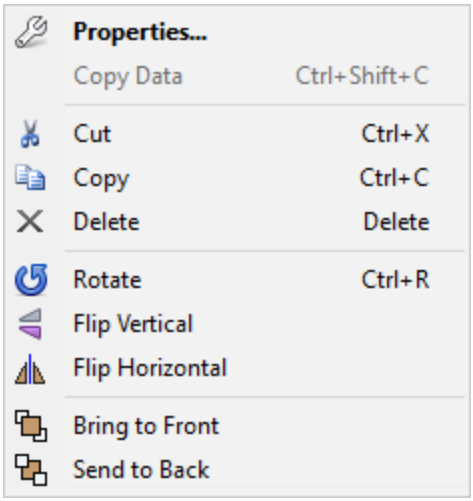
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you right-click in the symbol editor grid, this shortcut menu appears:



These options are a subset of those available on the **View** menu.

When you right-click an object in the symbol editor, this shortcut menu appears:



Click **Properties** to view the properties of the selected object. The other options are a combination of entries on the **Edit**, **View**, and **Draw** menus.

Symbol Draw Menu

The symbol editor **Draw** menu presents options for drawing graphical primitives and text, and manipulating selected objects. The **Draw** menu options include

- Arc
- Circle
- Line
- Polygon

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Rectangle
- Text
- Image
- Ellipse
- Curve
- Pin
- Rotate
- Align Horizontal
- Align Vertical
- Flip Vertical
- Flip Horizontal
- Bring to Front
- Send to Back

Arc

This option, also available on the **Symbol Draw** toolbar , initiates creation of an arc. To draw an arc, click the symbol editor grid at the two points that determines the arc's ends, then drag the arc to the appropriate radius.

Once you have created an arc, adjust its radius or move its endpoints by clicking the arc to select it, and dragging the appropriate handle.

Circle

This option, also available on the **Symbol Draw** toolbar , initiates creation of a circle. To draw a circle, click the symbol editor grid to select a center point, then drag the circle to the appropriate diameter.

Once you have created a circle, adjust its diameter in either of two ways:

- Click the circle to select it, then drag one of its handles.
- Double-click the circle to open its **Properties** window, type a new value for the **Radius** parameter, then click **OK**.

By default, a new circle is hollow. You can fill a circle with solid color or parallel lines in one of several styles as follows:

a. With the circle's properties displayed, click in the **Value** cell for the **FillStyle** property.
b. Select the appropriate fill style on the list.
c. Click **OK**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Line

This option, also available on the **Symbol Draw** toolbar , initiates creation of a polyline with one or more segments. To draw a line:

1. Click the symbol editor grid where you want the line to start, then click at one or more points to continue the line.
2. To complete the line, do one of the following after defining its final segment:

- Press the **Spacebar**.
- Right-click, then click **Finish**.

After you have completed a line, change the endpoints of its segments by doing one of the following:

- Select the line and drag the appropriate handle(s).
- Double-click the line to open its **Properties** window, type new position values (in the form X: Y) for the appropriate vertices, and click **OK**.

Alternately, change the endpoints of a selected line by clicking **Layout** > **Line Styles** to open the **Line Styles** window, then click in the Cap Type field to change the cap or line-ending type (Flat, Extended, or Round).

Because a line has nonzero width, its CapType determines how its ends behave:

- **Extended** – Extends the line by its own width past the endpoint you click.
- **Flat** - Cuts the line perpendicularly at the endpoint you click.
- **Round** - Ends the line in a semicircle.

The existing technology files define line/trace styles, including a CapType (typically Flat) for each style. If you select None as the technology type, you'll automatically get a palette of four styles with a variety of widths, BendTypes and CapTypes.

Polygon

This option, also available on the **Symbol Draw** toolbar , initiates creation of a polygon. To create a polygon:

1. Click the symbol editor grid to specify the position of one vertex, then click wherever you want to place additional vertices.
2. To complete a polygon, specify the position of its final vertex, then do either of the following:

- Press the **Spacebar**.
- Right-click, then click **Finish**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Once you have drawn a rectangle, Edit its vertex positions and other properties, including its fill style, as follows:

- Click the rectangle, then edit its properties in the **Properties** window.
- Double-click the rectangle, then edit its properties in the **Properties** window.

By default, a new polygon is hollow. You can fill a polygon with solid color or parallel lines in one of several styles as follows:

a. With the polygon's properties displayed, click in the **Value** cell for the **FillStyle** property.
b. Select the appropriate fill style on the list.
c. Click **OK**.

Rectangle

This option, also available on the **Symbol Draw** toolbar , initiates creation of a rectangle. To create a rectangle, click the editor grid to specify the position of one corner, then drag the rectangle to the appropriate size.

Once you have drawn a rectangle, Edit its height and width, the position of its center, and its angle (its rotation, in degrees, relative the handles of its bounding box) as follows:

- Click the rectangle, then edit its properties in the **Properties** window.
- Double-click the rectangle, then edit its properties in the **Properties** window.

By default, a new rectangle is hollow. You can fill a rectangle with solid color or parallel lines in one of several styles as follows:

1. With the rectangle's properties displayed, click in the **Value** cell for the **FillStyle** property.
2. Select the appropriate fill style on the list.
3. Click **OK**.

Text

This option, also available on the **Symbol Draw** toolbar , adds to the symbol an editable "Default text" label in 12-point Arial. To edit the default text string immediately after placement, type the text you want, then press **Enter** or click elsewhere in the editor grid.

To change just the text of an existing label:

1. Click the label.
2. Click the label again to open its text for editing.
3. Type the appropriate text.
4. Press **Enter** or click elsewhere in the editor grid.

To change other properties of a label, including its color, font or size:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click the label and view its properties in the **Properties** window, or double-click the label and view its properties in the **Properties** window.
2. Click in the **Value** cell for the property you want to modify.
3. Modify the value.
4. Click **OK**, or click in another **Value** cell to commit the change and keep editing values.

Image

This option, also available on the **Symbol Draw** toolbar , adds an image to the symbol. To add an image, select an image file in the **Select Image** window and click **Open**. Click the editor grid to place the image, then drag the mouse until the image is the appropriate size, and click the mouse again.

Once you have placed an image, Edit its height and width, the position of its center, and its angle (its rotation, in degrees, relative the handles of its bounding box) as follows:

- Click the image to select it, then drag one of its handles.
- Click the image and edit its properties in the **Properties** window.
- Double-click the image and edit its properties in the **Properties** window.

To flip the image horizontally, select the image and in the **Properties** window, click **Mirrored**.

By default, a new image is not linked its original file. To ensure that the image is updated when changes are made to the file, select the image and in the **Properties** window, click **Link to File**.

A new image does not have a border. Add one as follows:

1. With the image's properties displayed in the **Properties** window, click the **Value** cell for the **DisplayBorder** property.
2. Set a **BorderWidth**.
3. Click the **Value** cell for the **BorderColor** property to add a color in the **Color** window.
4. Click **OK** to close the **Color** window.

Ellipse

This option, also available on the **Symbol Draw** toolbar , initiates creation of an ellipse. To draw a ellipse, click the editor grid to specify the top and left most position of the shape , then drag the ellipse to the appropriate size.

Once you have drawn an ellipse, edit its height and width, the position of its center, and its angle (its rotation in degrees relative the handles of its bounding box) as follows:

- Click the ellipse to select it , then drag one of its handles.
- Click the ellipse and edit its properties in the **Properties** window.
- Double-click the ellipse and edit its properties in the **Properties** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

By default, a new ellipse is hollow. You can fill an ellipse with solid color or parallel lines in one of several styles as follows:

1. With the ellipse's properties displayed, click in the **Value** cell for the **FillStyle** property.
2. Select the appropriate fill style on the list.
3. Click **OK**.

Curve

This option, also available on the **Symbol Draw** toolbar , initiates creation of a polycurve with one or more segments. To draw a curve:

1. Click the symbol editor grid where you want the curve to start.
2. Clicke or more points to continue the curve.
3. To complete the curve, mark its final segment and do one of the following:
    - double-click.
    - Press the **Spacebar**.
    - Right-click and click **Finish**.

After you have completed a curve, change the endpoints of its segments by doing one of the following:

- Select the curve and drag the appropriate handle(s).
- Double-click the curve to open its **Properties** window, type new position values (in the form X: Y) for the appropriate vertices, and click **OK**.

Pin

This option, also available on the **Symbol Draw** toolbar, initiates placement of a symbol pin with a default stem length of 10. You can rotate a pin once you've started to place it by iteratively pressing R until the pin is oriented to your liking. Click in the editor grid to finish placing the pin.

Rotate

This option, also available on the **Symbol Draw** menu and by pressing **Ctrl**+**R** on the keyboard, rotates a selected object or group of objects 90° to the left.

Align Horizontal

This option horizontally aligns the uppermost edges of the members of a group of selected objects with the uppermost edge of the first-selected object. To align multiple objects horizontally:

1. Press **Ctrl**, then click the object with which you want to align the others.
2. Still pressing **Ctrl**, click the additional objects in turn to add them to the selection.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that the first-selected object is highlighted in red, and that the subsequently selected objects are highlighted in dark red.

3. From the **Draw** menu, select **Align Horizontal**.

Align Vertical

This option vertically aligns the leftmost edges of the members of a group of selected objects with the leftmost edge of the first-selected object. To align multiple objects vertically:

1. Press Ctrl, then click the object with which you to align the others.
2. Still pressing Ctrl, click the additional objects in turn to add them to the selection.

   Note that the first-selected object is highlighted in red, and that the subsequently selected objects are highlighted in dark red.

3. From the **Draw** menu, select **Align Vertical**.

Flip Vertical

This option, also available on the **Symbol Draw** toolbar, flips a selected object about the X axis.

Flip Horizontal

This option, also available on the **Symbol Draw** toolbar, flips a selected object about the Y axis.

Bring to Front

This option, also available on the **Symbol Draw** toolbar, places a selected object in front of all other elements in the editor, i.e., closest to the user.

Send to Back

This option, also available on the **Symbol Draw** toolbar, places a selected object behind all other elements in the editor, i.e., furthest on the user.

Symbol Menu

The Symbol menu lists options for symbol naming, property displays, and file operations. Its entries include:

Update Project

This option updates the current project with changes made in the current symbol.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note: Update Project** updates your symbol changes to the current project in memory and does not save your changes to disk. To save your changes to disk, you must also click **Save** on the **File** menu. If you like the changes you have made in a symbol to be available for use in other projects, you must export the symbol to a library as described in the Edit Libraries topic.

Set Name

This option opens a **Properties** window to rename the current symbol.

Property Display Setup

This option opens the Properties Display so add or modify display of the names and/or values of properties associated with the component represented by the current symbol.

Pin List

This option gives you access to settings related to pin properties by opening the Pin List window.

Grid Setup

This option gives you access to settings related to the symbol editor grid resolution, color, and visibility by opening the symbol editor Grid Setup window.

Import File

The Import File command allows you to import an existing SVG-formatted symbol into the symbol editor.

Export File

This option saves the current symbol as a Microsoft Enhanced Meta File (**.emf**) graphic.

Edit Component

This option launches the Edit Component window to modify the component associated with the active symbol.

Normalize Symbol on Save

Allows you to normalize the symbol you are editing when you save after making modifications.

List

Opens the **Design List** window, which lists the objects in the layout. Each column may be sorted by clicking on the heading. One or more object rows can be selected and operated by clicking **Delete** and **Properties**.

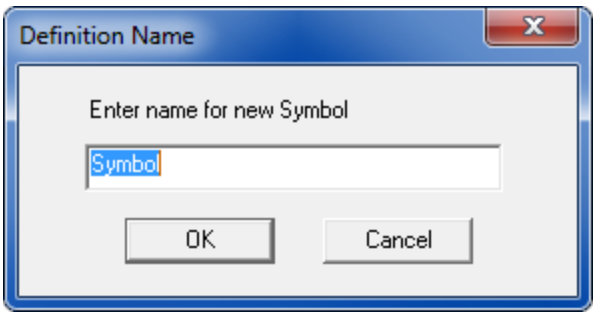
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Import File (Symbol)

The **Symbol** > **Import File** command allows you to import an existing SVG-formatted symbol into the symbol editor. The SVG (Scalable Vector Graphics) format is an open, xml-based web standard graphics format supported by many third-party tools.

To import an SVG (**.svg**) file when creating a new symbol definition do the following:

1. From the **Project Manager** window, right-click **Definitions** > **Symbols** in the appropriate project and select **Add Definition**.
2. In the **Definition Name** window, enter a name for the new symbol and click **OK** to open a new blank **Symbol Editor** window.



3. From the main menu bar, select **Symbol** > **Import File** and choose an existing **.svg** symbol file to import it into the symbol editor. A window opens advising that importing the **.svg** file removes any existing symbol graphics in the editor, and asking for confirmation to continue the import.

> **Note:**
>
> - There is no “Undo” for the import operation. Consequently if you make a mistake on import, you must close the project without saving, then reopen it to recover.
> - Currently **.svg** symbols can be imported into any symbol editor window - including those containing pre-existing component symbols in which case the current symbol in the editor is replaced by the imported symbol, so ensure that you import the symbol in the appropriate place.

4. Click **OK** to import the **.svg** symbol for further editing, or **Cancel** to abort the operation. Once the new symbol has been created, attach it to a component via the Property Display Setup command.

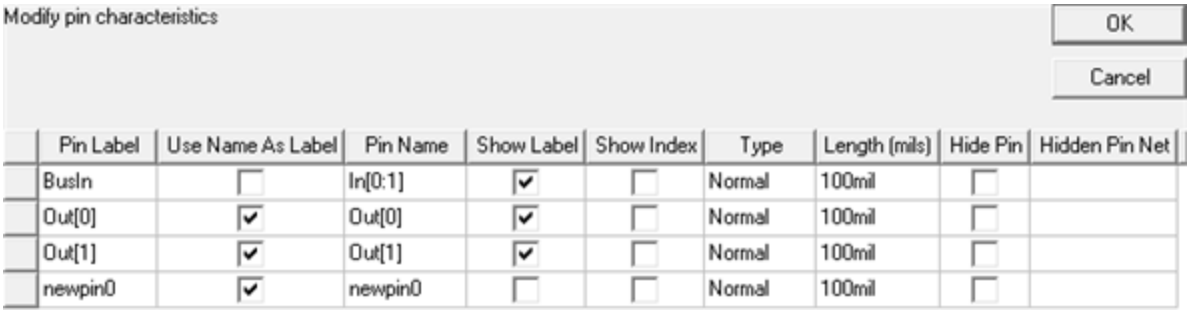
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Editing Pin Properties

To edit a property of a symbol pin, open the symbol editor's **Pin List** window by doing either of the following:

- In the **Symbol** menu, select **Pin List**.
- From the **Symbol** toolbar, click the **Pin list** icon .

The **Pin List** window displays and sets values for the properties of symbol pins.



Pin properties include:

- **Pin Label** – provides a field to add a user-defined label for the chosen pin on a component symbol without having to change the actual **Pin Name** associated with the chosen pin. Pin labels for pins in a component should be unique.

  To specify or change the label of a pin, click in the **Pin Label** cell for that pin, type the name, and do either of the following:

  - Press **Enter**.
  - Click in another cell.

  The **Pin Label** can be displayed for the chosen pin by unchecking **Use Name As Label**.

- **Use Name As Label** – controls whether the **Pin Name** or **Pin Label** is displayed for the chosen pin when **Show Label** is enabled. For a new pin, **Use Name As Label** is checked by default.
- **Pin Name** – sets the actual pin name. To specify or change the name of a pin, click in the **Pin Name** cell for that pin, type the name, and do either of the following:
  - Press **Enter**.
  - Click in another cell.
- **Show Label** – controls visibility of the either the **Pin Name** or **Pin Label** text for the pin (as determined by **Use Name As Label**). For a new pin, it is unchecked by default.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Show Index** – controls visibility of the pin number in the schematic editor. For a new pin, it is unchecked by default.
- **Type** – drop-down menu sets the pin type. Options include **Normal** (the default), **ANSI In**, **ANSI Out**, and **Zero Length**. To change a pin's type, click in its **Type** cell, then click the appropriate option.
- **Length (mils)** – sets the length of the pin stem (the graphical line associated with the pin port symbol). By default, new pins have a pin stem length of 100 mils. To change the length of a pin, click in its **Length** cell, type a new value, and do either of the following:
  - Press **Enter**.
  - Click in another cell.
- **Hide Pin** – controls whether or not a pin is visible in the schematic editor. For new pins, it is unchecked by default.
- **Hidden Pin Net** – editable when **Hide Pin** is checked, specifies the net (circuit node) to which a pin is connected if it is hidden. To specify a net, click in the **Hidden Pin Net** cell, type a net name, then do either of the following:
  - Press **Enter**.
  - Click in another cell.

Edit Symbol Pin Locations

To edit the location of a symbol pin, open the **Edit Symbol Pins** window by doing one of the following:

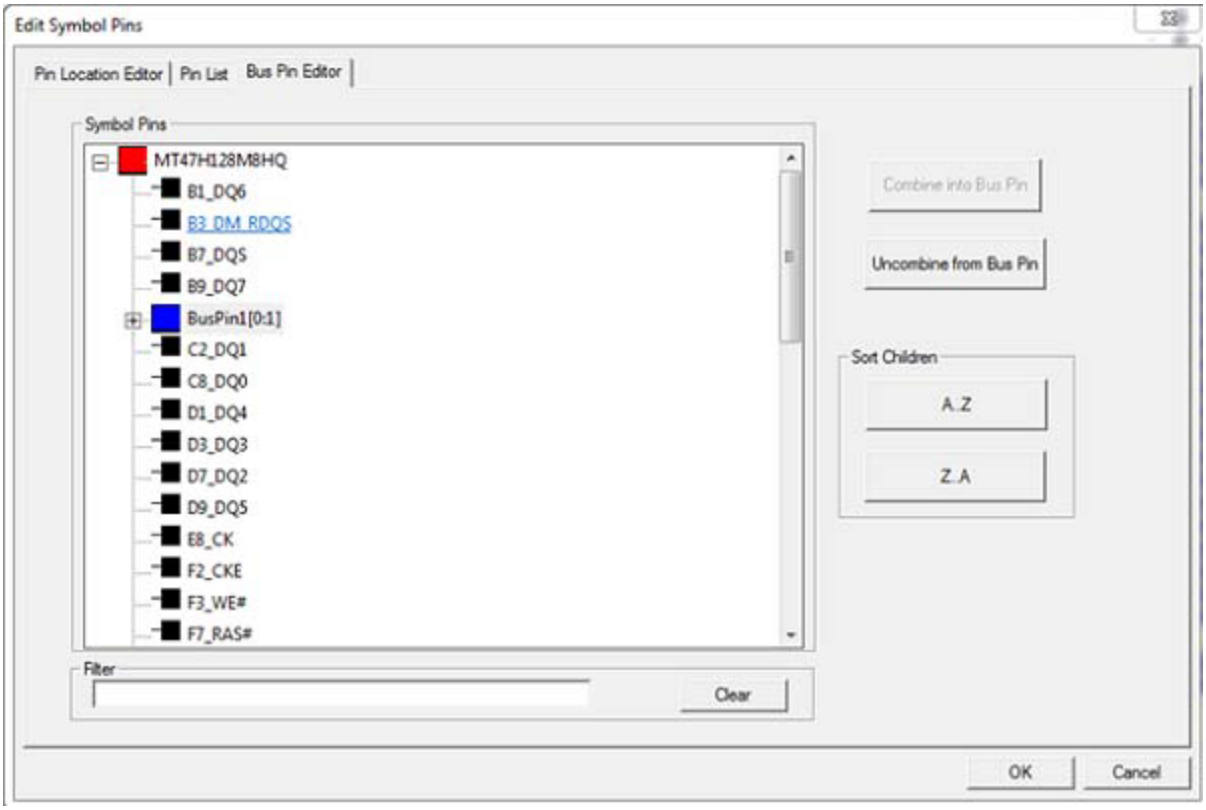
- In the Symbol Editor **Symbol** menu, select **Edit Pins**.
- In the component instance shortcut menu, select **Edit Symbol Pins**.

The window has three tabs: **Bus Pin Editor**, **Pin Location Editor**, and **Pin List**. Bus Pin Editor and Pin Location Editor appear only for symbols that are drawn as a rectangle with pins.

Bus Pin Editor

The **Bus Pin Editor** tab allows you to create, edit, and delete bus pins from regular pins. The pin names are the terminal names associated with the component model and cannot be changed, but the bus pin names can be anything the user wants.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



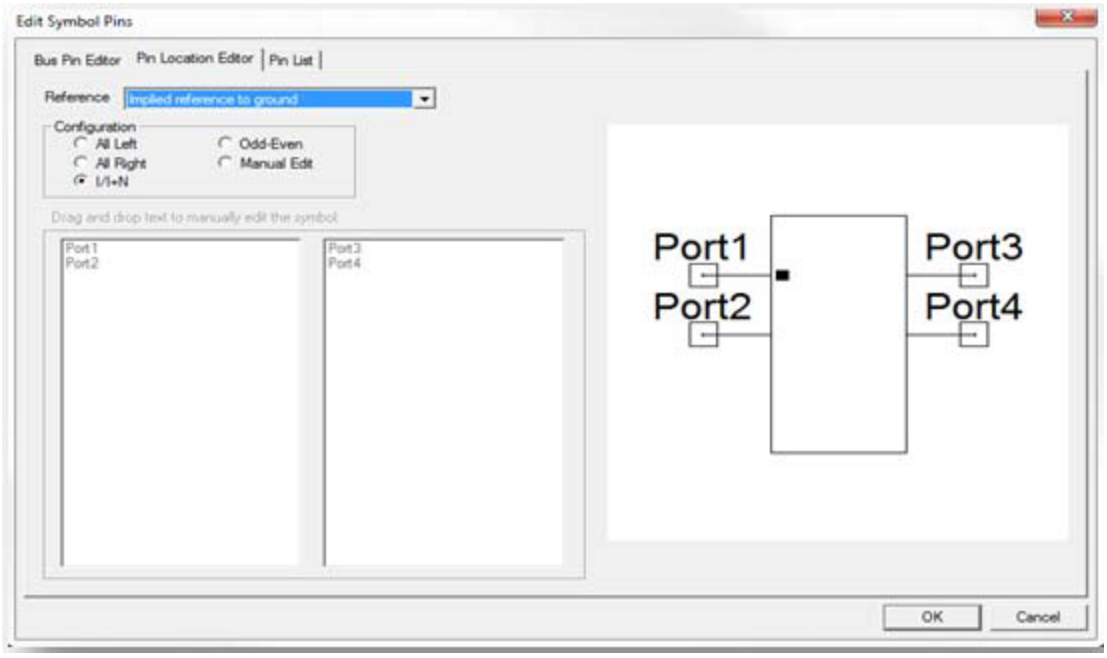
- **To create a new bus pin**
  - Select the pins needed for the bus pin and click **Combine in to Bus Pin**. Both regular pins and bus pins can be combined to create the new bus pin.
  - Pins can be selected with Ctrl + Click or Shift + click.
  - Pins can be dragged and dropped onto an existing bus pin.
- **To remove pins from a bus pin**
  - Select the whole bus pin or individual pins inside the bus pin and click **Uncombine from Bus Pin**.
  - Pins can be dragged out of the bus pin to the root level to remove on the bus pin.
- **Re-ordering Pins**
  - The pins inside a bus pin can be reordered by dragging them to the appropriate location. The order is important since the first pin with index 0 connects to bus[0] when a bus is connected, and so on.
  - When either the root node (red symbol) is selected OR a bus pin is selected, Sort features become available. Click the buttons to order the pins and bus-pins immediately under the selected node by name in either alphanumeric or reverse-alphanumeric order.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Searching the Tree**
  - The tree can be filtered by the pin name by typing in the filter box.
  - The filter text supports wildcard search (e.g., "P*1" should search for everything that begins with a "P" and ends with a "1").
  - Search works for both the root level and leaf level of pins (e.g., if a search matches the pin name under a bus pin, the bus pin is displayed in the search result).

Pin Location Editor

The **Pin Location Editor** allows you to change the locations of the symbol pins to the left or right side according to the settings chosen in the window.



Selecting a new option on the **Reference** drop-down menu changes the symbol displayed to show or hide additional reference pins. The **Reference** list is not shown in the window if the symbol does not support reference pins. The options are:

- **Implied reference to ground**: the global ground is the reference port for each N port connection. Its voltage is always 0V.
- **Common reference port**: an additional reference pin is added that is a common reference for the definition of port voltages. This reference can have any voltage and corresponds to the net for that extra pin.
- **Add individual hidden reference pin per port**: an additional reference pin is added for each port. These reference pins are hidden, set to 0V by default, and can each have their

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

own reference voltage. You can connect each reference pin to its own net. Select this option if you do not want to see the reference pins.

- **Add individual reference pin per port**: an additional reference pin is added for each port. Every reference pin can have its own reference voltage and be connected to its own net.

> **Note:**
>
> See Reference Nodes on S-Parameter Elements in the S-parameter Technical Notes for how the Circuit solver handles reference node configurations.

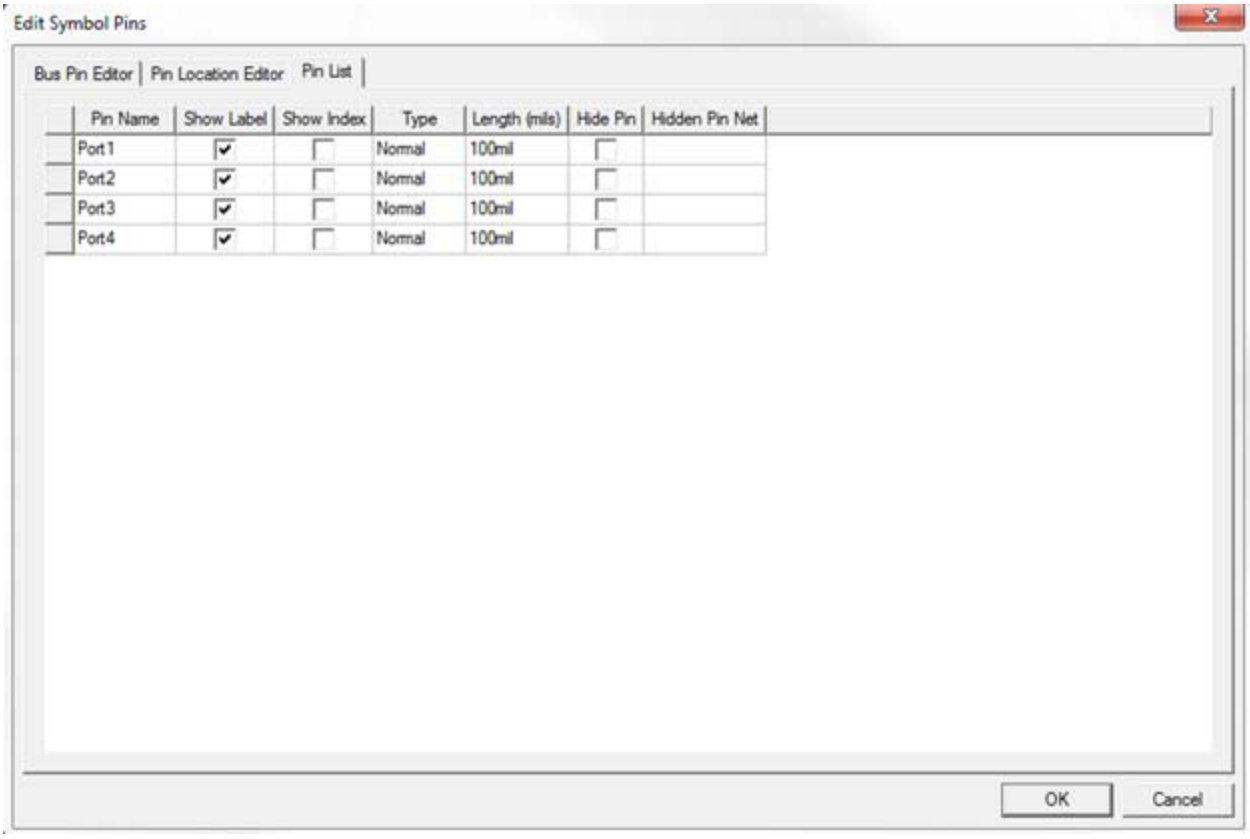
Clicke of the five **PinDirection** options to change the symbol shown to a fixed configuration.

- **All Left** puts all pins on the left side
- **All Right** puts all pins on the right side
- **I/I+N** puts the first half of the pins in order on the left side and the remaining pins on the right side
- **Odd-Even** puts every other pin on the left side and the remaining pins on the right side
- **Manual Edit** enables the two list boxes in the bottom left quadrant. When those boxes are enabled, the user can drag pins from one box to the other to switch sides and drag pins within a box to switch the pin order.

Pin List

The **Pin List** window displays and sets values for the properties of symbol pins.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The available pin properties include:

- **Name** – This editable cell displays and sets the pin name. To specify or change the name of a pin, click in the **Name** cell for that pin, type the name, then press **Enter** or click in another cell.
- **ShowLabel** – This check box controls visibility of the pin name in the schematic editor. For a new pin, it is unchecked by default.
- **ShowIndex** – This check box controls visibility of the pin number in the schematic editor. For a new pin, it is unchecked by default.
- **Type** – This drop-down menu sets the pin type, the options for which include **Normal** (the default), **ANSI In, ANSI Out,** and **Zero Length.** To change a pin's type, click in its **Type** cell, then click the appropriate option.
- **Length** – This editable cell sets the length of the pin stem (the graphical line associated with the pin port symbol). By default, new pins have a pin stem length of 10. To change the length of a pin, click in its **Length** cell, type a new value, then press **Enter** or click in another cell.
- **HidePin** – This check box controls whether or not a pin is visible in the schematic editor. For new pins, it is unchecked by default.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **HiddenPinNet** – This cell, editable when **Hide Pin** is checked, specifies the net (the circuit node) to which a pin is connected if it is hidden. To specify a net, click in the **Hidden Pin Net** cell, type a net name, then press **Enter** or click in another cell.

Changing Pin Properties

To change a pin property, either:

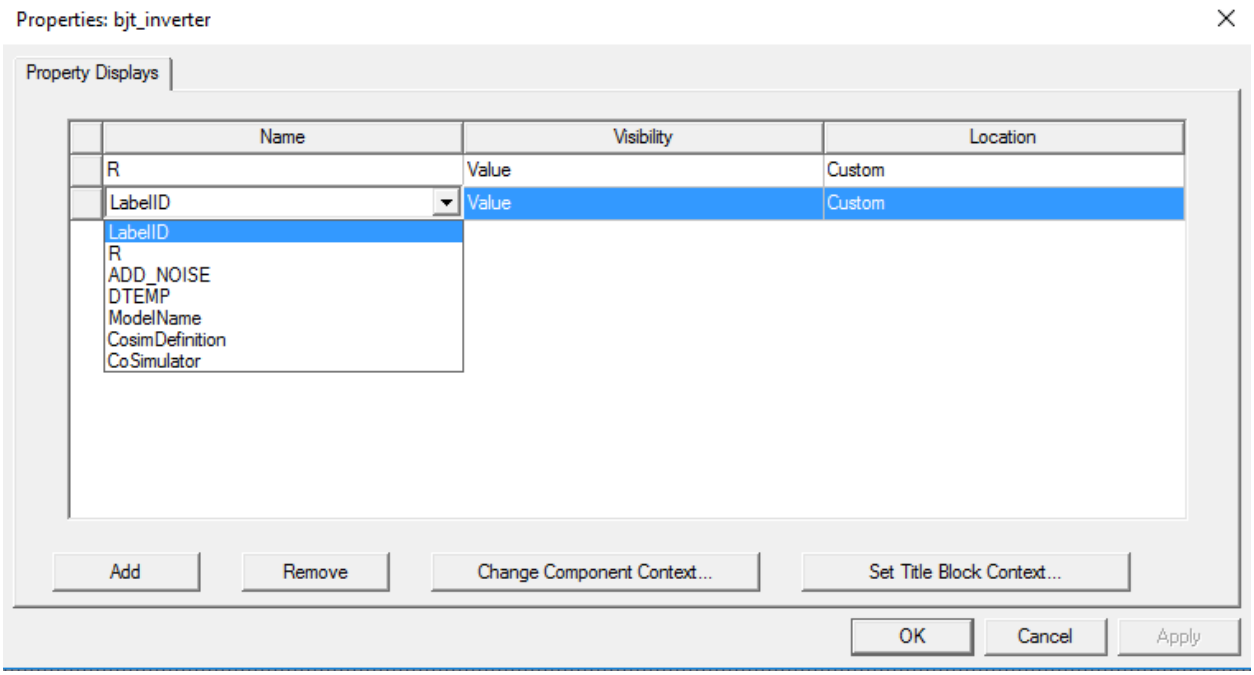
- Click the cell that displays the pin property value that you want to change and modify the property value appropriate to its type as described above.
- To commit the new value and continue editing pin properties, click another property value.

Committing Changes

Clicking **OK** in the window commits changes to the component symbol made in all three tabs.

Symbol Property Display Setup

The **Symbol** > **Property Display Setup** command opens a **Properties** window in which you can add or modify how the names, values, and locations of properties associated with the component or object (such as a title block) are represented by the current symbol.



The Symbol Property Displays Window

This window lists symbol display properties and controls their presence, visibility, and position. Every entry in this window corresponds to a property on a component associated with the symbol in question. Whoever creates or edits a component determines which property to add.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Some properties such as [LabelID] can appear in multiple components. The component is set automatically if you are editing an existing symbol. To change the reference component, click **Change Component Context**.

> **Note:** [LabelID] may display as **[InstanceName]** or *** in the Symbol editor.

Columns in the grid are:

- **Name** – Click in this cell to type a property display name, or to select an existing property display from a list. The component selected via **Change Component Context** determines which property displays are listed for selection.
- **Visibility** – Click in this cell to set the visibility (**None**, **Name**, **Value**, or **Both**) of a property display.
- **Location** – Click in this cell to set the location (**Left**, **Top**, **Right**, **Bottom**, **Center**) of the property display relative to the symbol center.

> **Note:** You can further position a property display by rotating it (**Ctrl**+**R**), or by dragging it to another location. When you do this for a given property display, its Location value is set to **Custom**.

The actions available in this window are:

- Click **Add** to add a new property display entry to the list.
- Click **Remove** to remove the selected property display.
- Click **Change Component Context** to open the **Select Definition** window, to select the component definition on which property displays are based. The properties of the selected component are then available in the **Name** cell properties list.

> **Note:** Because a symbol can be used by multiple components with differing properties, any property name can be entered by the user. If a component using the symbol does not have a property with that name, the symbol property isn't used in that component.

- Click **Set Title Block Context** to make the list of default page properties, and others as specified, available for display. Default properties include: ProjectPath, Project, Design, Title, Author and Date. This context is selected by default if the symbol has three or more propdisplays for default page properties.

Symbol Grid Setup window

The **Symbol** > **Grid Setup** window sets visibility, colors, resolution, and snapping for the Symbol Editor’s alignment grid. It contains the following controls.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Major** — Specifies the spacing of the major grid lines (default in mils). To change the color of the major grid lines, click the **Major** color button, specify a color in the **Color** window, then click **OK**.
- **Minor** — Specifies the spacing of the minor grid lines (default in mils). To change the color of the minor grid lines, click the **Minor** color button, specify a color in the **Color** window, then click **OK**.
- **Show Grid** — Toggles grid line visibility.
- **Snap to Grid** — Controls whether graphics and text placed on the grid automatically snap to the nearest grid intersection.
- **Background Color** — To set the background color used for symbol editing, click the color box to open its window, specify a color, then click **OK**.
- **Save as Default** — Save the current **Grid Setup** values as defaults for future Electronics Desktop sessions.
- **Defaults** — Restore all **Grid Setup** settings to their defaults.
- **OK** — Commit changes and close the window.
- **Cancel** — Close the window without committing changes.

Note:

- To ensure electrical connectivity among schematic elements, placed symbol pins snap to a 100-mil grid (2.54-millimeter) regardless of the graphical Major and Minor grid line settings. This snapping cannot be deactivated, and the spacing of the connectivity grid cannot be adjusted.
- The units used for grid line spacings are controlled by the **Tools** > **Options** > **General Options** window.
- Use of the **Minor** grid setting varies depending upon the active editor. In the Layout Editor, the minor grid-line setting specifies the number of units between each minor grid division. However in the Schematic and Symbol Editors, the **Minor** grid-line setting specifies the number of minor grid lines that appear between major grid lines.

Using the Footprint Editor

The Desktop **Footprint Editor** supports the viewing, modification, and saving of footprint information for components and layout primitives. After modifying a footprint, the information can be saved, used to define a design, or exported.

A footprint is a set of information that defines the planar space consumption, physical orientation, metal usage, and electrical connectivity of a component. Footprint definitions are stored in library files with an extension of **.aflb**. You can access material definitions using the **Footprint Editor**.

Creating or editing a footprint involves using the footprint editor to:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Draw graphical primitives, such as rectangles, circles, and arcs, using options on the Draw menu
- Add pins using the Pin option on the Draw menu
- Add text labels using the Text option on the Draw menu
- Update the current project with the new or revised footprint definition using the **Update Project** option on the **Footprint** menu

You can export a footprint to a footprint library (**.aflb**) file for use in other projects. For information on how to do this, see the Edit Libraries window topic.

Opening the Footprint Editor

You can create a new footprint definition or edit an existing footprint.

Creating a New Footprint

To create a new footprint, do one of the following:

- Open the **Tools > Edit Libraries > Footprints** window, then click **Add Footprint**.

  — The **Definition Name** window opens and allows you to specify the name for the new footprint.

  — Type a name for the footprint into the **Enter the name for this new Footprint** group box, then click **OK** or press **Enter**. The **Choose Technology** window opens and allows you to specify the original layers of the definition

  — Double-click the entry for the new footprint, or click **Edit Footprint**.

- Select **Add Definition** on the right-click menu in the **Footprints** folder of the **Project Manager** window.

  — The **Definition Name** window opens and allows you to specify the name for the new footprint.

  — Type a name for the footprint into the **Enter the name for this new Footprint** group box, then click **OK** or press **Enter**. The **Choose Technology** window opens and allows you to specify the original layers of the definition

- Open the **Tools > Edit Libraries > Footprints** window, then locate and select an existing footprint on which to base your new footprint.

  — Click **Clone Footprint(s)** to open the **Get Name** window.

  — Type a name for the new footprint into the **Enter the name for this new Footprint** group box, then click **OK** or press **Enter**.

  — Double-click the entry for the new footprint, or click **Edit Footprint** to open the **Footprint Editor**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Editing an Existing Footprint

To edit an existing footprint, do one of the following:

- From the **Project Manager** window, expand the **Definitions** /**Footprints** subfolder for the project that contains the footprint you want to edit. Double-click the entry for the footprint you want to edit, or right-click the entry and select **Edit Footprint**.
- Open the **Tools > Edit Libraries > Footprints** window, then locate and select the Project version of the footprint that you want to edit. Click **Edit Footprint**, or double-click the selected entry.
- Select the footprint you wish to edit and click **Edit Footprint** in the shortcut menu.

The footprint editor runs and opens the selected footprint for editing.

Load a Footprint into the Current Project

To load a footprint into the current project:

1. With a new local project open, select **Tools > Edit Libraries > Footprints**.
   This opens the Edit Libraries window.
2. Select a component in the Name column, for instance, BENDA, and click **Edit Footprint**.
   This loads the component into the local project.

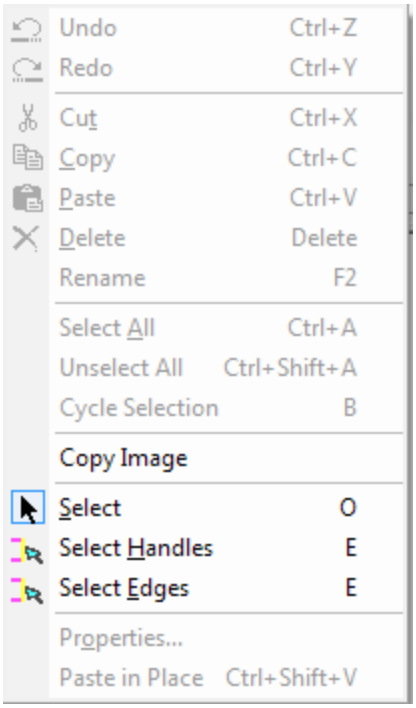
Footprint Editor Operations

The footprint editor opens when a footprint is created or modified. Like the schematic and symbol editors, the footprint editor is a graphical tool that keeps the placement of footprint elements uniform by snapping them to points on a grid.

With the footprint editor, add and manipulate graphical primitives, text labels, pins, vias, and padstacks, and/or adjust the resolution, color, and visibility of the editor grid. Once changes are made, update the current project with any changes to the footprint. Options related to these operations are available on the **Footprint** menu and the Footprint Draw menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Footprint Edit Menu



The fields on this menu become active as appropriate to the type of footprint being edited.

Undo*<last edit operation>* — Undo the last edit operation.

Redo*<last edit operation>* — Redo the last edit operation.

Cut – Delete the selected object, and retain a copy for pasting into a layout in the same application.

Copy – Create a local copy for pasting into a layout in the same application.

Paste – Put the object on the last cut or copy into the layout.

Delete – Delete the selected object without retaining a copy.

Rename – Click to rename the selected object in the layout.

Select All – Select all objects in the layout.

Unselect All – Unselect all objects in the layout.

Cycle Selection – Cycle selection of two or more objects that overlap. Each time you click **Cycle Selection**, a different one of the overlapping objects is selected.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copy Image — Create a global copy of the selected objects on the clipboard for pasting into a different application.

Select – Sets cursor mode to selection of objects.

Select Handles – Sets cursor mode to selection of object handles.

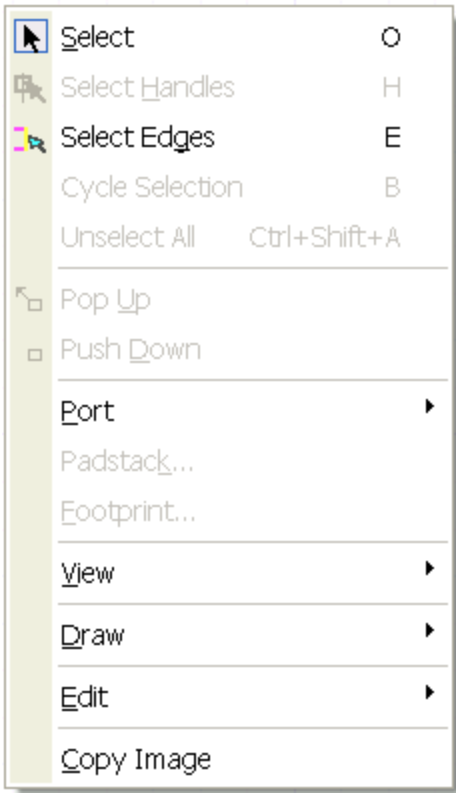
Select Edges – Sets cursor mode to selection of object edges.

Properties – Opens the **Properties** window to view and edit the properties of the selected object.

Paste in Place – Paste object to same location, without any X,Y offset/displacement.

Footprint Shortcut Menu

When a footprint design is active, right-click to open the shortcut menu for the Footprint Editor, similar to the following:

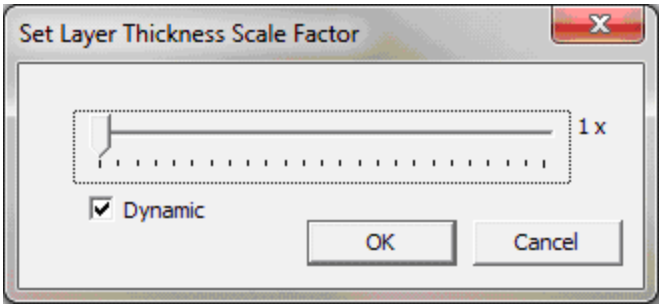


The fields on this menu become active as appropriate to the type of footprint being edited.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Footprint View Menu

- Control the display by selecting the check box of the following: **Status Bar**, **Message Manager, Project Manager window, Properties Window, Progress Window, Component Manager, Layout Window.**
- **Variables**: View the project and design variables with their values in a grid view.
- **Rotate**: **click+drag** to rotate the view; the same double-click options hold true for the alt+drag editor display options.
- Change the zoom display by selecting one of the following: **Zoom In, Zoom Out, Zoom Area, Zoom Previous.**
- Fit or Pan by selecting: **Fit Drawing, Pan.**
- Select **Stretch Z** to open the **Set Layer Thickness Scale Factor** window.



- Dragging the slider applies a Z scaling factor to the current view. If **Dynamic** is selected, the scaling happens in real time otherwise it is applied when **OK** is chosen.

Footprint Draw Menu

The footprint editor **Draw** menu presents options for drawing graphical primitives and text, and manipulating selected objects. The **Draw** menu options include:

Primitive > Arc

This option, also available on the toolbar, initiates creation of an arc. To draw an arc, click the footprint editor grid at the two points that determines the arc's ends, and then drag the arc to the appropriate radius.

Once you have created an arc, adjust its radius or move its endpoints by clicking the arc to select it, and dragging the appropriate handle.

Primitive > Circle

This option, also available on the **Footprint Draw** toolbar, initiates creation of a circle. To draw a circle, click the footprint editor grid to select a center point, then drag the circle to the appropriate diameter.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click the circle to select it, then drag one of its handles.
- Double-click the circle to open its **Properties** window, type a new value for the **Radius** parameter, and click **OK**.

Primitive > Line

This option, also available on the **Footprint Draw** toolbar, initiates creation of a polyline with one or more segments. To draw a line:

To complete the line, do one of the following after defining its final segment:

- Press the **Spacebar**.
- Right-click and click **Finish**.

After you have completed a line, change the endpoints of its segments as follows:

- Select the line and drag the appropriate handle(s).
- Double-click the line to open its **Properties** window, type new values for the parameters you want to change, and click **OK**.

Primitive > Rectangle

This option, also available on the **Footprint Draw** toolbar, initiates creation of a rectangle. To create a rectangle, click the editor grid to specify the position of one corner, then drag the rectangle to the appropriate size.

Once you have drawn a rectangle, Edit its height and width, the position of its center, and its angle (its rotation, in degrees, relative the handles of its bounding box) as follows:

- Double-click the rectangle, then edit its properties in the **Properties** window.

Primitive > Polygon

This option, also available on the **Footprint Draw** toolbar, initiates creation of a polygon. To create a polygon:

1. Click the footprint editor grid to specify the position of one vertex, then click wherever you want to place additional vertices.
2. To complete a polygon, specify the position of its final vertex, then do either of the following:

Once you have drawn a rectangle, Edit its vertex positions and other properties, including its fill style, as follows:

- Click the rectangle, then edit its properties in the **Properties** window.
- Double-click the rectangle, then edit its properties in the **Properties** window.

Primitive > Text

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This option, also available on the **Footprint Draw** toolbar, adds to the footprint an editable "Default text" label in 12-point Arial. To edit the default text string immediately after placement, type the text you want, then press **Enter** or click elsewhere in the editor grid.

To change just the text of an existing label:

1. Click the label.
2. Click the label again to open its text for editing.
3. Press **Enter** or click elsewhere in the editor grid.

To change other properties of a label, including its font and size:

1. Click the label and view its properties in the **Properties** window, or double-click the label and view its properties in the **Properties** window.
2. Click in the **Value** cell for the property you want to modify.
3. Modify the value.
4. Click **OK**, or click in another **Value** cell to commit the change and keep editing values.

Void > Circle

Create a void, or remove material from an object, in the shape of a circle, rectangle, polygon, or line using the Void > Circle command. One object can have multiple voids, but each void can only be associated with one "parent" object.

Void > Line

Create a void, or remove material from an object, in the shape of a circle, rectangle, polygon, or line using the Void > Line command. One object can have multiple voids, but each void can only be associated with one "parent" object.

Void > Rectangle

Create a void, or remove material from an object, in the shape of a circle, rectangle, polygon, or line using the Void > Rectangle command. One object can have multiple voids, but each void can only be associated with one "parent" object.

Void > Polygon

Create a void, or remove material from an object, in the shape of a circle, rectangle, polygon, or line using the Void > Polygon command. One object can have multiple voids, but each void can only be associated with one "parent" object.

Port > Create

Click the Port > Create command to create a port.

Port > Remove

Click the Port > Remove command to remove a port.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pin

This option, also available on the **Footprint Draw** toolbar, initiates placement of a footprint pin with a default stem length of 10. You can rotate a pin once you've started to place it by iteratively pressing R until the pin is oriented to your liking. Click in the editor grid to finish placing the pin.

Via

This option, also available on the **Footprint Draw** toolbar, initiates placement of a through hole (via) of the default size (1mm).

Handle

This option, also available on the **Footprint Draw** toolbar, allows you to create an explicit footprint handle that can later be used to resize or reshape the footprint based on the handle's reshaping rules.

3D Structure

Opens a menu that allows you to create and perform operations on a 3D structure in the Drawing Region.

Coordinate System

Opens a menu that allows you to create and perform operations on a coordinate system in the Drawing Region manage. For more information see Coordinate Systems in the **Layout Editor** topic.

Rotate

This option, also available on the **Footprint Draw** menu or by pressing **Ctrl**+**R** on the keyboard, rotates a selected object or group of objects 90° to the left.

Flip Vertical

This option, also available on the **Footprint Draw** menu, flips a selected object about the X axis.

Flip Horizontal

This option, also available on the **Footprint Draw** menu, flips a selected object about the Y axis.

Reverse Line

Reverse selected objects in the Drawing Region.

Align...

Position Relative

When a relative coordinate system (CS) and an object independent of that CS are selected, the "Draw > Position Relative" command positions the CS relative to that object.

Clear Position Relative

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When a relative coordinate system (CS) and an object are selected, the “Draw > Clear Position Relative” command positions the CS unrelative to that object.

Duplicate

Duplicate a selected object in the Drawing Region.

Expand

Expand selected objects in the Drawing Region.

Split Polygon Region

Split selected polygon regions in the Drawing Region.

Geometry Healing

Geometry heal selected objects in the Drawing Region.

Stitch Lines

Stitch together connected/crossing lines into polygons and lines.

Defining Footprint Handles

Handles appear around the perimeter of a selected element as small squares in the selection color. If the cursor is moved over one of these handles, it changes to indicate its proximity to the handle. When clicking on a handle, the user can then resize or reshape the element based on the handle's reshaping rules.

Component footprints can also have handles which can be specified in the footprint editor. These handles can likewise be used to reshape a component footprint, and at the same time altering one or more of its electrical parameters. In other words, the user can use handles to directly manipulate an element's geometry corresponding to electrical parameters.

Defining a Handle

Open the footprint editor for the footprint appropriate. From the **Draw** menu, choose **Handle**. Next, click to place the handle anywhere in the drawing, exact placement does not matter.

Editing Handle Location

Next, open the property window (**Edit > Properties**). There is one property listed: location. For this property, edit the X & Y locations to place the handle in the correct position based on the parameters.

Specifying Parameters to Change

Next, click **Add** in the **Property** window. In the responding window, specify the name of the electrical parameter, and the expression, based on the X & Y location of the handle that sets this parameter. For example:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Edit the footprint for the MSTRL component. Place a handle that is on the top edge of the line. The component has the parameters P for length and W for width. The handle has its location (X,Y) set to: (P/2, W/2). One property is added called "W" and its expression is "abs(Y*2)". The "abs" (absolute value) of "Y*2" ensures that as Y approaches zero, the line width stays positive. The two entries in the window should look like the following:

Location: P/2, W/2

W: abs(Y*2)

How it works

When the user drags the handle, for each new location of the cursor, each of the handles properties are recalculated for that new position. For the TRL example, the y position of the top-side handle is used to set the line's "w", which of course is twice that of y. After the new parameter values are calculated, the handle's own position is recalculated based on the updated parameters and the expressions for its location. Thus, as the cursor goes up, the line gets wider, and the handle tracks with the new width.

Using the Padstack Editor

A padstack is a set of information that defines the physical structure and electrical connectivity of a pin or via (the region in which a component pin connects to a layout or a connection between layers). A padstack includes information about the layers involved in making the connection, on the size and shape of the layout area to the type and dimensions of the accompanying hole. Padstack definitions are stored in library files with an extension of **.pslb**.

The Electronics Desktop Padstack Editor supports the viewing, modification, and saving of padstack definition information for physical structure and connectivity. After modifying a padstack definition, the information can then be used to define a via or pin. Padstack definitions can be exported to library files.

To open the **Padstack Editor**, load a project and select **Tools** > **Edit Libraries** > **Padstacks**. When the Edit Libraries window opens, use its controls to open the **Padstack Editor**.

Creating a New Padstack

To create a new padstack, do one of the following:

- Select **Tools** > **Edit Libraries** > **Padstacks**, then click **Add Padstack**.

  The **Definition Name** window opens and allows you to specify the name for the new padstack.

  The **Choose Layout Technology** window opens, and you may browse for a technology file to define the original layers of the definition. If no technology file is chosen, the definition is created with initial layers named **Start, Stop, and Default** .

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select **Tools** > **Edit Libraries** > **Padstacks**, then locate and select an existing padstack definition on which to base a new definition.

  Search for and select a padstack name and then click **Clone Padstack(s)**.

- From the **Project Manager** window, right-click the **Padstack** folder and select **Edit Library** to open the Padstack tab of the **Edit Libraries** window. Click **Add Padstack** or **Clone Padstack(s)** to open the **Edit Padstack Definition** window.
- In the Project Tree, right-click **Padstacks** and select **Add Definition**.

After a padstack is created, edit the padstack.

Editing a Padstack

Edit and define a padstack in the **Edit Padstack Definition** window. Open this window in one of the following ways:

- Select **Tools** > **Edit Libraries** > **Padstacks**. Find and select the padstack you want to edit, then double-click its name or click **Edit Padstack**.
- In the **Definitions** directory of the **Project Manager** window, right-click the **Padstack** folder and select **Edit Library**. Find and select the padstack you want to edit, then double-click its name or click **Edit Padstack**.
- Select a via or pin that uses the padstack. In the **Properties** window, click **Padstack Definition** to open the **Padstack Usage and Definition** window. When opened on the **Properties** window, information related to the padstack definition is highlighted in blue.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

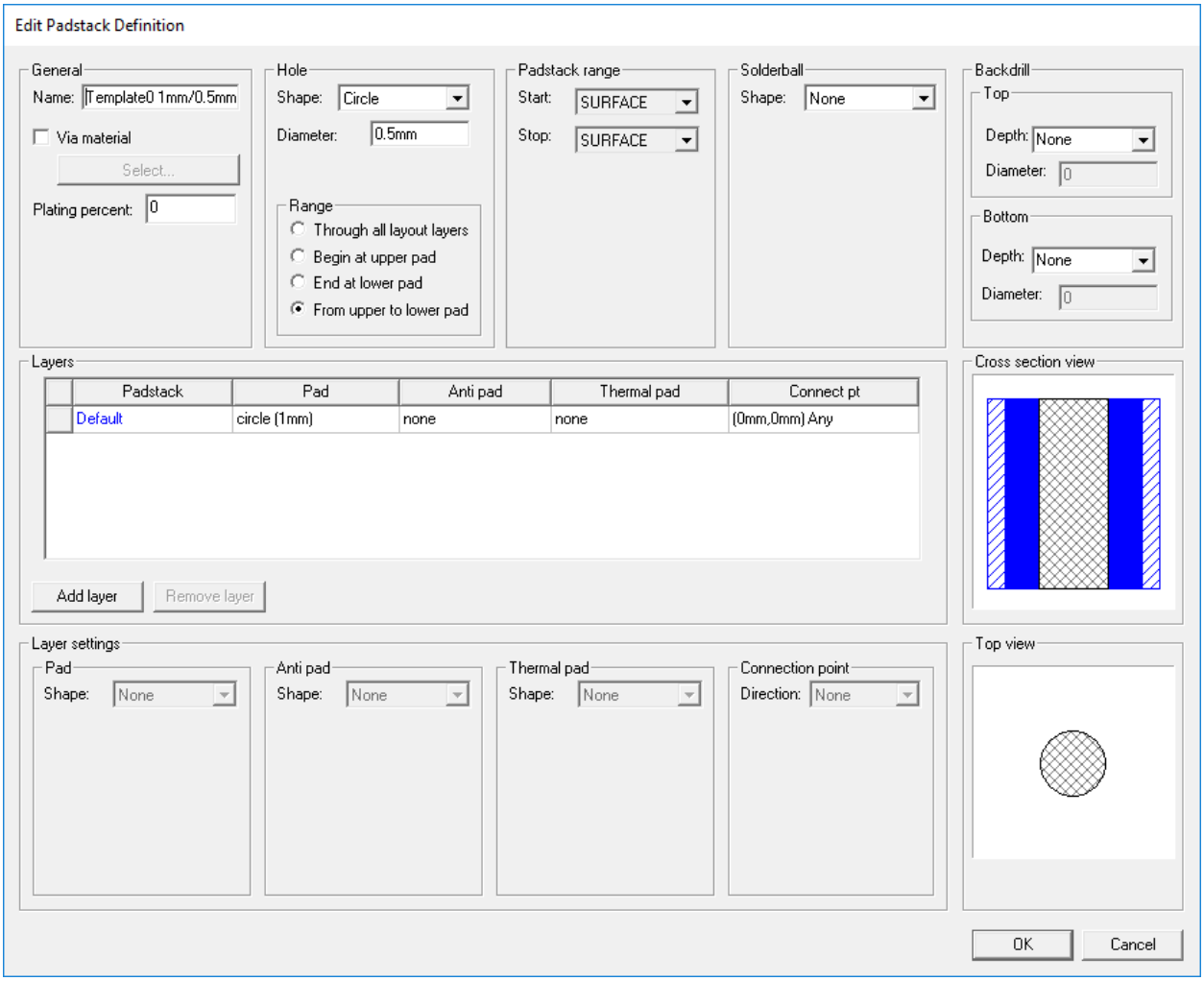


Figure 13-1 The Edit Padstack Definition window

> **Important:** Some options may be unavailable or may not appear, depending upon the context and which padstack is being edited.

Options include:

General

- **Name** – the name of the padstack.
- **Via Material** – the material for plating the via or pin hole. Click **Select** to choose from available materials.
- **Plating Percent** – the percent of the hole's radius that is filled by the plating material. At 100%, plating fills the hole. The **Cross section view** and **Top view** areas show the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

vertical cross-section and top of the layer stackup. This setting is used by SIwave Solvers and Icepak **Electronics Desktop**.

> **Note:** Zero plating (i.e., 0%) is not recognized and is treated as fully plated.

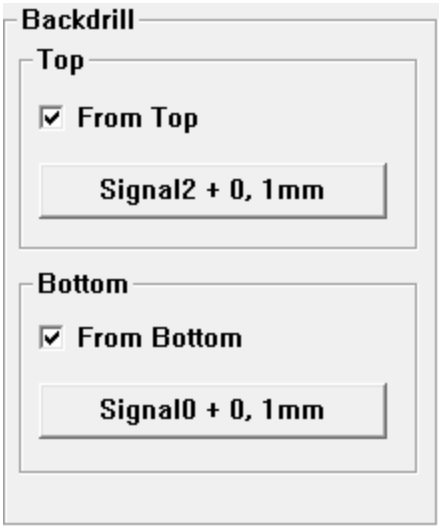
Hole

- **Shape** – choices are None, Circle, Square, Rectangle and Polygon. The Polygon choice is currently only supported for padstacks; polygon holes may not be defined through the window. Polygons can be modified by selecting a layer and clicking **Edit Polygon** to open the **Polygon Editor** . Selecting a shape enables additional options:
  - **Size / Diameter / XY Size** – specifies the size of the hole.
  - **Range** – can specify the hole range to be:
    - Through all the layout stackup layers
    - Beginning at the upper pad of the padstack and continuing to the lowest elevation of the layout stackup layers
    - Beginning at the highest elevation of the layout stackup layers and ending at the lowest pad of the padstack
    - Beginning at the upper pad and ending at the lowest pad of the padstack

Padstack Range

This area contains controls to specify the **Start** and **Stop** layers for the padstack.

Backdrill



Backdrilling creates empty holes on one or both sides of a via. From the **Backdrill** group box, define backdrills from a selected via's top layer, bottom layer, or both. Refer to Backdrilling.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Backdrills are rendered in the **Cross section view** and **Top view** group boxes as empty space. In the following script example, the backdrill is recorded as a block:

```
Array("NAME:Backdrill Top Diameter", "MustBeInt:=", false, "Value:=",
"0.4mm").
```

- **From Top** – define a backdrill that starts on the selected via's top layer. Check the adjacent box to activate or remove the check to deactivate. Click the associated button to proceed to the **Backdrill Definition** window.
- **From Bottom** – define a backdrill that starts on the selected via's bottom layer. Check the adjacent box to activate or remove the check to deactivate. Click the associated button to proceed to the **Backdrill Definition** window.

  Refer to Define Backdrill Parameters.

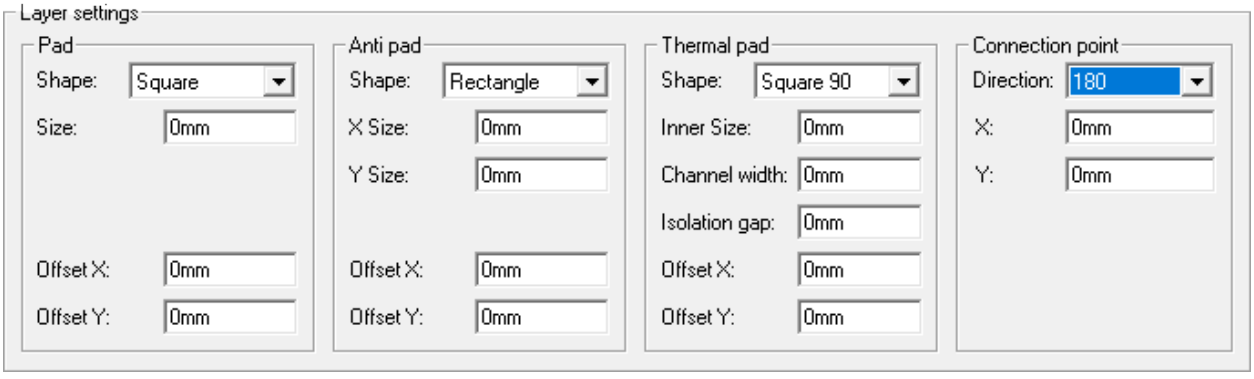
Solderball

- **Shape** – specifies the shape of the solderball; choices are none, cylinder, and spheroid.
- **Diameter** – specifies the diameter of the solderball.
- **Mid diameter** – for spheroid solderballs, specifies the mid diameter.
- **Solder** – click to open the **Material Editor** for the solderball in order to specify the material used.
- **Connection** – select to specify the location of the solderball either above or below the padstack hole.

Layers

Layer names are used when a padstack definition is used in a via or pin. A default mapping of padstack definition layers to layout layers occurs based on the layer names.

- None of the **Layers** group box grid control cells are directly editable.
- When one or more rows in the **Layers** group box is selected, the **Layer settings** group box is enabled and can be used to change the pad settings for selected rows.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Add Layer** inserts a new layer above the selected row.
- **Remove Layer** removes the selected row(s). You may also rearrange rows to be any order.

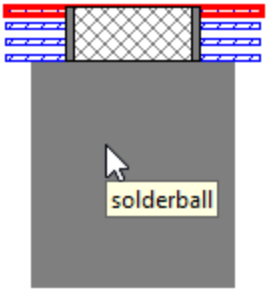
Layer Settings

The **Layer settings** area displays pad settings for padstack definition layers. The Polygon choice is currently only supported for imported padstacks; polygon pads, anti pads, and thermal pads may not be defined through the window. Dimension controls appear to support the shape choice for each pad. Controls to specify the pad offset are also available.

- **Pad** and **Antipad** shape choices are None, Circle, Square, Rectangle, Oval, Bullet, N-Sided Polygon, and Polygon. Polygons can be modified by selecting a layer and clicking **Edit Polygon** to open the **Polygon Editor** .
- **Thermalpad** shape choices are None, Polygon, Round45, Round90, Square45, and Square90. Polygons can be modified by selecting a layer and clicking **Edit Polygon** to open the **Polygon Editor** .
- **Connection point** allows you to choose a direction in degrees or specify None. There are also controls to specify the X,Y location of the connection point.

Cross Section View

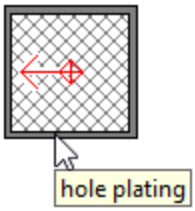
This area displays all definition layers and their pads from a side view. Selected layers are highlighted. Hover the mouse over this area to see tooltips indicating padstack components.



Top View

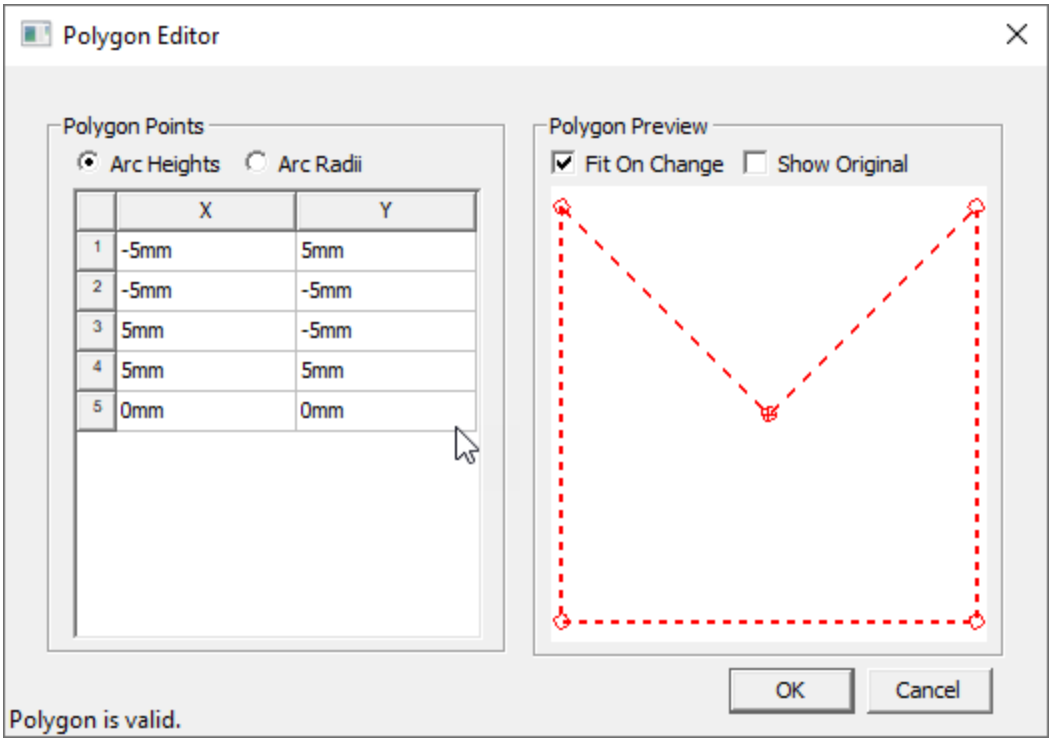
This area displays the pads of selected layers from a top down perspective. Hover the mouse over this area to see tooltips indicating padstack components.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Editing a Polygon Pad

Pads, antipads, thermal pads, and holes can have polygon shapes which can be modified in the **Polygon Editor**. This editor lists the points of the polygon in a grid and displays a diagram of the polygon. The **Polygon Editor** is available on the **Edit Padstack Definition** and the **Padstack Usage and Definition** windows.



Each row in the grid defines a vertex or an arc segment. Vertices have **X** and **Y** coordinates that are located in relation to the center point of the pad, antipad, thermal pad, or hole. The center point is marked by a plus sign (**+**) on the diagram. Enter a distance on the center point for each coordinate. Select a row to find its corresponding point in the diagram.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Arc segments are curved segments defined by three points: a beginning, end, and apex. Each arc segment must have a vertex before and after it to define its beginning and end. Thus, arc segments cannot be adjacent to one another. Each arc segments has an arc height or arc radius which can be changed in the grid.

The grid has a shortcut menu where you perform the following tasks. Different options appear depending on whether you select a row and whether you Right-click a vertex or arc segment.

- **Append Vertex** adds a new row for a vertex to the grid with the coordinates 0 mm, 0mm.
- **Insert Vertex Before** inserts a point before the current selection.
- **Insert Vertex After** inserts a point after the current selection.
- **Remove Vertex/Segment** removes the row you Right-clicked on.
- **Convert to Arc Segment** converts a vertex to an arc segment.
- **Insert Arc Segment Before** inserts an arc segment before the current selection.
- **Insert Arc Segment After** inserts an arc segment after the current selection.

The diagram displays a rendering of the polygon and the object's center point. Hover over a point to see its coordinates. Click a point to select it in the grid. Right-click the diagram to zoom in or out or to resize it to show the entire polygon. The diagram offers two options:

- **Fit On Change** redraws the diagram to fit the polygon whenever you make a change.
- **Show Original** displays an outline of the original polygon to compare it with any changes.

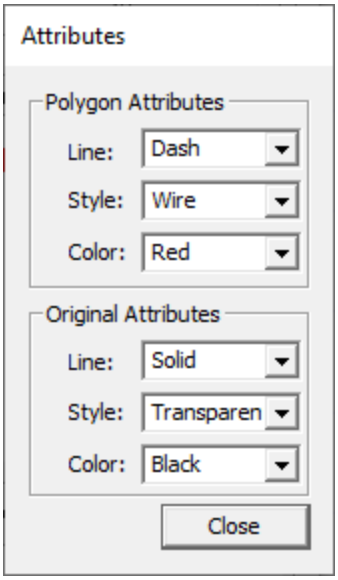
Edit the Polygon

1. Add new vertices as necessary.
    a. Highlight the row of the point just before where you want to add a vertex.
    b. Right-click and choose **Insert Vertex After**. A new row appears in the grid with the coordinates 0 mm, 0mm.
    c. Enter coordinates in the **X** and **Y** columns of the new row.
2. Add new arc segments as necessary.
    a. Highlight the row of the point just before where you want to add an arc segment.
    b. Right-click and choose **Insert Arc Segment After**. A new row appears in the grid.
    c. Above the grid choose whether you want to work with **Arc Heights** or **Arc Radii**.
    d. Enter the arc height or arc radius of the new segment in the new row.
3. Remove unnecessary vertices or segments.
    a. Highlight the row of the unwanted vertex or segment.
    b. Right-click and click **Remove Vertex/Segment**.
4. Check your work in the diagram.
5. Click **OK** to close the **Polygon Editor**. The polygon appears in the **Top view** diagram of the **Edit Padstack Definition** window. The **Cross section view** diagram is unchanged.

Change how the diagram looks

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Right-click the diagram and click **Show/hide attributes** to open an **Attributes** window. Here customize the lines, fill, and color of the polygon. Controls are available for both the edited polygon and the original polygon if **Show Original** is enabled.



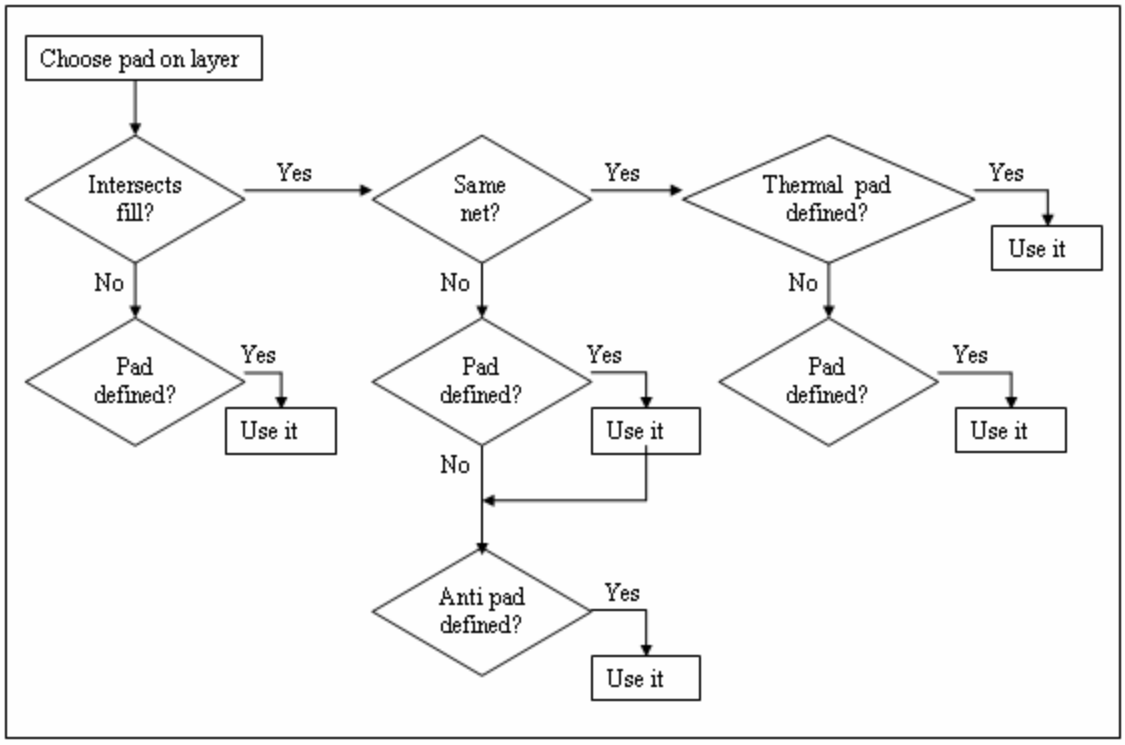
Pad Behavior in Vias and Pins

The pad(s) used in a particular via or pin are determined by the padstack definition and by the intersected primitives in their nets. Multiple pads are sometimes defined for the same layer.

- The intersection point between the central axis of a via or pin and the intersected layers is inspected to determine its metal type (fill, trace, or un-present). By definition, ground layers are fill-metal with primitives defining scratch areas.
- If a fill is intersected, the net of the fill is also taken into consideration.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Based on the intersection of the fill/net/pads that are defined in the padstack definition, the choice of pad that is used is determined using the following algorithm:



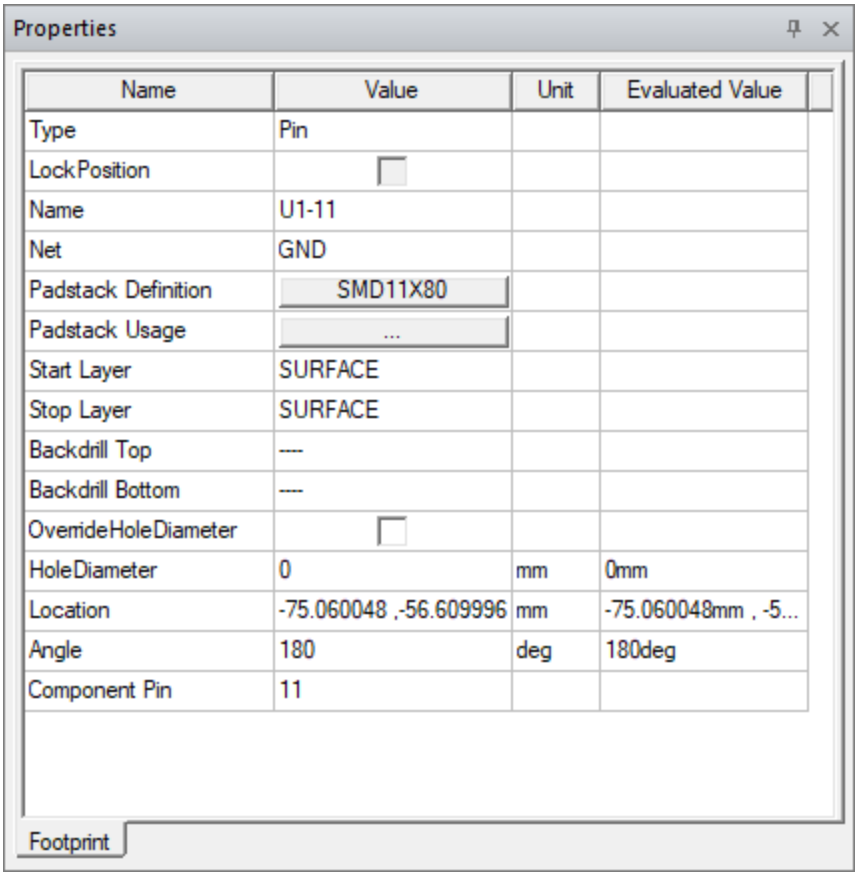
- The pad choice on the algorithm is subject to suppressing of non-functional internal pads as described in Layout Editor Options: Object Panel.

> **Note:**
>
> Anti pads subtract material on the fill area.

Editing Via and Pin Padstacks in Layout

To edit a via or pin padstack, click the via or pin to display its properties in the **Properties** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Name** is the name of the object.
- **Net** lists the net to which the object is assigned.
- **Override Hole Diameter** and **Hole Diameter** are used to override the diameter of the padstack definition for this particular via or pin.
- **Padstack Definition** opens the Edit Libraries window, which allows you to switch the padstack definition that is used by the via or pin.
- **Padstack Usage** opens the **Padstack Usage and Definition** window, to edit the padstack. When opened on the Properties window, information related to the padstack definition is highlighted in blue.
- **Type** determines whether the object definition is a via or pin.

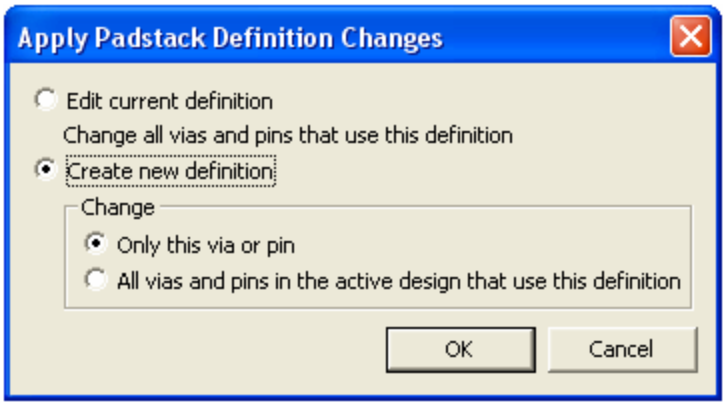
Initial Padstack Definition Layers

Special padstack definition layer names (**Start**, **Stop**, and **Default**) are provided to allow you some control over default mapping of the definition layers to stackup layers when the padstack is placed into a layout as a via or pin. Using these special layer names is optional. When a via or pin is placed, a default mapping occurs in the following order:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Any layout stackup signal layer before the start layer of the padstack or after the stop layer of the padstack does not get mapped to any definition layer.
2. Any layout stackup signal layer whose name matches a definition layer name is mapped to that definition layer.
3. If the start layer of the padstack is not yet mapped, and there is a definition layer named "Start", they are mapped.
4. If the stop layer of the padstack is not yet mapped, and there is a definition layer named "Stop", they are mapped.
5. If there is a definition layer named "Default", any remaining layout stackup signal layers are mapped to it.
6. If there are remaining layout stackup signal layers, they are mapped (in order) to the remaining definition layers.
7. Any layout stackup signal layers that are unmapped, remain that way.

When the **Padstack Usage and Definition** window is closed by clicking **OK**, you are given a choice of how to apply the definition changes that are made; the **Apply Padstack Definition Changes** window opens only if changes are made to the padstack definition (not all changes to the **Padstack Usage and Definition** window change the padstack definition).

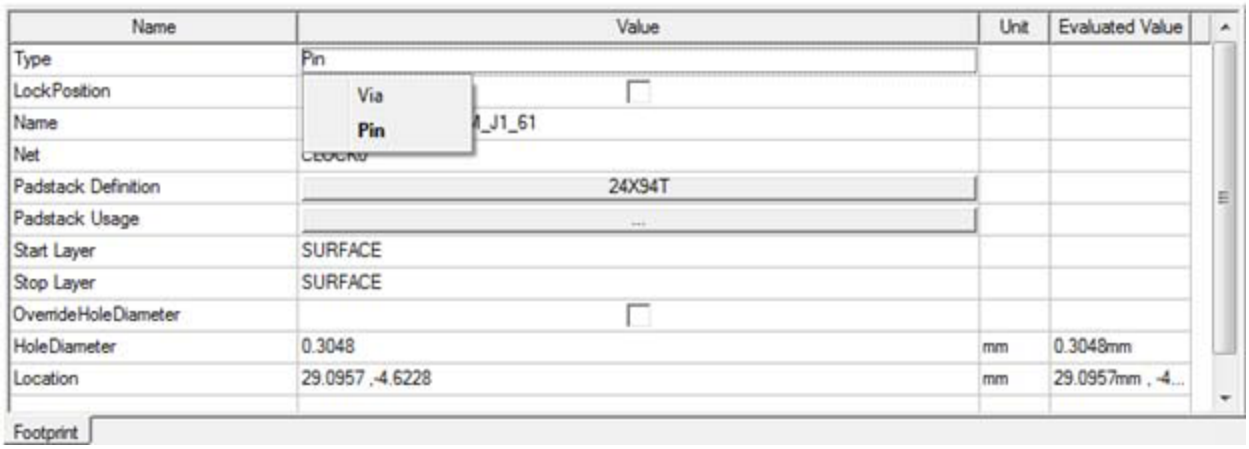


- **Edit current definition** — changes all vias and pins that use this definition
- **Create new definition** – allows you select whether the changes apply to **Only this via or pin** or whether the changes apply to **All vias and pins in the active design that use this definition** of the original padstack.

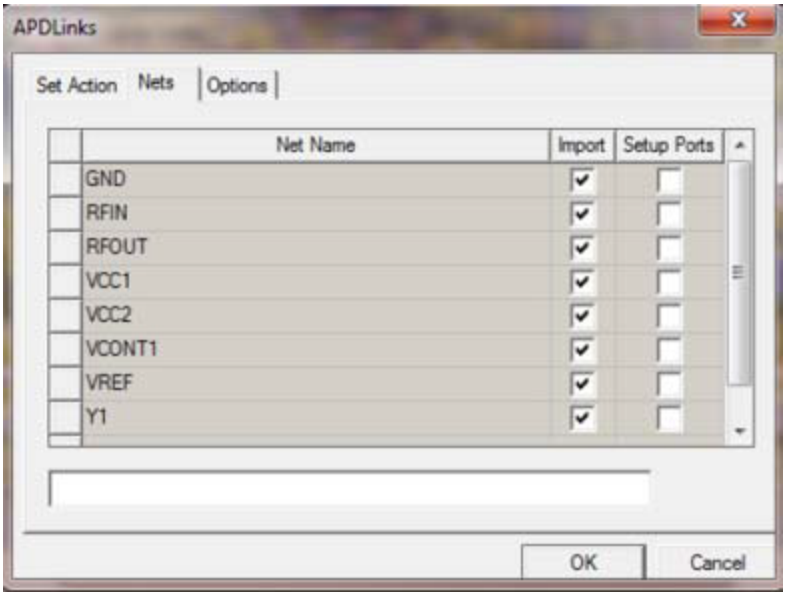
Pin and Net Padstack Operations

Padstack instances are pins or vias; either type may have an associated port. The "pin" designation does not imply that a port must be associated with the padstack. When a port is removed from a pin, the object retains its "pin" type. The type is changed through the "Type" property:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Both APDLinks and ANFV2 imports correctly retain the "pin" and "via" type for padstacks. For APDLinks, only those nets for which "Setup Port" has been selected, has associate ports with the pins.



The following commands act on pins and nets. These are accessed through the shortcut menu on the Layout tab under "Nets" and also on the right-click layout menu. See Pin and Net Padstack Operations for information about the following scripts.

AddPortsToNet

Use:

— Add ports to all the pins on the designated nets.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command:

— In Layout, **Right-click > Port > Create Ports on Net**

— Layout tab under Nets, **Right-click > Create Ports**

RemovePortsFromNet

Use:

— Removes ports from all the pins on the designated nets.

Command:

— In Layout, **Right-click > Port > Remove Ports from Net**

— Layout tab under Nets, **Right-click > Remove Ports**

AddPortsToAllNets

Use:

— Adds ports to all the pins in all the nets.

Command:

— Layout tab under Nets, **Right-click > Create Ports**

RemovePortsFromAllNets

Use:

— Removes ports from all the pins in all the nets.

Command:

— Layout tab under Nets, **Right-click > Remove Ports**

Using the Models Editor

1. To open the window for a model library, click **Tools** > **Edit Libraries** > **Models**. The **Edit Libraries** window opens on the **Models** tab.
2. From the **Models** tab, Click the **Libraries** scrolling window to select the library file for contents display. Use the buttons described to manage selected library objects:

- **Edit Model** - Edits properties or attributes for a selected model in the corresponding model editor. The resulting model may have the same name as the original, but it is saved to the current project, rather than back to the library. In effect, you have “checked out” the part to the project in order to edit it. If you want to write it back to the library, overwriting the original part, use **Export to Library**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(Click the **Show Project Definitions** check box to include current project elements in the **Edit Libraries** window display.)

Some models - such as those that have been encoded - are not editable (button is unavailable). Encrypted models can be edited after the password has been entered to unlock it.

- **Add Model** – Creates a new model object in the selected library. The **Add Model** window allows you to name the model, and choose the model type you want to add. By default, the model is added via the appropriate model editor.
- **Clone Model(s)** – Creates a copy of selected object(s) with a different name in the selected library. Encoded models cannot be cloned (button is unavailable).
- **Remove Model(s)** – Removes selected object(s) on the library. System library models cannot be removed (button is unavailable).
- **Export to Library** – Exports a selected object to a different library. (Use also to export edited objects from the project to the library if necessary.) System library models cannot be exported (button is unavailable).

3. When finished, click **OK** to close the **Edit Libraries** window box.

Using the Material Editor

The Electronics Desktop **Material Editor** supports the viewing, modification, and saving of material information for substrate elements and stackup primitives. After modifying a material, the information can then be saved, used to define a design, or exported.

A material is a set of information that defines the physical properties of a substance — such as that used as a substrate, metallization, or solder mask layer — for its inclusion in a *stackup*, the physical foundation of a manufacturable model. Material definitions are stored in library files with an extension of **.amat**. You can access material definitions using the **Edit Material** window, also referred to as the **Material Editor**.

To start the **Material Editor**, select **Tools** > **Edit Libraries** > **Materials**. When the Edit Libraries window opens, use its controls to open the **Material Editor**.

Using Vendor Components

Electronics Desktop allows you to download vendor components from a server. After being downloaded, libraries are automatically configured so these components can be used in Electronics Desktop.

Some installations of Electronics Desktop include a set of RF Vendor Library components. To use these elements, you unzip the files into the **syslib/Vendor Elements** directory in **Electronics Desktop** installation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using RF Vendor Library Components

The RF Vendor library contains vendor-specific components for Electronics Desktop RF installations. The library is delivered as a compressed file, **Vendor_Libraries.zip**, in the **syslib/Vendor Elements** folder under your Electronics Desktop installation directory.

Here are the steps to follow to enable and place these elements.

Unzip the library directory:

1. Browse to the **syslib** folder in your Electronics Desktop RF installation. The directory path should be something like the following:

   `C:\Program Files\AnsysEM\v`*`xxx`*`\`*`Win64`*`\syslib`

   The entry, *xxx*, is the Ansys EM release number. The operating system field (*Win64* in the example, for the Windows OS) is *Linux64* for the Linux OS.

2. Ensure you have write permission in the **syslib** folder.
3. Open the folder **Vendor Elements**.
4. Locate the file **Vendor_Libraries.zip**.
5. Using any compression tool, extract the contents of the zip file to the **Vendor Elements** folder.

   The folder **Vendor_Libraries** should appear in the **Vendor Elements** directory.

> **Note:**
>
> Since configuration happens automatically at Ansys Electronics Desktop startup, if you unzip the archive while **Electronics Desktop** is open, you may need to save work, close, and restart **Electronics Desktop**.

Place the Component, select Vendor, Series, and Part.

1. In the **Component Manager** group box of Electronics Desktop, you should see a **Vendor Elements** directory with the **Vendor_Libraries** folder. Expand the folder, the **vendors** folder, to display a listing of the component types in the library (Amplifier, Capacitor, etc). Each component type represents a number of components from different vendors.
2. Double-click the icon for the type of part you want to place, then drag the symbol into the design area.
3. Right-click the component, and edit the **Properties**.
4. Select a **Vendor** on the listing of available vendors for the component type.
5. Select a **Series** on the listing available series on the selected **Vendor**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Verify that the **Part** property has the part you wish to use. If the **Part** menu is not populated, or if the part you want is not in the list, change the **Data Type** property (**S-parameter**, **State-Space**, or **subcircuit**) until the part appears on the Part property menu.
7. Select a **Part** on the listing of available parts in the selected **Series**.
8. Optionally, use the **Sweep** window to select a text array variable containing part names to sweep while simulating the component. The text array variable must have been defined previously. See: Adding a Project Variable Array.
9. Click **OK** to close the **Properties** window.
10. Connect the component to the circuit.

Optionally, browse through the folders under **Vendor_Libraries** in the **syslib/Vendor Elements** directory. You can see what Vendors, Series, and Parts are available for each component type. The file extensions on the Part files indicate the **Data Type**:

- S-Parameter data types use the **.sNp** file extension.
- State-Space data types use the **.sss** file extension.
- Subcircuit data types use either the **.ckt** file extension or the **.lib** file extension.

Encrypted Libraries

Electronics Desktop lets users create personal, encrypted files of design information (typically in SPICE format) that they may be added to other designs via *.lib*, *.va*, or *.inc* statements. Encrypted library files are password-protected. If the user references an imported encrypted *.lib* or *.va* file with a user-defined password, the first time a component from the file is used, **Electronics Desktop** prompts the user for the encryption password. The library file and password are then added to the list of password definitions in the **Tools** > **Password Manager** window.

> **Note:** For security reasons, encrypted components are not saved in the **Most Recently Used** list or the **Favorites** list.

Creating and Managing Encrypted Libraries

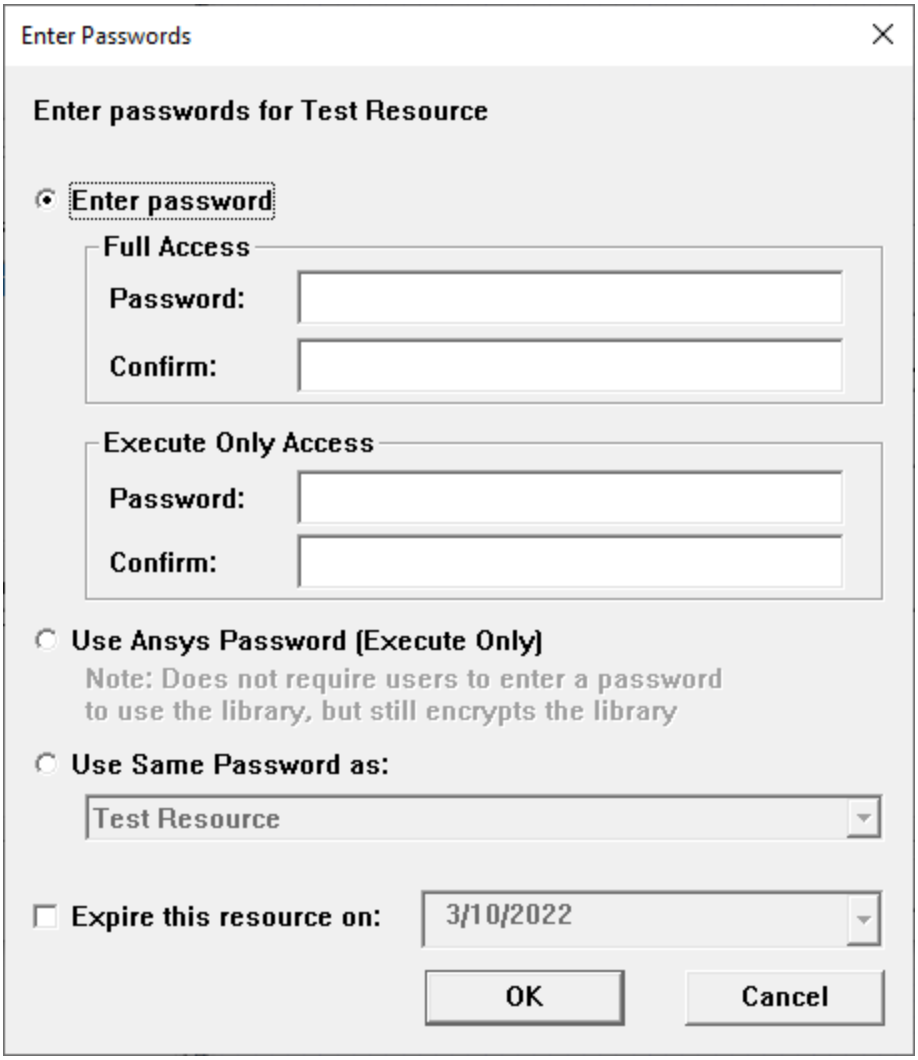
Electronics Desktop lets users create encrypted password-protected files (typically in SPICE format) that may be added to other designs via *.lib*, *.va*, or *.inc* statements. Access the encryption tools through the **Tools** > **Password Manager** window.

Password Assignment Types

From the **Enter Passwords** window, accessible on the **Password Manager** > **New...** option, an encrypted file may be assigned a choice of password types:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Enter password**: User-defined passwords for an encrypted library file. Entering and confirming **Full Access** passwords grants users simulation, read, and edit permissions, while **Execute Only Access** permits users to add components and simulate a design, but they cannot read or edit the contents.
- **Use Ansys Password (Execute Only)**: A generic, hard-coded execute-only password shared across all Ansys products. Any Ansys product may read an applicable file encrypted in any other Ansys tool using the **Ansys Password**, making it the easiest way to share encrypted library files among Ansys applications if the files are not expected to be viewed or editable.
- **Use Same Password as**: Choose an existing resource on the drop-down menu to use the same password settings and synchronize passwords across multiple library files.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

1. Passwords are not saved between sessions and must be re-entered each time **Electronics Desktop** is started. Passwords *are* available after one project is closed and another is opened, if the application is not restarted.
2. **Electronics Desktop** and Nexxim use the Advanced Encryption Standard (AES) to encrypt libraries and models. **Electronics Desktop** and Nexxim use a 128 bit block cipher with a 256 bit key.
3. You can encrypt selected parts of a file by enclosing them within a protected “block”. You can set up multiple .protect/.unprotect blocks in the same file, but you cannot nest them.

   e.g.,

   .protect

   This block is encrypted.

   .unprotect

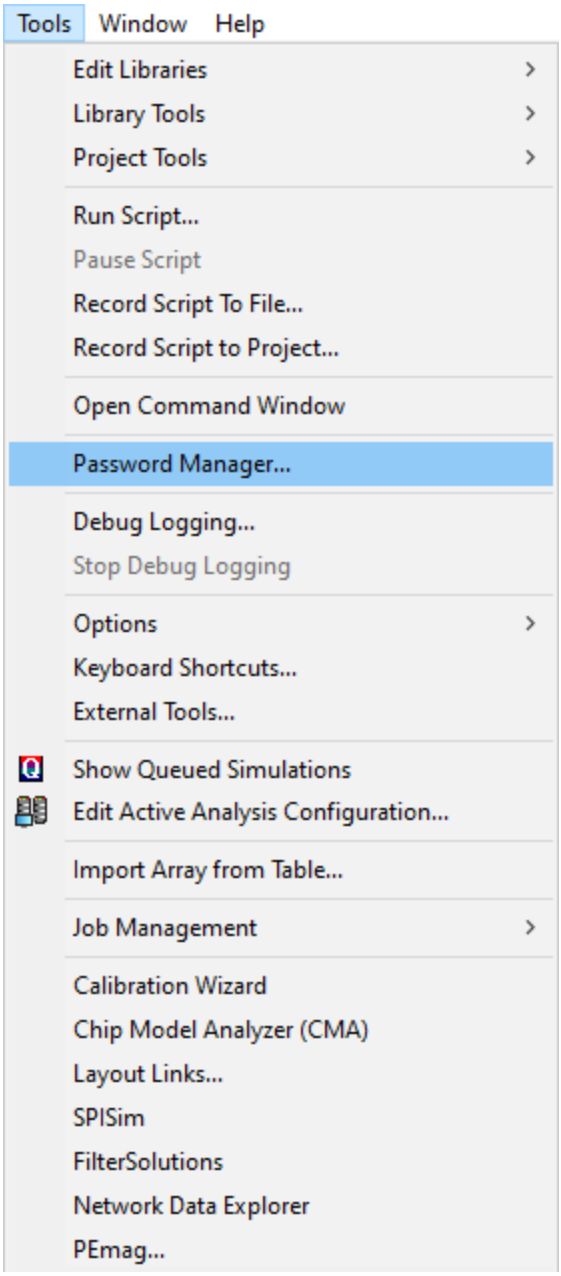
Setting Up Passwords and Encrypted Libraries

Because an encrypted library requires a password, encryption is done through the **Password Manager** window. The **Password Manager** windows lets users create a password or password set as a "password protected resource", which is a group of encrypted files which use the same password. Once the resource and its password type is defined, select it as the password for one or more newly encrypted libraries.

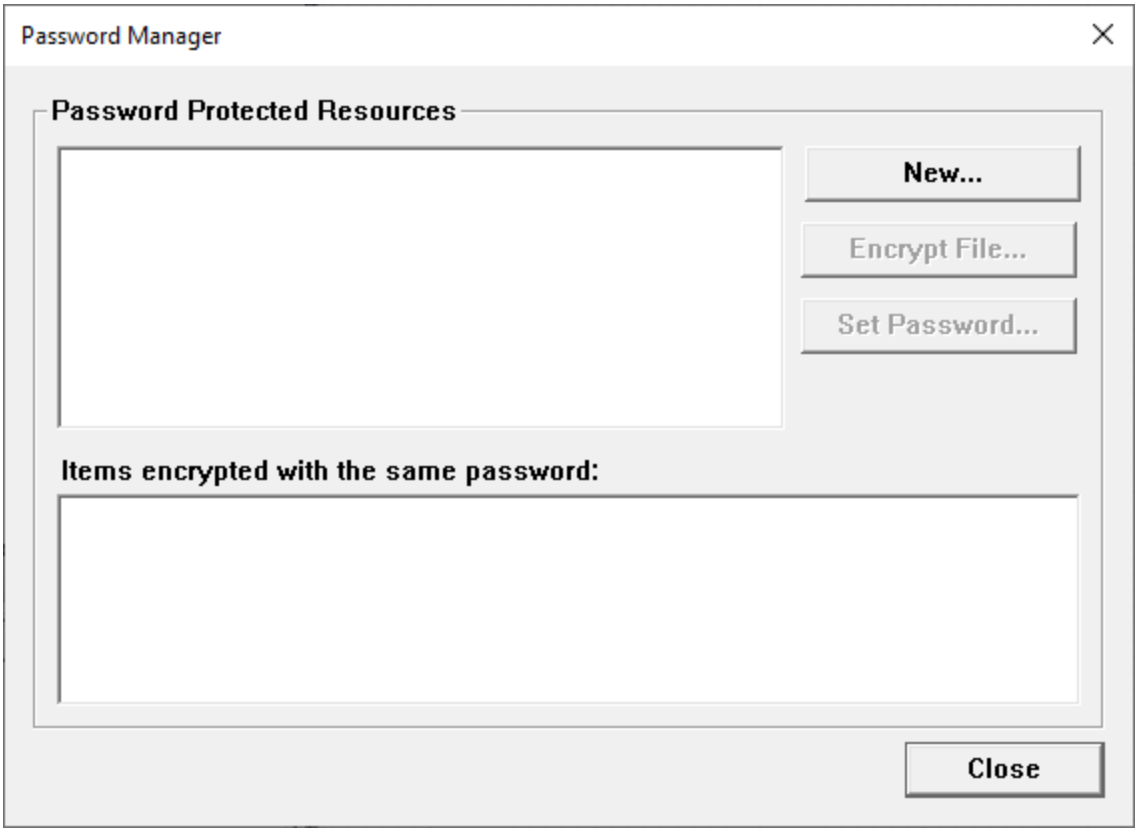
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Generate a New Password Protected Resource

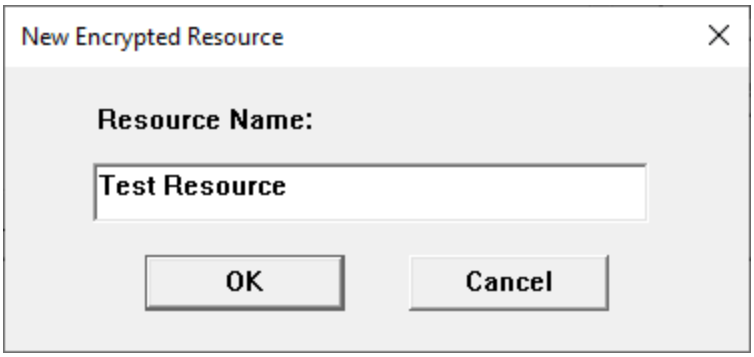
1. From **Tools**, select **Password Manager...** to open the **Password Manager** window.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

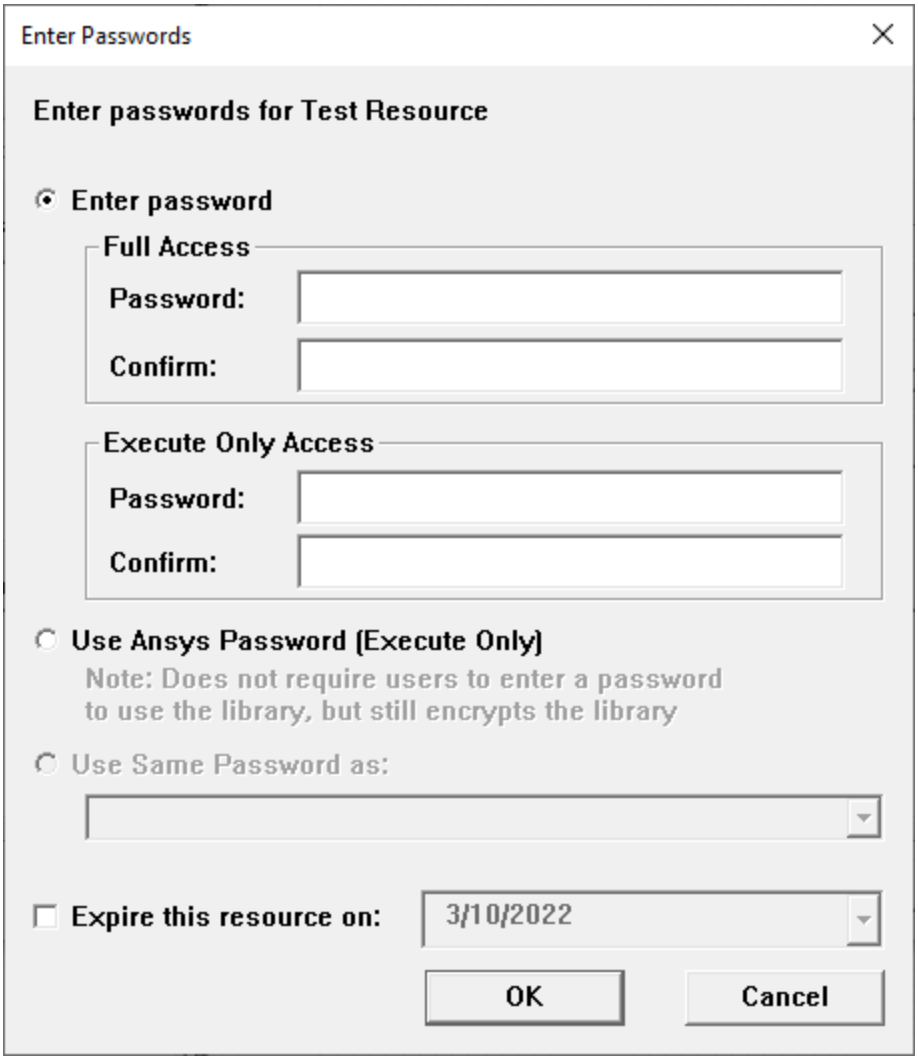


2. Click **New...** to open the **New Encrypted Resource** window.



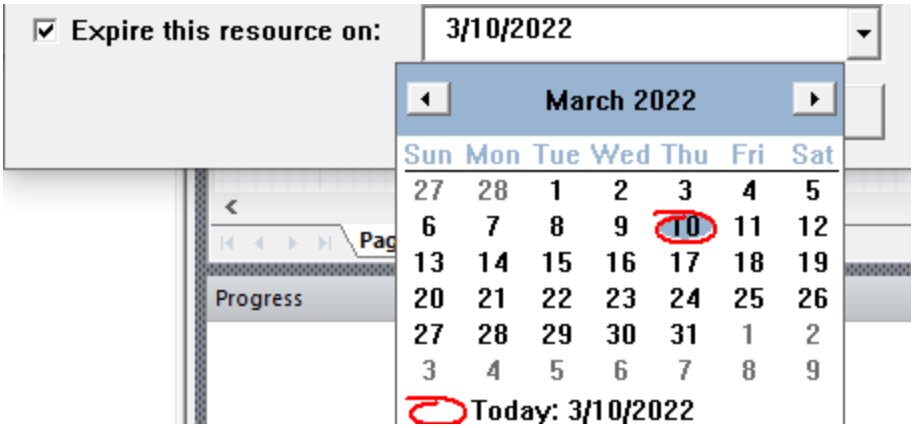
3. Type a name for the resource into the field. Then click **OK** to simultaneously close the **New Encrypted Resource** window and open the **Enter Passwords** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

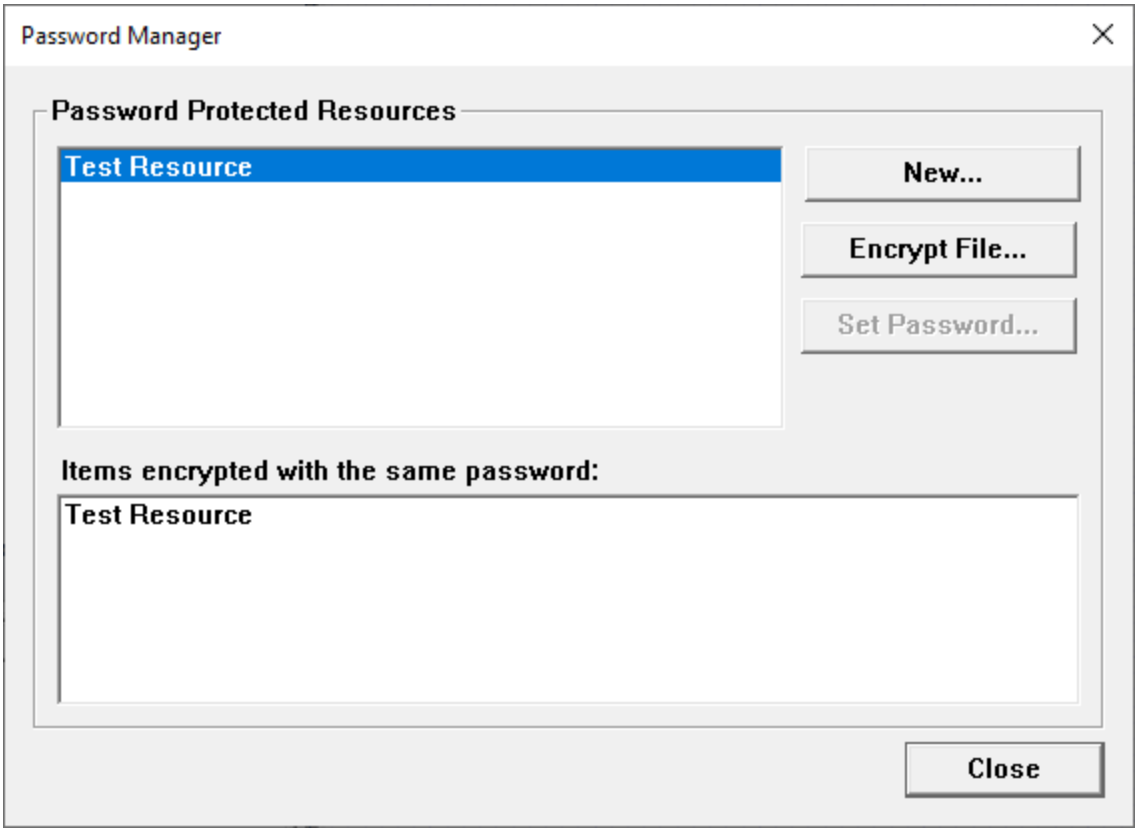


4. Choose on the following:
    - **Enter password**: choose between **Full Access** and **Execute Only Access**, then enter and confirm the new password in the given text fields.
    - **Use Ansys Password (Execute Only)**: as stated, this choice does not require users to enter a password to use the library, but still encrypts the library.
5. Optionally, assign an expiration date for the encrypted library by checking the **Expire this resource on** box and selecting a date on the drop-down calendar. If an expiration date is assigned to a library during encryption, the library cannot be used after the assigned date.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

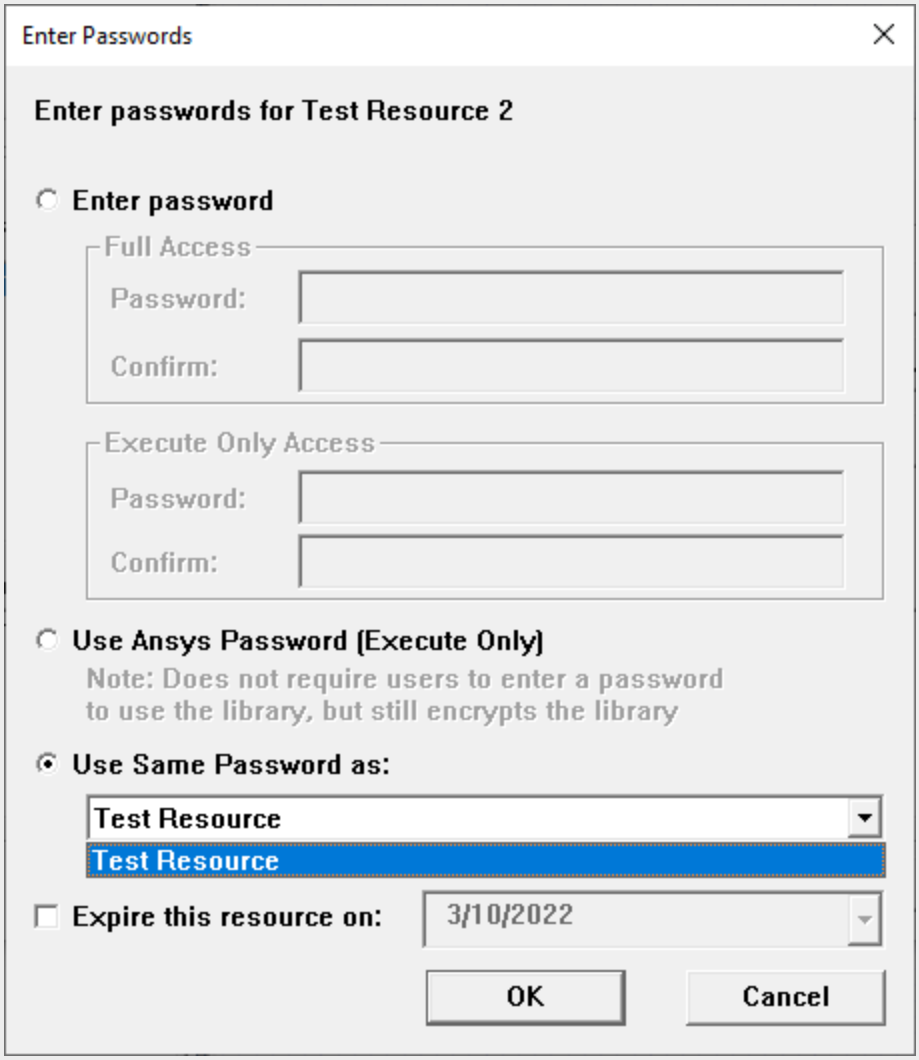


6. Click **OK** to add the new resource to the **Password Manager** list and simultaneously close the **Enter Passwords** window. Any number of libraries may be assigned to the new resource.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: Once one or more resources have been added to the **Password Protected Resources** list on the **Password Manager** window, **New** resources can use the same password profile. From the **Enter Passwords** window, select **Use Same Password as**, which was previously unavailable, and choose the existing resource on the drop-down menu.

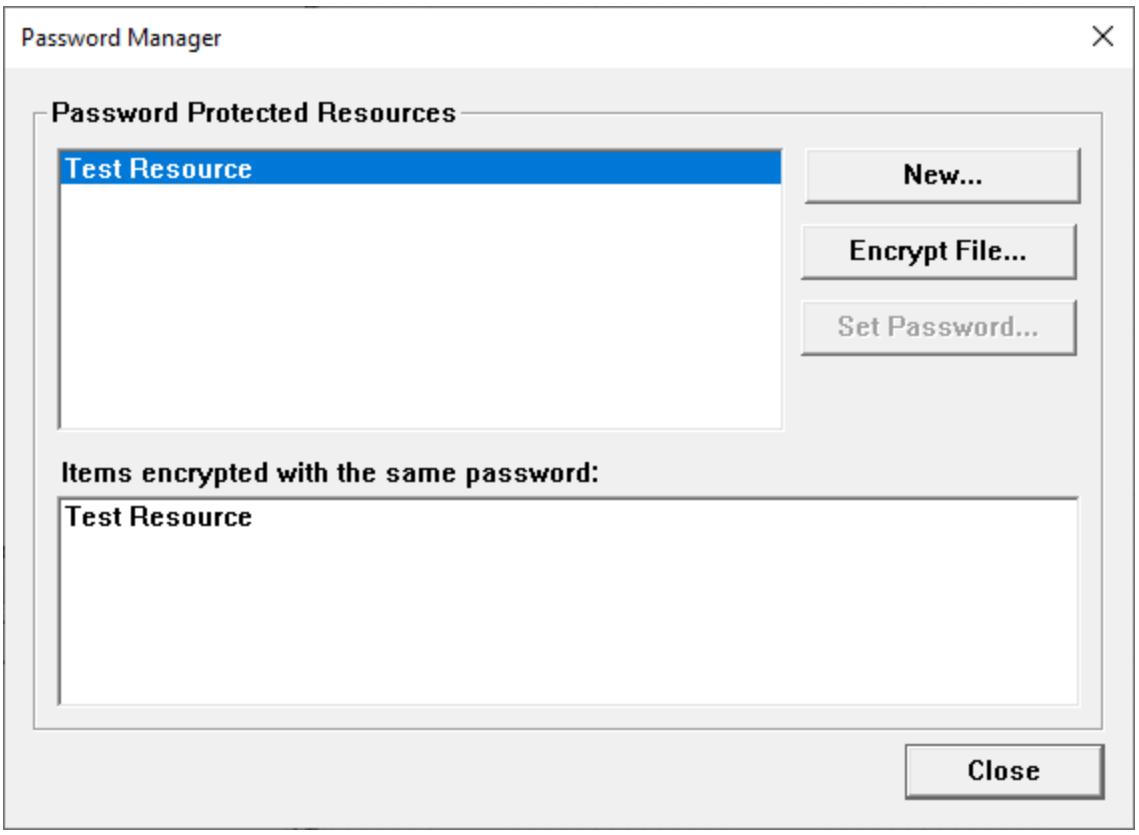


Generate a New Encrypted Library

When a library file is encrypted, the user can choose to overwrite the existing target library with the encrypted one, or save the encrypted library file with a new name and/or to a new location. During encryption, the user is prompted to choose whether the process respects .prot[ect] statements or ignores them. To generate a new encrypted library, complete the following steps.

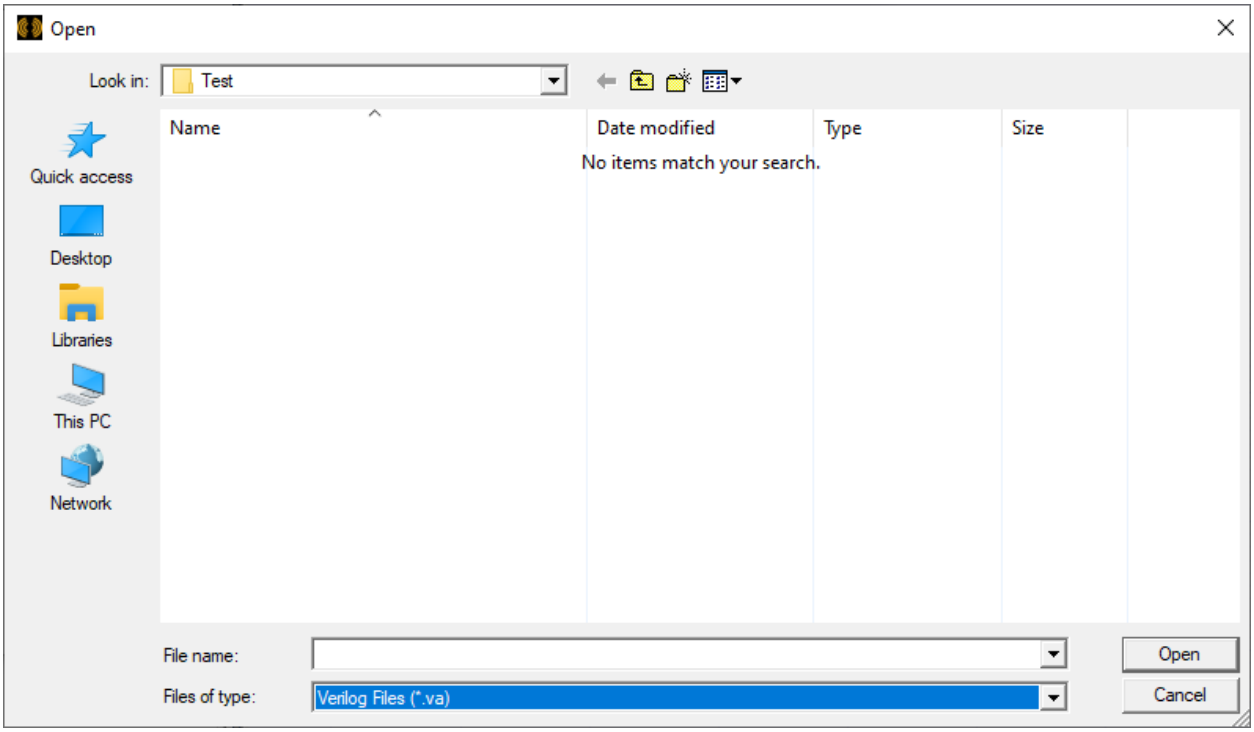
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. From the **Password Manager** window, select a resource on the **Password Protected Resources** list to enable the **Encrypt File...** button.



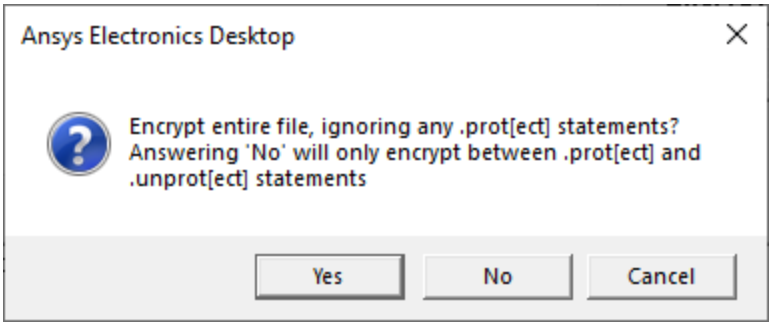
2. Click **Encrypt File...** to open an explorer window. Navigate to the appropriate library file.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:** Only library files with **Circuit (.*lib)** and **Verilog (.va)** file extensions are valid.

3. Select a library file, or first narrow the available choices by selecting **Circuit Files (.*lib)** or **Verilog Files (.va)** on the **Files of type** drop-down menu. Then click **Open** to immediately open the prompt: **Encrypt entire file, ignoring any .prot[ect] statements?**



4. As stated in the prompt, choose on the following:
   - Click **Yes** to permanently encrypt the entire library file.
   - Click **No** to recognize any **.prot[ect]** statements and encrypt only the remaining

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

aspect.

- Click **Cancel** to exit out of the prompt and return to the previous explorer window.

5. After the library file is encrypted, an explorer window will appear. Navigate to an appropriate location to save and/or rename the file, if appropriate. If the encrypted library file is saved to the same location and with the same file name as the original unencrypted library file, the unecrypted file is overwritten permanently.
6. Click **Save** to close the explorer window and create the encrypted library file.

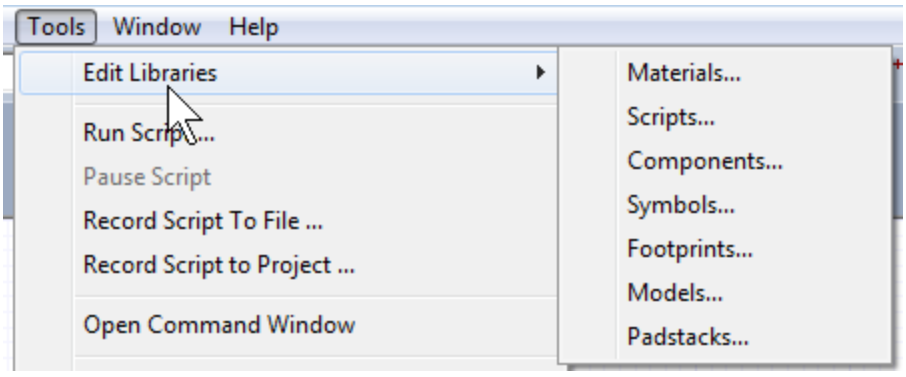
Using the Library Editor

The Electronics Desktop **Library Editor**, also called the **Edit Libraries** window, allows you to access, manage, save, and control the contents of various development libraries which contain information and properties for the following design entities:

- Materials
- Scripts
- Components
- Symbols
- Footprints
- Models
- Padstacks

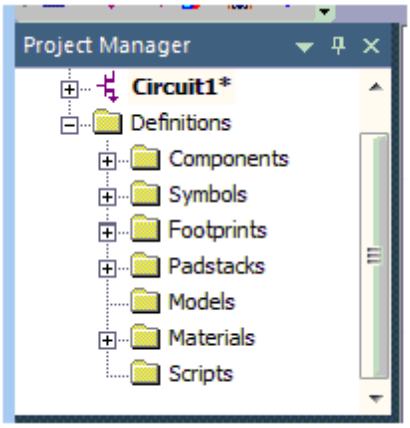
Starting the Library Editor

To start the **Library Editor**, select **Tools** > **Edit Libraries**, then select the type of library you wish to access on the following menu:



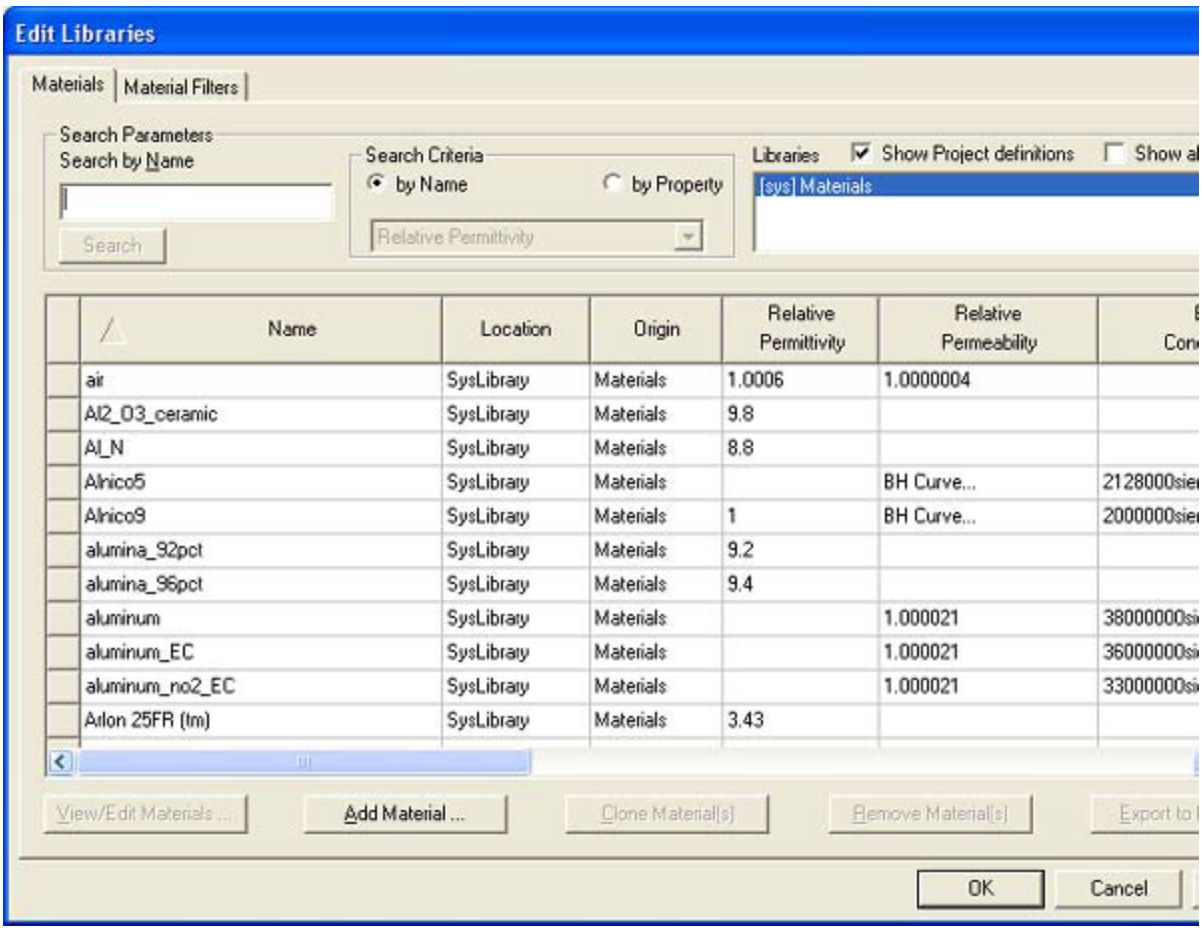
You may also right-click any of the folders listed in the **Definitions** directory of the **Project Tree** in the **Project Manager** window, then select **Edit Library**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Edit Libraries Window

After choosing a library to edit, the **Edit Libraries** window opens with a tab open to the library you selected, such as the **Materials Browser**.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Edit Libraries** browser searches library objects using filters based on a number of attributes:

- Name
- Property
- Model type

You can customize the **Edit Libraries** window by clicking **Show Project definitions** and **Show all libraries** (to override filtering).

The following **Edit Libraries** browser controls are available:

- **Edit** – Edits properties/attributes for the selected object. The resulting object can have the same name as the original, but it is saved to the current project, rather than back to the library. In effect, you have “checked out” the part in order to edit it. If you want to write it back to the library and overwrite the original part, use **Export to Library**.
- **Add** – Creates a new object in the selected library.
- **Clone** – Creates a copy of a selected object with a different name in the selected library.
- **Remove** – Removes an object on the library
- **Export** – Exports edited objects from a project to a library, or exports selected objects from one library to another. If the component you are exporting is hierarchical, after you click to **Export** and specify a library name in the **Export to user library** window to open the **Export Hierarchy** window. For more information see Exporting Hierarchical Components.

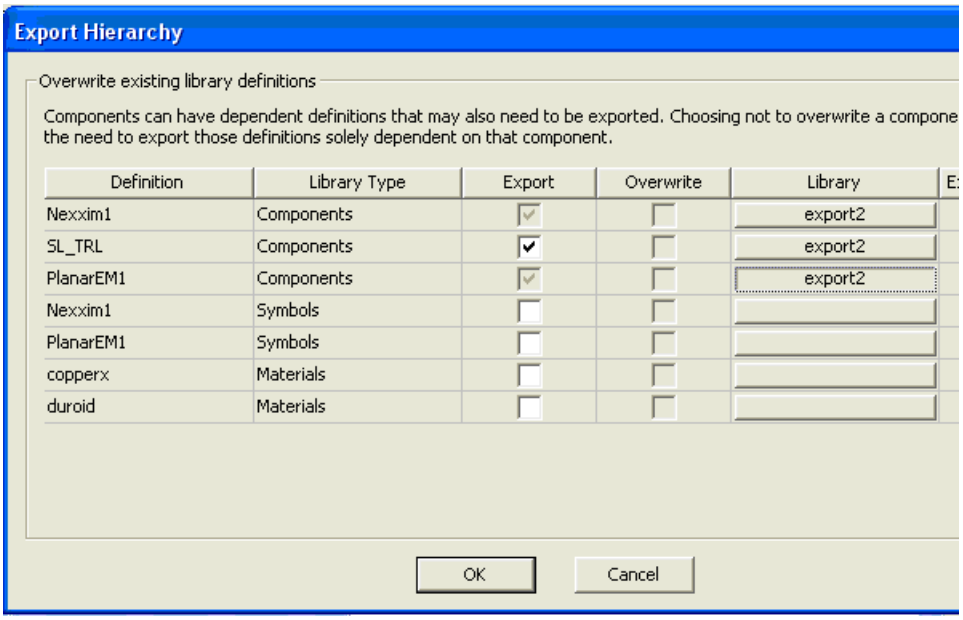
Note:

1. When you modify a library component, the modified definition is automatically transferred to the components available to the current project. In effect, you have “checked out” the component to modify it, and it is now in the project. The original library definition is intact. To update the library definition, you must do the extra step of exporting the modified component on the project to the library.
2. Subcircuits automatically inherit higher-level library configurations.
3. Project variables and project datasets used in designs represented by hierarchical components is saved to the library with the component and restored when the component is used. If the variable or dataset already exists in the project (determined by name), the library version is ignored. When creating project variables or datasets for hierarchical component designs to be saved in a library, ensure the names of the variables and datasets are appropriate and not likely to clash with those of other hierarchical components or common defaults.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Exporting Hierarchical Components

Electronics Desktop allows you to export hierarchical components, along with their solutions. Click **Export** to export a hierarchical component in the Library Editor and specify a library name in the **Export to user library** window to open the **Export Hierarchy** window.



- The component selected to be exported appears in the top-most row of the window. The library to which it is exported is listed beneath the **Library** column (in this example, “export2”). To select a different library to export to, click the button in the **Library** column (in this example, “export2”).
- Dependent definitions appear underneath the top-most row of the window and may also be exported. Definitions that are mandatory for export has an unavailable **Export** check box.
- You can export a component’s solution by checking its **Export** group box. At import time, the solution is re-associated with the design.
- Symbols need not be exported, they is auto-generated at import time.
- Each definition is exported as a definition in its own right. That is to say, at import time, using the above example, "Planar EM1" is visible in the "export2" library as a component independent of "Nexxim1".
- Components that originated in libraries (e.g., the "SL_TRL" component) need not be exported. At import time, each library component is retrieved from its original library and re-associated with the design.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14 - Adding Solution Parameters

When you assign a solution parameter, the solver calculates the parameter value as part of the solution process. The table below shows the solution parameters that may be calculated based upon the solver type.

Parameter	3D Solvers	2D Solvers
Force/ Torque	Magnetostatic Electrostatic AC Magnetic (Eddy Current) 3D AC Conduction Transient Electric Transient	Magnetostatic Electrostatic AC Magnetic (Eddy Current) Transient
Matrix	Magnetostatic Electrostatic AC Magnetic (Eddy Current) DC Conduction AC Conduction	Magnetostatic Electrostatic AC Magnetic (Eddy Current) AC Conduction DC Conduction

Assigning a Force Parameter

> **Note:** Force parameters are available for calculation in the 2D and 3D Magnetostatic, Electrostatic, AC Magnetic, and Transient solvers, and for the 3D AC Conduction and 3D Electric Transient solvers.

To solve for a force parameter:

1. Select the object on which you want to apply the parameter.
2. Click **Maxwell 2D** or **Maxwell 3D,** and then select **Parameters** > **Assign** > **Force**. You can also right-click on **Parameters** in the Project Manager tree, and then select **Assign** > **Force** from the context menu.
3. In the **Force Setup** window that appears, type a name for the force in the **Name** box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. For **Maxwell 3D** projects only, select **Virtual** or **Lorentz** as the **Type** for the force.

> **Note:**
>
> - For 3D Magnetostatic, Electrostatic, AC Magnetic T-Omega, and Transient solvers, you can select either **Virtual** or **Lorentz** force to be calculated.
> - For the 3D AC Magnetic A-Phi solver, only **Lorentz** force is supported.
> - For 3D AC Conduction and 3D Electric Transient solvers, only **Virtual** force is calculated.
> - See Average Force in an AC Magnetic Solution for a discussion on DC, AC, and instantaneous components of force.

5. Select the **Post-Processing** tab to specify the coordinate system reference for calculating the force parameter.

> **Note:** If object or face coordinate systems are used as reference in the post processing tab of the force setup in a transient analysis with motion, it should be noted that the original position of the CS will be used for this coordinate transformation. This coordinate transformation will not consider the motion enforced by the motion setup.

6. Select the appropriate coordinate system from the **ReferenceCS** drop-down menu.
7. Click **OK**.

> **Note:** The calculation of forces using the A-Phi formulation requires an extremely fine mesh because B is solved as 0th order and is constant at each node of each tetrahedral element.

Assigning a Torque Parameter

> **Note:** Torque parameters are available for calculation in the 2D and 3D Magnetostatic, Electrostatic, AC Magnetic, and Transient solvers, and for the 3D AC Conduction and 3D Electric Transient solvers.

To solve for a torque parameter:

1. Select the object on which you want to apply the parameter.
2. Click **Maxwell 2D or Maxwell 3D**, and then select **Parameters** > **Assign** > **Torque**. You can also right-click on **Parameters** in the Project Manager tree, and then select **Assign** > **Torque** from the context menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Torque** setup window appears.

3. Type a name for the torque in the **Name** box.
4. Select **Virtual** or **Lorentz** as the **Type** for the torque.

> **Note:** For 3D AC Conduction, 3D Transient and Electric Transient solvers, only **Virtual** torque is calculated.

> **Note:** For the 3D AC Magnetic A-Phi solver, only the **Lorentz** torque is calculated.

5. To assign an axis:
   a. Select the axis from the **Axis** drop-down menu.
   b. Select either the **Positive** or **Negative** radio button.
6. Click **OK**.

Assigning a Matrix

To solve for a capacitance, inductance, impedance, or conductance matrix:

1. Click **Maxwell 2D** or **Maxwell 3D**, and then select **Parameters** > **Assign** > **Matrix**.

   The **Matrix** dialog box appears.

2. Click the **Setup** tab.
3. Type a name for the matrix in the **Name** box.
4. To specify the sources to be included in the matrix, do one of the following:
   - For 3D Electrostatic, 2D and 3D Magnetostatic, 2D AC Magnetic, and 3D AC Magnetic T-Omega designs, select or clear the **Include** check box for any of the listed sources. For **Maxwell 2D** designs, the return path for each source may be specified. By default, the return path is at infinity; however, any conductor with a source specified can act as the return path.
   - For 2D Electrostatic, 2D and 3D DC Conduction, 2D and 3D AC Conduction, and 3D AC Magnetic A-Phi, select or clear the **Signal** or **Ground** check box for any of the listed sources. Grounded terminals are treated as reference voltages with zero voltage values. Excitations checked as **Signal** will be in the matrix (excited one at a time with 1 V during the parameters extraction process). Those excitations checked as **Ground** will be kept at 0 V and will not be part of the matrix. Any excitations not checked will not be considered.
5. For Magnetostatic designs, click the **Post Processing** tab.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If more than one source is listed on the left side, you can click the **Post Processing** tab and **Group** them to organize multiple sources into a single group. The calculations defined in the Post Processing tab do not impact the field solution and can be set up or changed after the solution has been completed.

To **Group** multiple sources together:

a. If desired, set the number of **Turns** for each source.
b. Select the sources to group in the list **Entry** list. Multiple source can be selected by holding the **Ctrl** key and clicking each source line.
c. With all sources selected, click **Group** to combine the sources into one series winding.
d. Set the number of parallel **Branches** the winding should be divided into and click **OK**.
e. The results of the post processing calculation on the matrix can be seen by selecting **PostProcessed** on the **Matrix** tab of the solutions dialog box.

6. Click **OK**.

Note:

- For a matrix setup, the elements are defined based on excitations, not solids. Only excitations will be available in the **Matrix** dialog box, regardless of the current geometry selection. Also, changes made to the excitations after matrix setup will not be flagged until the setup validation performed prior to the solution process.
- Matrix setups are recognized when coupling Maxwell designs to Twin Builder components such as Equivalent Circuit, Dynamic Inductance and Capacitance, and State Space. For more information on coupling Maxwell designs to Twin Builder components, refer to the *Maxwell Component Subcircuits* topic in the Twin Builder Help.
- For 2D AC Magnetic and 3D AC Magnetic T-Omega designs, you can also *Assign a Reduce Matrix*.
- For the 3D AC Conduction and 3D AC Magnetic A-Phi solvers, the current excitations might also be considered during the matrix extraction. They will be considered in the same way as the regular voltage excitations.
- The 3D AC Magnetic A-Phi allows to extract both RL and CG matrices. For more information about the matrices setup, refer to Technical Notes: 3D AC Magnetic A-Phi Parameter Extraction.

In a matrix calculation:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

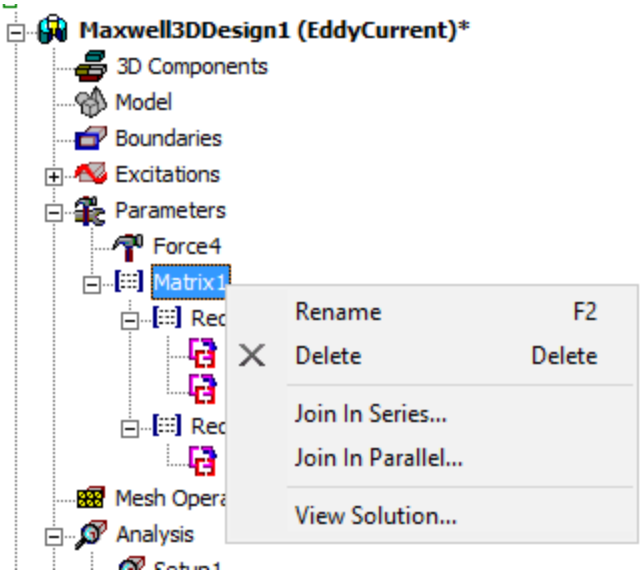
- For coil terminals, all terminals are listed and can be selected.
- For outer terminals, the solver can only handle a conduction path with two terminals, and only one terminal is listed for each conduction path:
- If they are current terminals, the listed terminal is in the “in” direction.
- If they are current density terminals, the first terminal is listed and can be selected.
- When setting up an inductance matrix, the source must be a current excitation. Voltage sources cannot be used in this case.
- In 2D matrix calculations, the solver calculates loop currents; therefore, the direction of return path current is always opposite of the source current. If a conductor with excitation is specified as the return path, the polarity setting of the excitation in the nominal problem setup is not respected.

Assigning a Reduce Matrix for 2D and 3D AC Magnetic Designs

For 2D and 3D AC Magnetic designs, you can join (group) two or more excitations to one excitation in either a series or a parallel connection. The result of each join (group) operation is known as a Reduce Matrix. This would be useful when analyzing designs such as a motor with four poles, each pole having three wires coming into it, with the physical design having each set of three wires in series, and the four bundles of wires in parallel.

To create a reduce matrix:

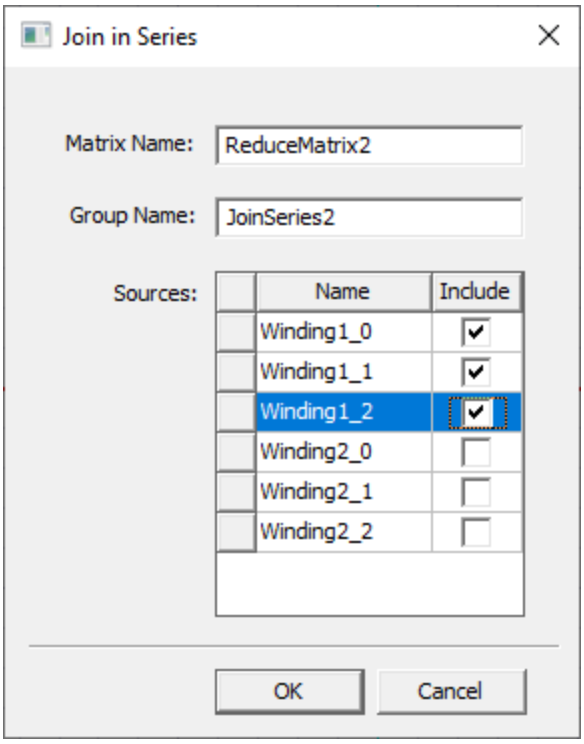
1. Right-click on an existing Matrix parameter in the Project Manager tree.



2. Select either **Join in Series** or **Join in Parallel**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A corresponding **Join in Series** or **Join in Parallel** dialog box opens. You can accept the default reduce **Matrix Name**, or enter the desired name.



3. You can enter a **Group Name** (or accept the default), and check **Include** for those sources you wish to join in series (or parallel).

   > **Note:** Multiple group operations are supported. For example, you could join conductors 1, 2, and 3 in one series; and conductors 4, 5, and 6 in another series.

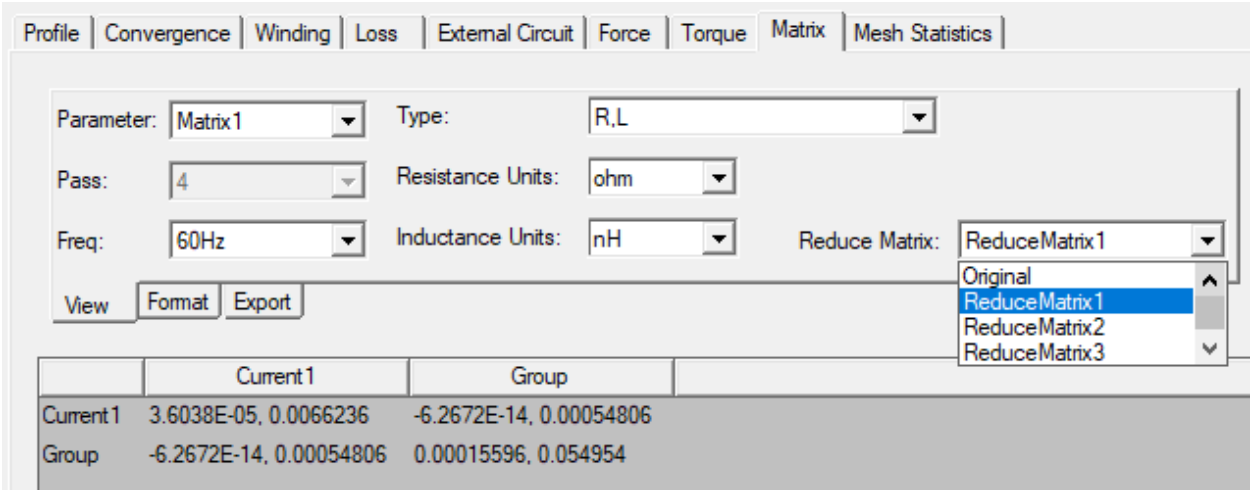
4. Click **OK** to close the dialog and add the new reduce matrix group under its parent Matrix.

> **Note:** If you want to make changes to a reduce matrix, you can double-click on the reduce matrix group item in the tree to reopen the dialog box. You can also right-click the item to rename or delete it.

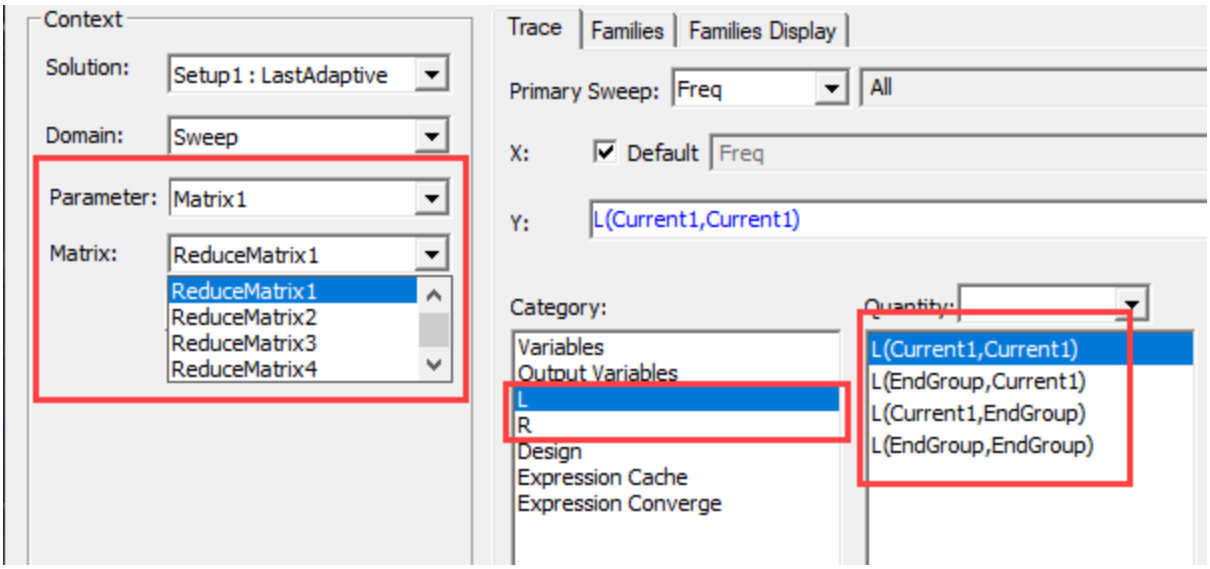
If you want to make changes, you can double-click on the reduce matrix group item in the tree to reopen the dialog box. You can also right-click the item to rename or delete it.

You can view the results of reduce matrix calculations on the Matrix tab of the Solutions dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can select reduce matrices to plot inductance and resistance quantities for various report types.



Reassigning Parameters

To reassign the geometry a parameter applies to:

1. Select the new object to which you want to assign an existing parameter.
2. Click **Maxwell 2D** or **Maxwell 3D**, and then select **Parameters** > **Reassign**.
3. In the **Reassign Parameter** window that opens, select the force or torque you want to

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

reassign from the **Reassign Geometry for** list.

4. Click **OK**.

The parameter is reassigned to the new object.

Viewing Parameter List

To view a list of all parameters in the design:

1. Click **Maxwell 2D** or **Maxwell 3D,** and then select **Parameters** > **List**.

   The **Design List** window appears, with the **Parameters** tab visible.

2. To view other design choices, click one of the following tabs:
   - Model
   - Boundaries
   - Excitations
   - Mesh Operations
   - Analysis Setup

Deleting All Parameters

To delete all parameters assigned in the model:

Click **Maxwell 2D** or **Maxwell 3D**, and then select **Parameters** > **Delete All**.

All parameters are deleted.

> **Note:** Maxwell provides no warning or option to stop deletion. If you accidentally delete all parameters, click **Edit** > **Undo Delete All Parameters** before you take any other action.

Showing and Hiding Solution Parameters

1. Click **Maxwell 2D** or **Maxwell 3D**, and then select **Parameters** > **Visualization**.
2. In the **Parameter Visualization Options** window that appears, select the type of parameter you want to view from the **Select** drop-down menu, or click the row in the table. Choices include **All**, **All Forces**, **All Torques**, **All Matrices**, and **By Name**.
   The object is highlighted in the geometry.
3. Select or clear the check boxes for each parameter to show or hide geometry, name, and vectors for each force or torque parameter.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **Deselect All** to hide all parameters.
5. Click **Close** to close the window.

Setting the Visibility for Parameters

To show or hide a parameter:

1. Click **View** > **Active View Visibility**.
2. From the **Active View Visibility** dialog box that appears, click the **Parameters** tab.
3. Select or clear the **Visibility** check box for each parameter listed.
4. Click **Done**.

Changing the Parameter Setup

To change a parameter's setup:

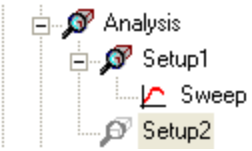
1. In the Project Manager tree, expand the **Parameters** entry.
2. Right-click the parameter you want to change, and select **Properties** from the shortcut menu.
3. Make the desired changes, and then click **Close**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15 - Specifying Solution Settings

Specify how Maxwell computes a solution by adding a *solution setup* to the design. You can define more than one solution setup per design.



Each solution setup includes general data about the solution's generation and adaptive mesh refinement parameters if you want the mesh to be refined iteratively in areas of highest error.

To add a solution setup to a design:

1. Select a design in the Project Manager tree.
2. Click **Maxwell 3D** > **Analysis Setup** > **Add Solution Setup** or **Maxwell 2D** > **Analysis Setup** > **Add Solution Setup**.

   Alternatively, right-click **Analysis** in the Project Manager tree, and then click **Add Solution Setup** on the shortcut menu.

   The **Solve Setup** dialog box appears containing several tabs, which are determined by the solution type (see Available Tabs by Solution Type).
3. Specify the desired settings on the tabs.
4. If you want to disable an individual analysis, uncheck the **Enabled** check box. Refer to Disabling and Enabling an Analysis Setup for additional information.
5. If you want to use the default values, click **Use Default**.
6. Click **OK**.

Available Tabs by Solution Type

For Non-Transient Solutions, the following tabs are present:

General	Includes general solution settings. Also contains a button for setting **HPC and Analysis Options**, which allows you to select or create an analysis configuration.
Convergence	Includes settings for convergence.
Expression Cache	Contains an optional list of expressions that you can specify for convergence criteria.
Solver	Includes settings for nonlinear residual and matrix solver types.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	• If an AC Magnetic solution is selected and the Maxwell 3D AC Magnetic Solution Refinement Control has been enabled from **Tools** > **Options** > **General Options** > **Desktop Configuration** > **Beta Options**, this tab will also include Loss Adaptive Control for better modeling of skin effects. • If it is a Maxwell 2D design, this tab will also include a Thermal Two-Way Coupling\Relaxation setting.
Frequency Sweep	Includes setting for varying the frequency of an AC Magnetic or AC Conduction solution. **NOTE:** This tab is present only for AC Magnetic and AC Conduction solutions.
Defaults	Enables you to save the current settings as the defaults for future solution setups or to revert the current settings to Maxwell's standard settings.

For Transient Solutions, the following tabs are present:

General	Includes general solution settings. Also contains a button for setting **HPC and Analysis Options**, which allows you to select or create an analysis configuration.
Save Fields	Enables you to create a time point list for saving field solutions.
Advanced	Supports user control program, import mesh options, and starting/continuing from a previously solved setup.
Solver	• Includes settings for nonlinear residual solver type, for enabling the solver to fast reach steady-state, and for selecting the Time Decomposition Method. • If it is a Maxwell 2D design, this tab will also include a Thermal Two-Way Coupling\Relaxation setting.
Expression Cache	Contains an optional list of expressions that you can specify for evaluating solutions.
Defaults	Enables you to save the current settings as the defaults for future solution setups or to revert the current settings to Maxwell's standard settings.

For Electric Transient Solutions, the following tabs are present:

General	Includes general solution settings. Also contains a button for setting **HPC and Analysis Options**, which allows you to select or create an analysis configuration.
Solver	Includes settings for initial condition, temporal tolerance, computing power loss data, and importing a mesh.
Expression	Contains an optional list of expressions that you can specify for evaluating

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cache	solutions.
Defaults	Enables you to save the current settings as the defaults for future solution setups or to revert the current settings to Maxwell's standard settings.

Related Topics

Analysis Parameters

Setting Analysis Parameters for Non-Transient Solutions

Setting Analysis Parameters for Transient Solutions

Setting Analysis Parameters for Electric Transient Solutions

Disabling and Enabling an Analysis Setup

Resetting Analysis to Time Zero

Changing Memory Settings

Specifying the Solver Type

Technical Notes: Meshing Aspects for 3D Transient Applications With Motion

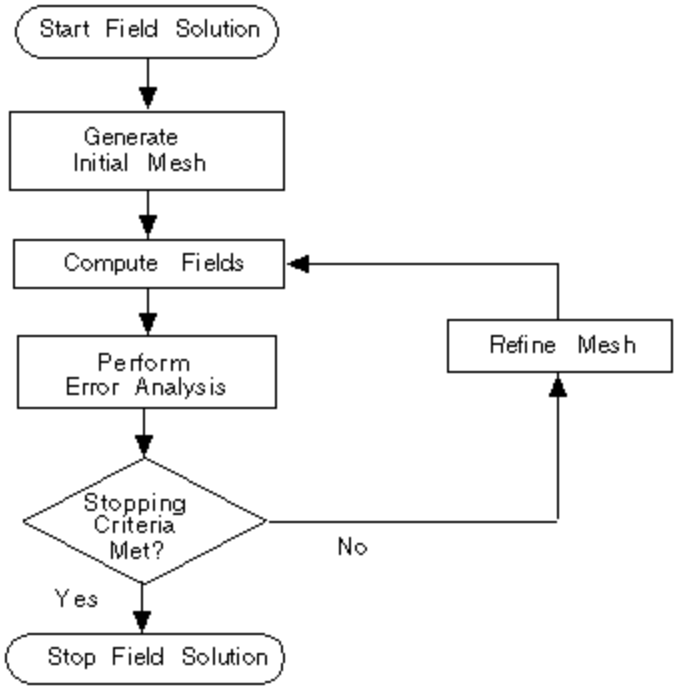
Defining Mesh Operations

Copying a Solution Setup

Analysis Parameters

During adaptive analysis, the system iteratively refines the starting mesh in order to reduce the size of individual elements in areas of high error — thus improving the accuracy of the solution.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When an adaptive analysis is performed, the following steps are executed:

- The Maxwell application generates a field solution using the specified mesh.
- It then analyzes the accuracy of the solution by calculating an energy value based on the error in the solution. The exact mechanism for evaluating the error varies by solution type. For example, in magnetostatic it can use Curl H to find the current density and then subtract all input currents and other sources. For a perfect solution the result would be zero, for a real, finite mesh the result is some amount of residual current density. An energy value calculated from this residual current density is called the error energy. The “Energy Error %” is the error energy as a percentage of the total energy (calculated with the original sources).
- If more than one pass has been completed, the software also calculates the change in total energy from the previous pass. The percentage difference is the “Delta Energy (%)”. Adaptive refinement continues until both the “Energy Error %” and the “Delta Energy (%)” are below the target Percent Error specified by the user (or until it reaches the Maximum Number of Passes requested).
- When the error targets are not satisfied the mesh is refined. This is generally done by subdividing the elements with the highest error energy into smaller elements.
- The user can optionally request to “Use Output Variable Convergence”. This is an additional stopping criterion. The Energy Error % and the Delta Energy must still be below the target Percent Error, but the software will also compute the specified Output Variable for each adaptive pass and will calculate the percentage change in that value for each

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

pass after the first (this is the “Output Var. Delta (%)”). The solution will continue until the energy error criteria are met and the Output Var. Delta is below the target “Max. Delta Per Pass” specified by the user for output variable convergence (or until Maximum Number of Passes).

Setting Analysis Parameters for Non-Transient Solutions

To set up a non-transient analysis, define the parameters on the following tabs of the **Solve Setup** dialog box (which appears when you click **Maxwell** > **Analysis Setup** > **Add Solution Setup**):

- General (Name, Adaptive Setup, Parameters)
- Convergence
- Expression Cache
- Solver
- Frequency Sweep (AC Magnetic and AC Conduction solutions only)
- Defaults

Defining Settings on the General Tab for Non-Transient Solutions

The following settings can be defined on the **General** tab of the **Solve Setup** dialog box for non-transient solutions:

- Setup Name
- Adaptive Setup
- Parameters

Setup Name for Non-Transient Solutions

By default, **Setup1** appears as the name of the first analysis you set up.

To change the name of the setup, type the new name in the **Name** text box on the **General** tab of the **Solve Setup** dialog box.

To disable the setup, uncheck the **Enabled** check box. Refer to Disabling and Enabling an Analysis Setup for additional information.

Adaptive Setup for Non-Transient Solutions

To specify adaptive settings, click the **General** tab in the **Solve Setup** dialog box and use the options in the **Adaptive Setup** section. Adaptive settings include the following:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Maximum Number of Passes**: Defines the maximum number of mesh refinement cycles you would like Maxwell to perform. This value is a stopping criterion for the adaptive solution; if the maximum number of passes has been completed, the adaptive analysis stops. If the maximum number of passes has not been completed, the adaptive analysis continues unless the convergence criteria are reached.

  > **Note:** The size of the finite element mesh — and the amount of memory required to generate a solution — increases with each adaptive refinement of the mesh. Setting the maximum number of passes too high can result in Maxwell requesting more memory than is available or taking excessive time to compute solutions.

- **Percent Error**: Controls the desired solution accuracy. Smaller values produce more accurate (but slower) solutions; larger values produce less accurate (but faster) solutions. At each step in the adaptive process, the energy and error energy are computed, and you can view the most recent solutions as soon as they are completed. After the mesh is refined, the matrix is calculated on the refined mesh. The relative change between the previous matrix and the current matrix is then computed and reported as the matrix delta; the target matrix delta is the **Percent Error**. A small delta indicates that further mesh refinement will probably not change the solution.

Parameter Settings

On the **General** tab of the **Solve Setup** dialog box, if you want to solve for fields only and no other parameters, then click the **Solve Fields Only** check box.

When this check box is cleared, the following options are available:

- **Solve Matrix**: Select **After last pass** or **Only after converging**.

Defining Settings on the Convergence Tab for Non-Transient Solutions

To set the convergence data, click the **Convergence** tab in the **Solve Setup** dialog box. Convergence settings can be standard or optional. Standard convergence settings include the following:

- **Refinement Per Pass** – The percent you set for **Refinement Per Pass** determines how many tetrahedra are added at each iteration of the adaptive refinement process. The tetrahedra with the highest error are refined.
- **Minimum Number of Passes** – The minimum number of adaptive passes even if the convergence criteria are reaches.
- **Minimum Converged Passes** – The minimum number of adaptive passes to continue after the convergence criteria are reached.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Optionally, convergence criteria can also be based on the value of an output variable. This condition is in addition to the normal stopping criteria. To base the convergence criteria on an output variable value, refer to Defining Expressions for Non-Transient Solutions.

For 2D AC Magnetic solutions only, improved accuracy may be obtained by including loss in the convergence criteria. To include the loss calculation in convergence, select the **Use Loss Convergence** check box.

Defining Expressions for Non-Transient Solutions

> **Note:** For information on defining expressions for transient solutions see
>
> - Defining Expressions for Transient Solutions
> - Defining Expressions for Electric Transient Solutions

You can specify additional convergence criteria through the use of expressions and output variables. The **Max Delta** or the **Max Percent Delta** defined for expression convergence represents the difference in values of the expressions between consecutive adaptive passes. If the difference in the value of the expression between consecutive passes is less than the **Max Delta** or the **Max Percent Delta** value, this part of the convergence criteria is satisfied.

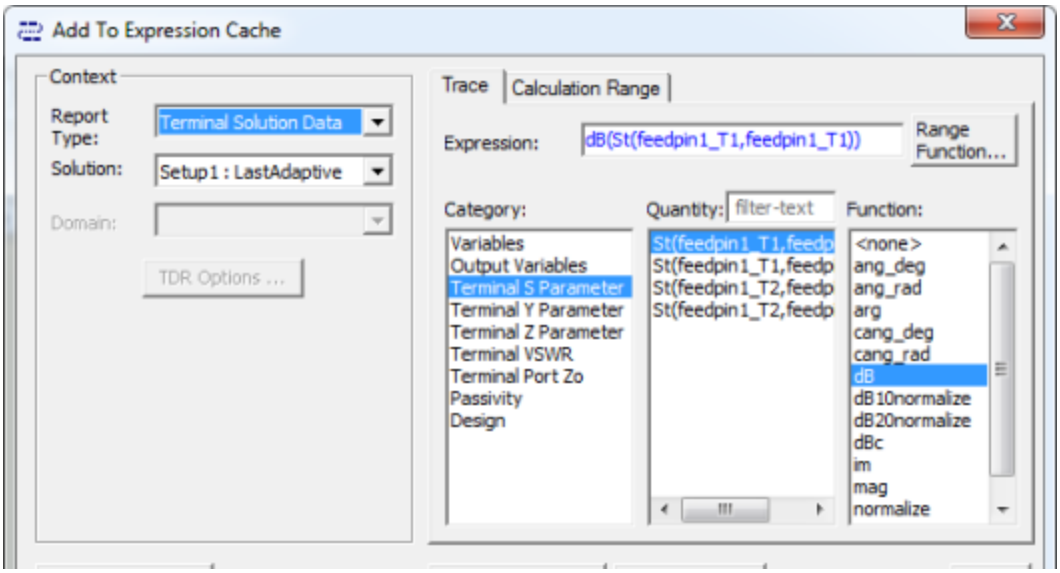
- For **driven solutions**, if the **Maximum Delta S**, **Maximum Delta E**, or alternate matrix convergence criteria are achieved in addition to any specified expression convergence criteria, the adaptive analysis stops. Otherwise, the solution continues until the requested number of passes is completed.
- For **eigenmode solutions,** if the **Maximum Delta Frequency Per Pass** criteria is achieved in addition to any specified expression convergence criteria, the adaptive analysis stops. Otherwise, the solution continues until the requested number of passes is completed.

To set expressions as **Convergence** criteria:

1. Double-click the Analysis setup icon in the Project tree to open the **Solution Setup** dialog.

2. Click the **Expression Cache** tab in the **Solution Setup** and click **Add...** to open the **Add to Expression Cache** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Specify the context for the expression you define. For Modal Solution Data, you do not need context other than the Solution. Other selections require more context:
    - Far Field also requires a geometry such as an infinite sphere.
    - Fields or Near Fields also require a geometry such as a polyline.
4. Under the **Trace** tab, select from the Category, Quantity and Function lists to create expressions.

    Selecting a listed category lists the Quantities and Functions available for each category. If you have defined one or more output variables, you can see them listed as Quantities by selecting the **Output Variables** Category. The Output Variables button opens a dialog that lets you define additional output variables.

    When you have created an expression, it appears in the Expression field of the **Trace** tab. If desired, you can use the Range Function button to select range functions to apply to the expression.

    Under the **Calculation Range** tab, you can view the values of available sweep variables. Clicking the ellipsis [...] button in the Edit column opens a list of values.

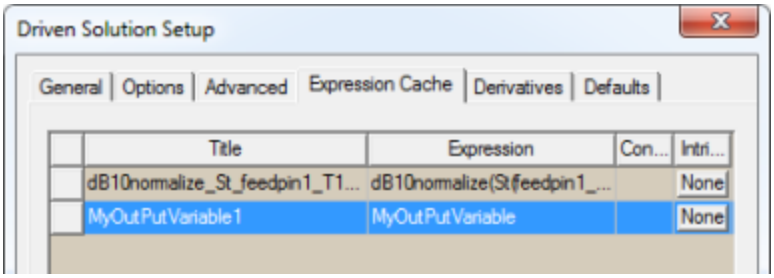
5. When you have created an expression that you want to add to the cache, click **Add Calculation**.

    This adds the selected expression and the associated context to a table in the **Expression Cache** tab. You can select any additional expressions with contexts and add them in the same way.

6. When you have added the expressions you want, click **Done** to close the **Add to Expression Cache** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Expression Cache** tab of the **Solution Setup** lists the expressions you have added as a table.



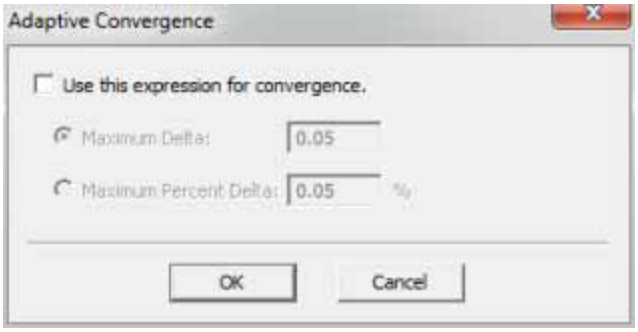
- The Title field is editable, by default showing the name as built from the expression, but removing underscores.
- The Expression field shows the full expression. If necessary, you can resize the **Solution Setup** dialog box. You can also resize each column in the table.
- The Context column shows None for Modal solutions, or the appropriate geometry for Fields calculations.
- The Intrinsics column shows a clickable button that opens an **Edit Calculation Range** dialog box. If the column button shows None you cannot edit the value. If the button shows variables, click the ellipsis [...] in the Edit column to display a list of the variable values that you can select. Click **OK** to close the **Edit Calculation Range** dialog box and apply your selections to the Expression Cache.
- The Convergence column contains a check box that lets you designate post processing variables, such as for renormalization or de-embedding. You can also use this feature to assign variables to non-model objects (for example, the properties of a solid, sheet or lines used for field calculations). You can also assign a PP Variable to a coordinate system (CS) that is not associated with model objects.

To excite particular sources for particular output variables, you can assign a post process variable to the magnitude of sources you might want to edit, and assign the expression value in the Expression Cache. With PP Variables, you can control convergence based on several combinations of active sources.

7. To designate one or more expressions for convergence, click the field for the Convergence column for each expression.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This opens an **Adaptive Convergence** dialog box.



Check **Use this expression for convergence** to enable the radio buttons. You can then specify the **Max Delta** between passes or the **Max Percent Delta** criteria. The **Max Delta** solves a potential issue if your expression is essentially zero and the numeric noise from pass to pass causes the maximum percentage delta to remain high. In that case adaptive refinement continues until you get to the maximum number of passes.

> **Note:** If you check **Use this expression for convergence**, the pass will be considered converged if both the energy error **and** the expression cache results have converged, not just expression cache.

8. For AC Magnetic: only check boxes allow you to save the expression cache either with **Evaluate Cache vs. Pass**, **Evaluate Cache vs. Freq**, or both.
9. Click **OK** to close the **Solution Setup** window.

Defining Settings on the Solver Tab for Non-Transient Solutions

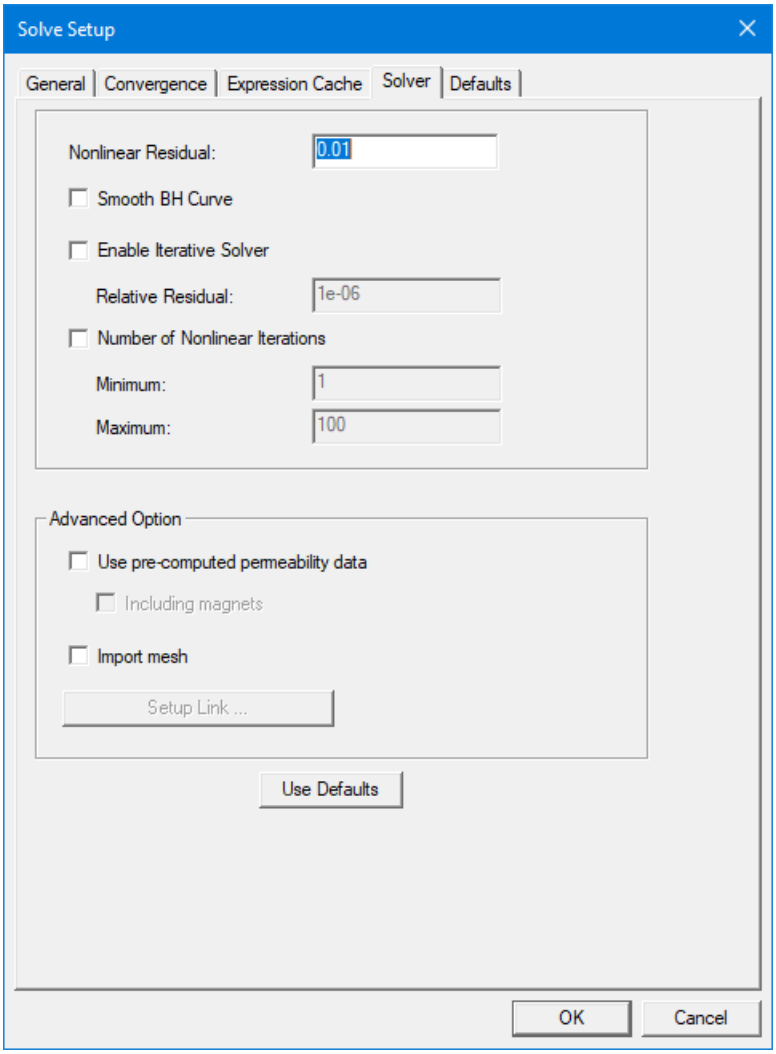
The settings on the **Solver** tab of the **Solve Setup** dialog box depend on the solution type selected for that project.

- Magnetostatic Solver Settings
- AC Magnetic Solver Settings
- Electrostatic Solver Settings
- AC Conduction Solver Settings
- DC Conduction Solver Settings

Magnetostatic Solver Settings

For magnetostatic solutions, you can define the following settings on the **Solver** tab of the **Solve Setup** window:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Enter a value in the **Nonlinear Residual** text box. This value specifies how close each solution must come to satisfying the equations that are used to compute the magnetic field.
- For both 2D and 3D magnetostatic designs, you can select the **Smooth BH Curve** check box. This allows the solver to "smoothen" the list of points to make a refined B-H curve for better solving. If this option is not selected, the solver just uses the discrete points.
- For 3D magnetostatic designs only, you can Enable Iterative Solver.
- For both 2D and 3D magnetostatic designs, you can select the **Number of Nonlinear Iterations** check box. This provides an option for you to set both a **Minimum** iteration number to avoid non-converged solution, and a **Maximum** iteration number to avoid taking too much computation time. These values must be integers greater than 0. By default the values for **Minimum** and **Maximum** are 1 and 100, respectively. Maximum number should be greater than the Minimum number.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Advanced Option** settings allow you to perform more sophisticated simulations of the magnetostatic field.
  - **Use precomputed permeability data** (for general soft magnetic materials) uses a previously solved mu (apparent permeability) distribution from a linked source design. Selecting this option enables the **Setup Link** button, and automatically opens the **Setup Link** dialog in which you set up the link. Optionally, you can override the solved mu with a user-specified program, which you can specify on the **Advance** tab of the **Setup Link** dialog box. Select the **Including magnets** check box if magnets are included in the distribution. Refer to "Permeability Options for Magnetostatic Solutions " on the facing page for details on using these settings.
  - The **Import mesh** check box can be selected in the target design if the mesh for the same geometry in the linked source design is to be used in the analysis. Selecting this option enables the **Setup Link** button, and automatically opens the **Setup Link** dialog in which you set up the link. Refer to "Import Mesh for Non-Transient Solutions " on page 15-39 for details on using this setting.
  - If an advanced option has previously been set up, you can click the **Setup Link** button to open the **Setup Link** dialog and edit the link settings.

    > **Note:** The data link used to link source and target designs uses data caching technology. Therefore, if the design changes, to make sure the new data (from the source design) is used in the target design, right-click the **Analysis** field of the target design in the project tree, and select **Clear Linked Data**. This flushes existing data.

- For **2D** designs, the **Thermal Two-Way Coupling\Relaxation** setting can be used to improve two-way coupling convergence by applying the relaxation factor to the temperature field. When the thermal modifier takes on a nonlinear form, the convergence of the solution field in Maxwell's equations may fail during two-way coupling iterations. With nonlinear iterations, using an under-relaxation factor can mitigate the risk of overshooting in the property updating process.

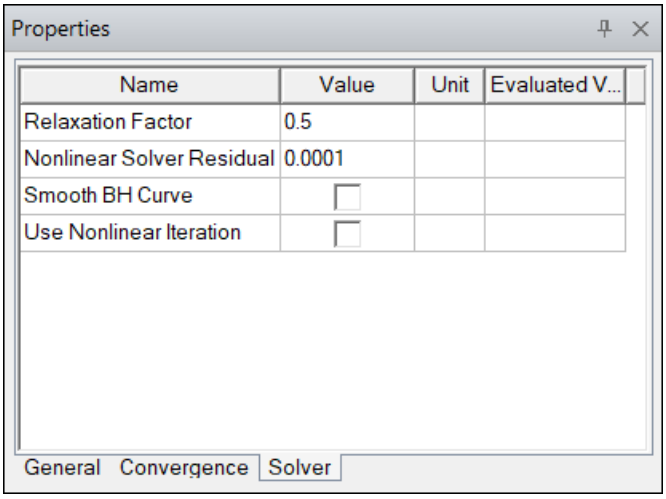
  The range of the relaxation factor is from 0 to 1. In general, a smaller relaxation value helps convergence, but may increase the total number of iterations. The default relaxation factor is 1.0.

  The **Thermal Two-Way Coupling\Relaxation** setting is applicable only when **Enable Feedback** is checked in Temperature of Objects window, which invokes two-way coupling instead of one-way coupling. The relaxation setting is

  - applicable to all Maxwell 2D designs,
  - applicable to both geometry modes (Cartesian XY and Cylindrical about Z),
  - does not support non-intrinsic and intrinsic variables.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If activated in the Solve Setup window, the Relaxation Factor is also available in the Solve Properties window:



Permeability Options for Magnetostatic Solutions

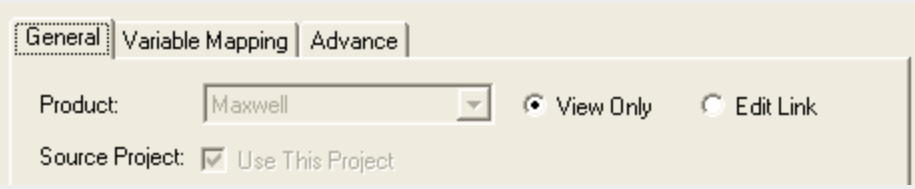
To set permeability options in the **Solve Setup** window:

1. Click **Maxwell** > **Analysis Setup** > **Add Solution Setup** to open the **Solve Setup** window.
2. Click the **Solver** tab.
3. In the **Advanced Option** section, select **Use precomputed permeability data**, which uses a previously solved mu (apparent permeability) distribution from a linked source design. Selecting this option enables the **Setup Link** button, and the **Setup Link** dialog opens, allowing you to specify the source project, design, and solution containing the desired information. The mesh between the two linked designs is the same. You can use the apparent permeability that has been frozen from a previously solved source design.

   By default, the **Setup Link** dialog box opens on the **General** tab with only the **This project** radio button selected.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: If a link had previously been set up, the **Setup Link** dialog box opens in **View Only** mode with all settings disabled.



To enable settings, you can select **Edit Link**.

4. To select a source project:
   - To use the current project as the source, check **Use This Project**. This disables the **Save source path relative to:** radio buttons and the ellipsis **[...]** button and its associated text field.
   - To specify a source project file other than the current project click the ellipsis **[...]** button to open a file browser window. When you have selected the project, click the **Open** button to accept the project file for the setup. You can use the check box to **Open as read only**.

   Use the radio buttons to specify whether to save the source path relative to **The project directory of selected product** or **This project**.

5. When you select a source project file, the **Source Design** and the **Source Solution** fields are filled in with default values, and their drop down menus contain any available designs and solutions. You can use the drop down menus to choose from the available designs and solutions.

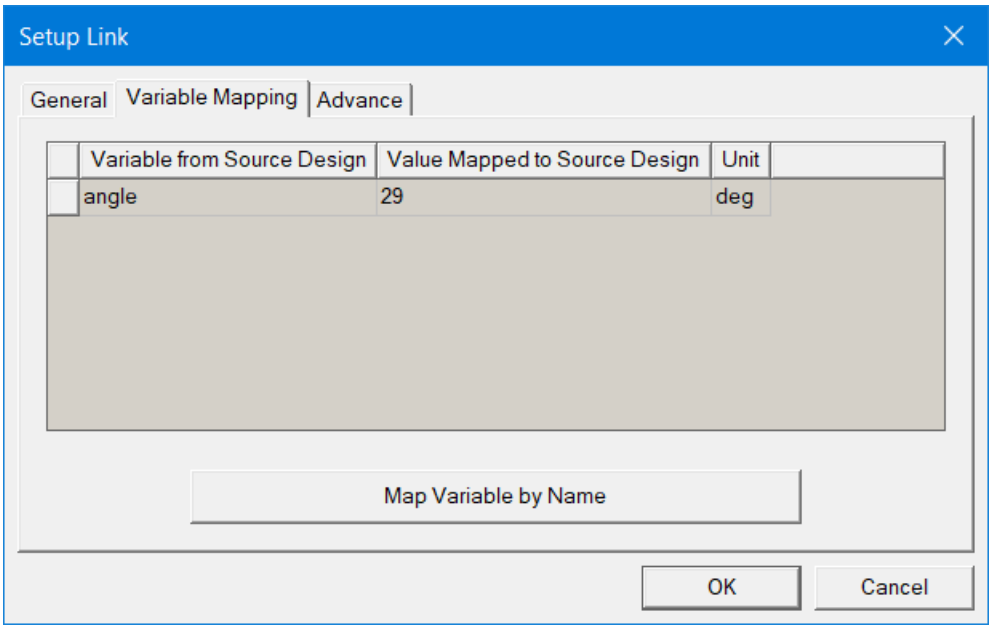
   The “Default” solution is the product-dependent solution of the first Setup – the setup listed first in the source design's project tree (alphanumerical order). A product-specific solution of this setup becomes the default solution. In most designs, it is LastAdaptive.

6. Use the check box to specify whether to **Simulate source design as needed**.
7. Use the check box to specify whether to **Preserve source design solution**. Note that in the extractor mode, the source project will be saved upon exit. Extractor mode means that the software is opened during the link solely for the purpose of solving.
8. The **Variable Mapping** tab lets you view any variables contained in the selected Project.

   When there are variables in the source design, you can choose to “map” these variables to constant values, expressions or variables in the target designs. Variable mapping becomes more important when the datalink type requires source and target design to be geometrically identical and source design is geometrically parameterized.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For linked designs with variables of the same name, you can click **Map Variables by Name** to automatically map same named variables.



9. Optionally, you can override the solved mu with a user-specified control program, which you can specify on the **Advance** tab of the **Setup Link** dialog box. Additionally, because the nonlinear characteristics of many magnetic materials are too complex to be defined through the definition of a BH-Curve, an advanced feature allows the permeability characteristic of the target design to be obtained by mathematical manipulation of the permeability from the source design via a user-defined control program. To use this feature:
    a. Check **Enable user program**.
    b. Specify the **Absolute path of program** to the user control program.
    c. Specify any **Additional arguments** needed by the user control program.

    > **Note:** For detailed information on using this feature, refer to Using the Advanced Permeability Option.

10. Click **OK** to accept the setup and close the Setup Link dialog and return to the Solve Setup window.
11. After setting up the link, you can select the **Including magnets** check box if magnets are included in the distribution.
12. When you are finished specifying solution settings, click **OK** to close the **Solve Setup** dialog box and finalize the solution setup.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** The usual way to create the linked designs is by copying and pasting an existing design from the project of your choice once the design is complete. Occasionally an existing design (possibly from a different project) can be used as a source.
>
> The data link automatically attempts to solve the linked (source) design if necessary (if no solution exists), then automatically copies the mesh and necessary data (calculated distribution of permeability), and finally performs the requested calculation in the target design. This data link allows you to calculate the magnetic field to be performed in a design with, for example, modified sources while keeping the permeability in a "frozen" state calculated in a different design (the same geometry and mesh but with different sources).

Using the Advanced Permeability Option

The nonlinear characteristics of many magnetic materials are too complex to be defined through the definition of a B-H Curve. This feature enhances the existing permeability link by enabling the permeability characteristic of the target design to be obtained by mathematical manipulation of the permeability from the source design. The nonlinear “apparent” permeability from the source design is frozen (linearized) and used by the target design. Because the permeability in the target design is linear, the apparent and the incremental permeabilities will be identical and the apparent and incremental inductances will also match.

This manipulation is done via a user-supplied control program. A user-specified executable is launched at the beginning of target design simulation and the linked permeability data (i.e., the “frozen” mu) from the source design is overridden by the values calculated by the user control program. The target design solver is then launched and uses the overridden value in its calculation.

The user control executable program is launched with two file names as arguments:

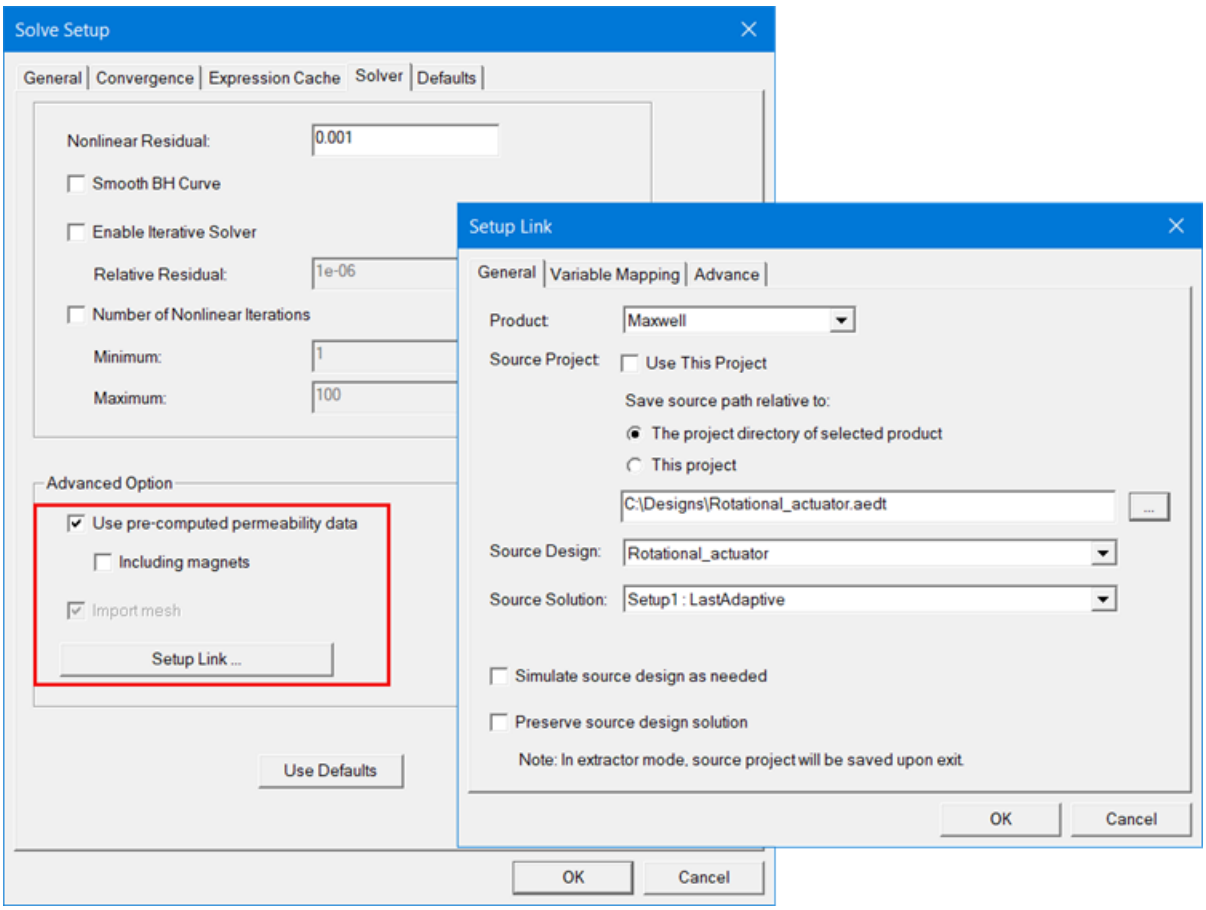
1. **Per mesh element permeability value.** This is the solved mu (*i.e.*, the apparent permeability not the incremental permeability) from the source design. Note that the format of this file is different between 2D and 3D designs.
2. **Per mesh element information.** Information includes the centroid location (x, y, z), Bx, By, Bz at the centroid and Hx, Hy, Hz at the centroid. The format of this file is the same between 2D and 3D designs.

> **Note:** Refer to Advanced Permeability Option File Formats for information on these file formats.

Considerations for Using this Feature

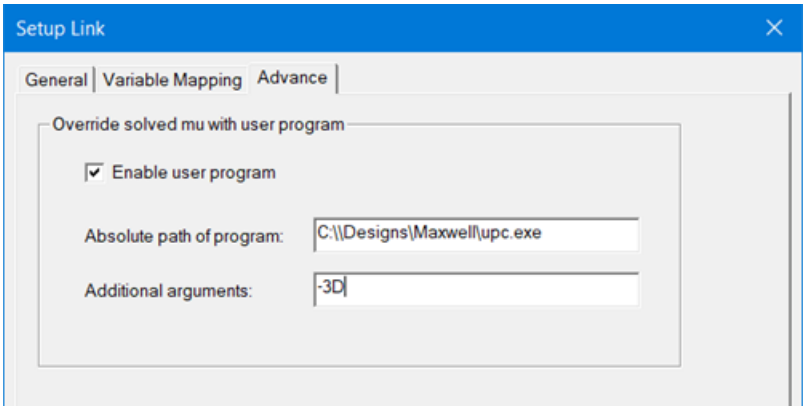
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. To set up the advanced user control program option for permeability, select **Setup Link** on the Solver tab to open the Setup Link dialog box.



2. Specify the **Source Design** on the **General** tab.
3. On the **Advance** tab, select the check box **Enable user program** and provide the absolute path to the executable, and provide any **Additional arguments** needed by the user program.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Simulate the target design.

   Simulation of the source design will be automatically started if it is not already simulated. The user-supplied executable is launched. Additional information is logged in **Setup** > **Profile** regarding the launch of user executable.

5. Simulation of the target design continues after user executable exits.

Note the following:

- Field post processing of target design will use the user-overridden mu.
- Editing boundary/excitation setup in the target design invalidates the current solution in target design.
- Linked data remains valid when boundary/excitation is changed. On next simulation of the target design, the prior user-overridden permeability will be used.
- Edit of the link setup (this includes the path to the user control program), or “removing” the user control program will invalidate the current solution and also the cached linked data. On the next simulation of the target design, the “frozen” permeability will be extracted again from the source design. If the link setup has opted to enable the user control program, the currently designated user program will be launched. If the link setup has opted *not* to override the linked data, the extracted mu from the source design will be used as is.
- **Clear Linked Data** in the target design invalidates the current solution and cached linked data in the target design.
- On the next simulation of the target design, the frozen permeability will be extracted again from the source design. The user executable will be launched to allow the permeability to be overridden.
- Editing the source design has no effect on the existing solution in the target design. Users need to **Clear Linked Data** in target design if a new simulation in the target design is desired.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- After the simulation of the target design, the user-overridden mu becomes part of the solution of the target design and this target design can then be used as the source of another design.

Advanced Permeability Option File Formats

The following are the file formats for this feature:

3D Per Mesh Element Permeability Value

This is the solved mu (*i.e.*, the apparent permeability not the incremental permeability) from the 3D source design.

```
SolidID   6   14
h  230  79578.5
h  231  79578.5
h  232  79578.5
h  233  79578.5
h  234  79578.5
h  235  79578.5
h  236  79578.5
h  237  79578.5
h  238  79578.5
h  239  79578.5
h  240  79578.5
h  241  79578.5
h  242  79578.5
h  243  79578.5
E_DATA
```

The average permeability calculated by Maxwell solver. The average is calculated from the value at each of the nodes of the mesh element.

In the above example, SolidID, h, and E_DATA are keywords. “6” is the object ID. “14” is the number of elements in the object with ID “6”. “230”, “231”, etc. are element IDs.

Note that there could be multiple "blocks" of the above data if there are multiple nonlinear objects in the design, as illustrated below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
SolidID  6   17
h  351  79578.5
h  352  79578.5
.
.
.
h  366  79578.5
h  367  79578.5
SolidID  93   12
h  1954  79578.5
h  1955  79578.5

.
.
.
h  1963  79578.5
h  1964  79578.5
h  1965  79578.5
E_DATA
```

2D Per Mesh Element Permeability Value

This is the solved mu from the 2D source design. The format is

<number of total elements>

<objectID> <elementID> <X/R center position of element> <Y/Z center position of element> <mu>

For example, the following solved mu indicates that there is ONE object that has TWO mesh elements. The first element has a mu value of 2.7482 and the second element has a mu of 2.76553.

```
2
76 1   0.000866667  6.66667e-005 2.7482
76 15  0.000733333 -6.66667e-005 2.76553
end
```

The following solved mu indicates a total of 16 mesh elements that belong to two objects.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
16
64 24   0.000733333 -0.000533333 1.90406
64 27   0.000866667 -0.000266667 1.88459
64 28   0.000733333 -0.000266667 1.89191
64 41   0.000666667 -0.000333333 1.88789
64 42   0.000666667 -0.000466667 1.8909
64 53   0.000933333 -0.000333333 1.87607
64 69   0.000833333 -0.0005 1.91924
64 70   0.000966667 -0.0005 1.93687
64 71   0.0009 -0.000433333 1.89957
64 74   0.000933333 -0.000566667 2.0507
64 75   0.000866667 -0.000566667 1.95098
76 20   0.000666667 0 1.94237
76 21   0.000933333 0 1.92467
76 22   0.0008 0.000133333 2.06578
76 25   0.000866667 -0.000133333 1.89876
76 26   0.000733333 -0.000133333 1.90766
end
```

Per Mesh Element Centroid Location

(x, y, z), Bx, Bx, By, Hx, Hy, Hz at the centroid. All values in SI units.

SolidID 6 14

h 230 x y z Bx By Bz Hx Hy Hz

.

.

.

E_DATA

AC Magnetic Solver Settings

The settings available in the AC Magnetic Solve Setup**Solver** tab are determined by the type of design selected (2D, 3D T-Omega, or 3D A-Phi).

For an AC Magnetic 2D Solution:

1. **Adaptive Frequency** – This value specifies the nominal problem frequency, which will be used in the adaptive mesh process. The units are specified by the user. The value can be specified using variables and expressions.
2. **Nonlinear Residual** – This value specifies how close each solution must come to satisfying the equations that are used to compute the electric field.
3. **Smooth BH Curve** – This option allows the solver to smoothen the list of points to make a refined BH curve for better solving. If this option is not selected, the solver just uses the discrete points.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

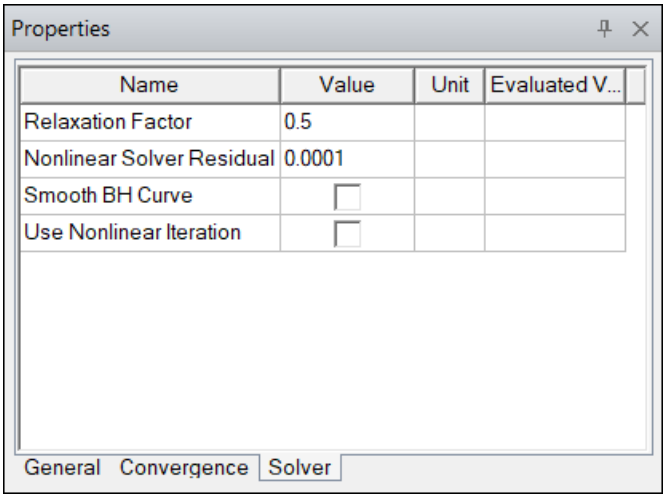
4. **Number of Nonlinear Iterations** – This provides an option for you to set both a **Minimum** iteration number to avoid non-converged solution, and a **Maximum** iteration number to avoid taking too much computation time. These values must be integers greater than 0. By default the values for **Minimum** and **Maximum** are 1 and 100, respectively. Maximum number should be greater than the Minimum number.
5. **Thermal Two-Way Coupling\Relaxation** – This setting can be used to improve two-way coupling convergence by applying the relaxation factor to the temperature field. When the thermal modifier takes on a nonlinear form, the convergence of the solution field in Maxwell's equations may fail during two-way coupling iterations. With nonlinear iterations, using an under-relaxation factor can mitigate the risk of overshooting in the property updating process.

   The range of the relaxation factor is from 0 to 1. In general, a smaller relaxation value helps convergence, but may increase the total number of iterations. The default relaxation factor is 1.0.

   The **Thermal Two-Way Coupling\Relaxation** setting is applicable only when **Enable Feedback** is checked in Temperature of Objects window, which invokes two-way coupling instead of one-way coupling. The relaxation setting is

   - applicable to all Maxwell 2D designs,
   - applicable to both geometry modes (Cartesian XY and Cylindrical about Z),
   - does not support non-intrinsic and intrinsic variables.

   If activated in the Solve Setup window, the Relaxation Factor is also available in the Solve Properties window:



6. If desired, you can Import Mesh.

For an AC Magnetic 3D T-Omega Solution:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

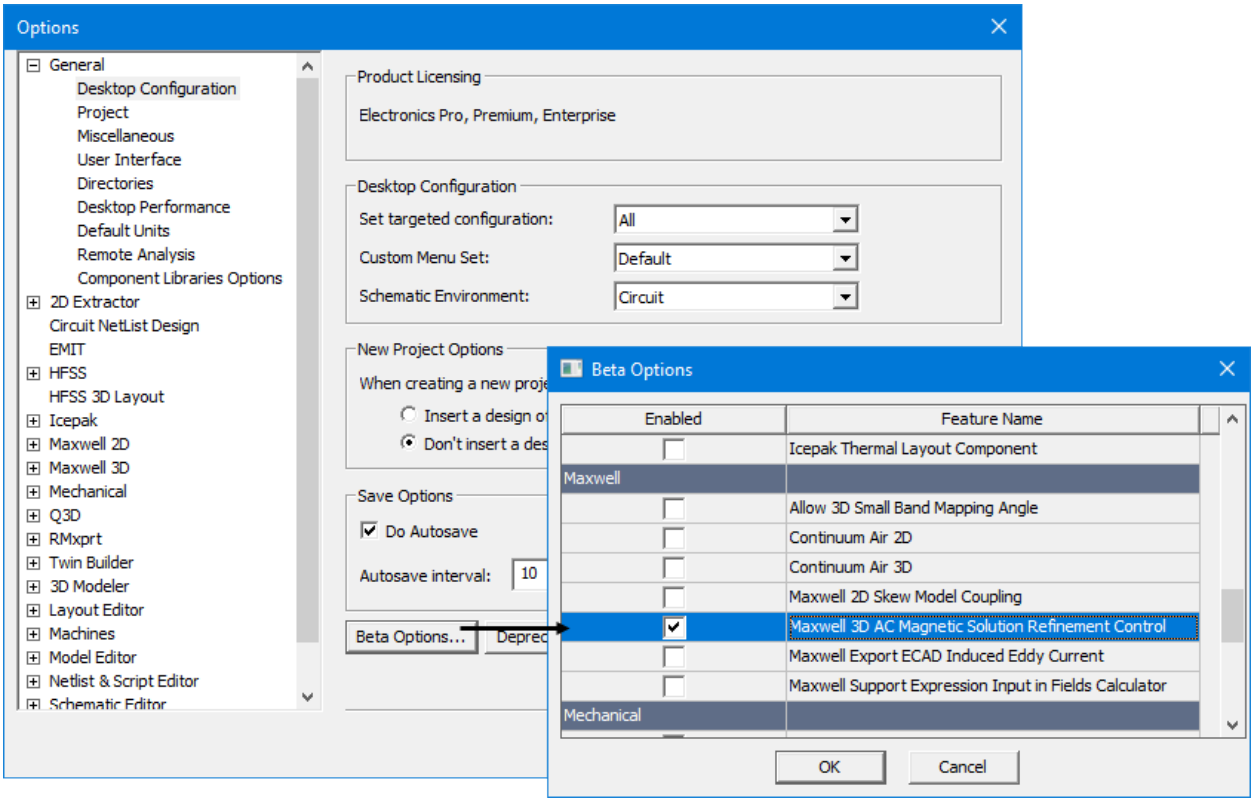
1. **Adaptive Frequency** – This value specifies the nominal problem frequency, which will be used in the adaptive mesh process. The units are specified by the user. The value can be specified using variables and expressions.
2. **Nonlinear Residual** – This value specifies how close each solution must come to satisfying the equations that are used to compute the electric field.
3. **Smooth BH Curve** – This option allows the solver to smoothen the list of points to make a refined BH curve for better solving. If this option is not selected, the solver just uses the discrete points.
4. Enable an Iterative Solver – This option enables the iterative solver, rather than the direct solver. For more information, see Iterative Matrix Solver.
5. **Use higher order shape functions –** This optional setting will increase the order of electric vector potential to 2 from the default order of 1. Enabling the higher order option gains better accuracy for eddy current regions.

   > **Note:** Higher-order shape functions are not supported for a project with windings. For this case, the solver falls back to lower-order shape functions.

6. **Number of Nonlinear Iterations** – This provides an option for you to set both a **Minimum** iteration number to avoid non-converged solution, and a **Maximum** iteration number to avoid taking too much computation time. These values must be integers greater than 0. By default the values for **Minimum** and **Maximum** are 1 and 100, respectively. Maximum number should be greater than the Minimum number.
7. **Loss Adaptive Control** – For a typical analysis in Ansys Maxwell, the energy error generated by an FEM residual is used as the mesh adaptive refinement criterion, but for modeling the skin effects of the eddy current problem, the eddy current region may not be sufficiently refined. To better model the skin effects, it is more efficient to use a combination of energy error and the power loss of the eddy current effects as a mixed criterion for mesh adaptive refinement. Better accuracy of skin effects modeling gives more accurate power loss. The Loss Adaptive Control slider bar controls the weight of energy error and the power loss. The value of 0 means that the mesh adaptive refinement is purely based on energy error while any nonzero value means that the mesh adaptive refinement is based on the mixed criterion. A value of 1 means that the mesh adaptive refinement is purely based power loss; although setting Loss Adaptive Control to 1 can give the most accurate power loss calculation, it may result in less accurate energy calculation and potentially need more adaptive passes. The default value for Loss Adaptive Control is 0.5, and projects created in earlier versions of Ansys Maxwell will use this value.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: The Loss Adaptive Control setting is only accessible if it has been activated from **Tools** > **Options** > **General Options** > **Desktop Configuration** > **Beta Options** dialog box. Select the **Maxwell 3D AC Magnetic Solution Refinement Control** feature.



8. **Use precomputed permeability data** – Optional setting that allows the use of the permeability data calculated in a source 3D magnetostatic design. Note that this option is not available if **Include DC Fields** is selected in the Solver Type window because permeability data will automatically included in this type of simulation; refer to Including DC Fields in a 3D AC Magnetic T-Omega Solution for details.
9. If desired, you can Import Mesh. This setting is not available if **Include DC Fields** is selected.

Note: The AC Magnetic 3D Omega solver can export RL matrices; see Impedance Matrix for an AC Magnetic Solution for more details.

For an AC Magnetic 3D A-Phi Solution:

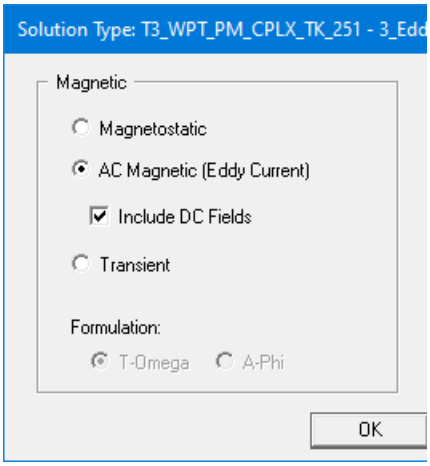
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. **Adaptive Frequency** – This value specifies the nominal problem frequency, which will be used in the adaptive mesh process. The units are specified by the user. The value can be specified using variables and expressions.
2. **Nonlinear Residual** – This value specifies how close each solution must come to satisfying the equations that are used to compute the electric field.
3. **Number of Nonlinear Iterations** – This provides an option for you to set both a **Minimum** iteration number to avoid non-converged solution, and a **Maximum** iteration number to avoid taking too much computation time. These values must be integers greater than 0. By default the values for **Minimum** and **Maximum** are 1 and 100, respectively. Maximum number should be greater than the Minimum number.
4. If desired, you can Import Mesh.

> **Note:** The AC Magnetic 3D A-Phi solver can export RL and GC matrices; see 3D AC Magnetic A-Phi Parameter Extraction for more details.

Including DC Fields in a 3D AC Magnetic T-Omega Solution

For projects with materials whose permeability is nonlinear, permeability can be included in the 3D AC Magnetic T-Omega calculations by checking the **Include DC Fields** option in the Solution Type dialog box:



When this option is selected, the solver runs a Magnetostatic simulation (DC) and then an AC Magnetic simulation. The AC Magnetic design operating point on the nonlinear B-H curve is calculated based on the DC fields, and permeability is determined by slope of B-H curve at the operating point.

After the simulation is complete, the AC field results and DC field results, such as the B and H fields, are available for plotting.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This feature can be used for small-signal analysis as the operating point in the B-H curve does not change significantly from the magnetic solution to the harmonic solution.

For the AC Magnetic\Include DC Fields analysis, note the following:

- DC and AC simulations can use independent adaptive meshes.
- Mesh operations are applied to both the AC and DC simulations except **Skin Depth**, which is not applicable for DC.
- Boundaries are applied to both AC and DC simulation, except **Impedance** and **Radiation**, which are not supported for DC.
- Settings in the tabs of the Solve Setup window only apply to AC simulation; the DC simulation is only configured from the Solve Setup's **DC** tab.
- Any matrix, force, and torque parameters set up will only apply to AC simulation.
- The Inductance type in the Magnetostatic simulation is **Incremental**, *i.e.*, defined as the ratio of delta flux to delta current at a certain operating condition. See Inductance Type Settings for 3D Magnetostatic Solutions for more details.
- If an object's material composition is a Litz wire, its properties are only applied to AC simulation. DC simulation does not support Litz wire and will treat it as solid composition.

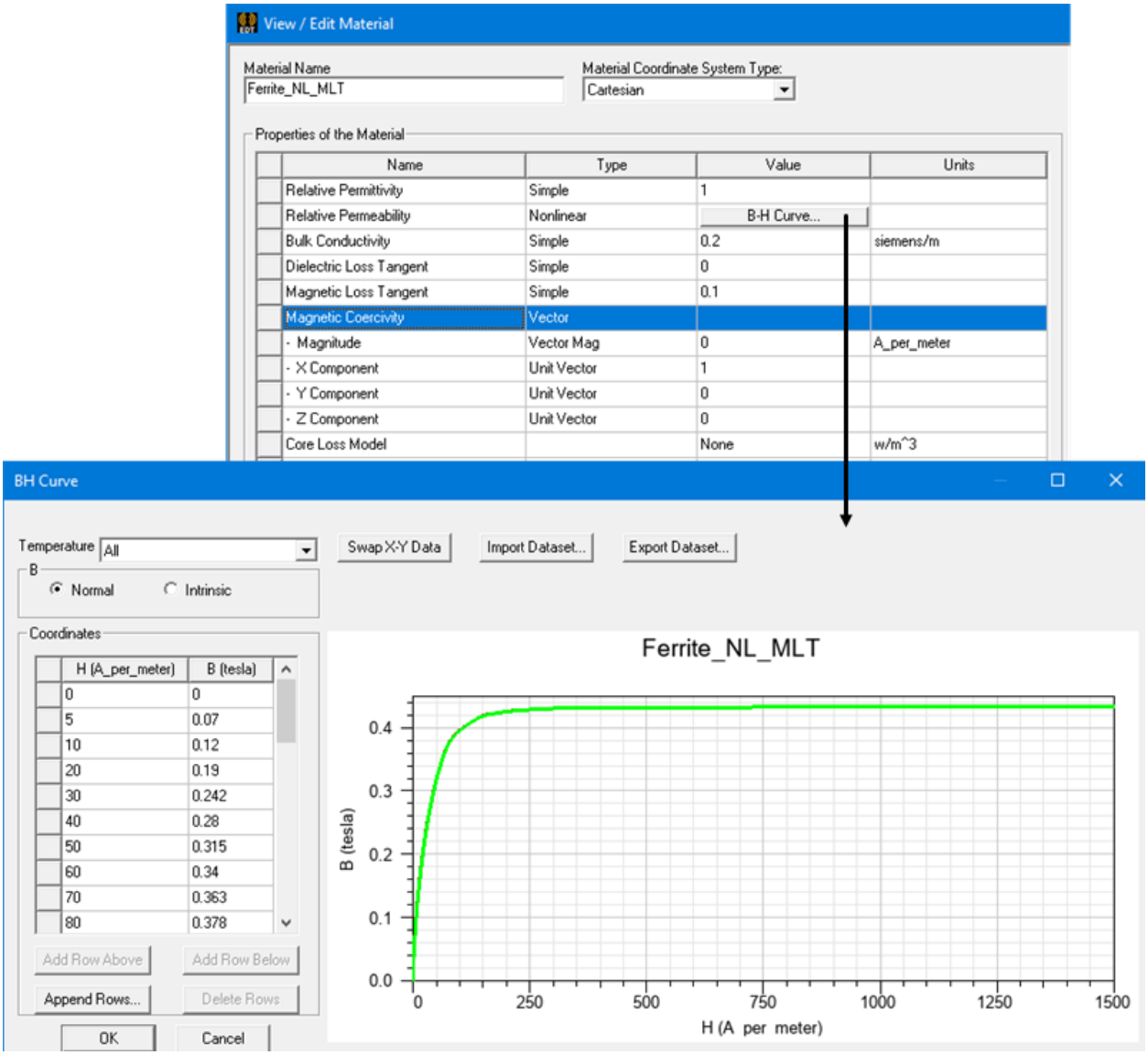
> **Note:** In the previous release, this type of simulation required two models: a source 3D Magnetostatic design and a 3D AC Magnetic target design. With the 25r1 release, only one design is needed for both DC and AC simulations.

To set up this type of analysis:

1. Create a Maxwell 3D design.
2. In Solution Type dialog box, select **AC Magnetic**, and check **Include DC Fields**.
3. For the permanent magnet material, define the **Relative Permeability** with a BH-curve, and define **Magnetic Coercivity** parameters. Below is an example of a soft magnetic and a nonlinear permanent magnet.

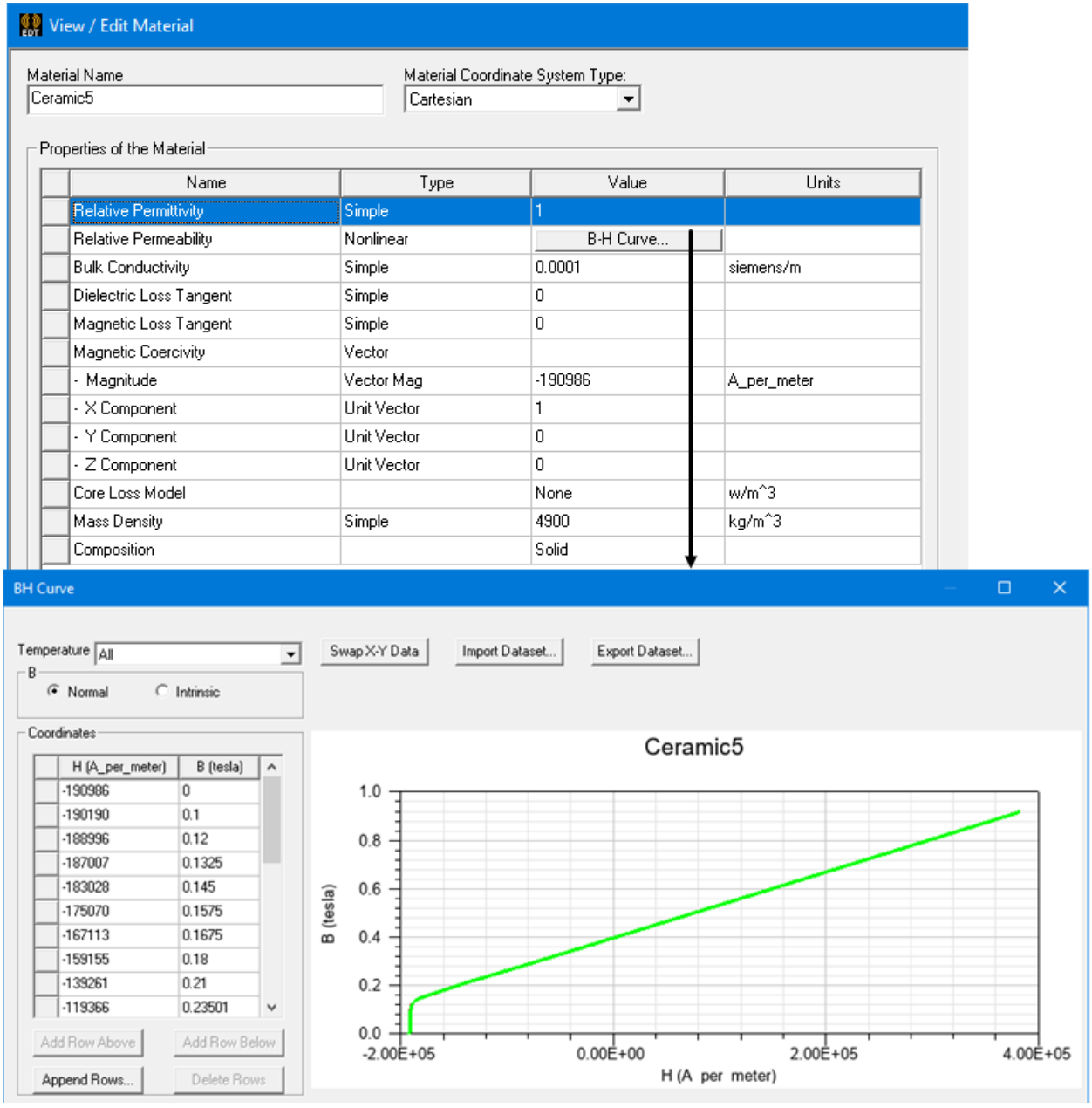
   Relative Permeability BH Curve for a Soft Magnet:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



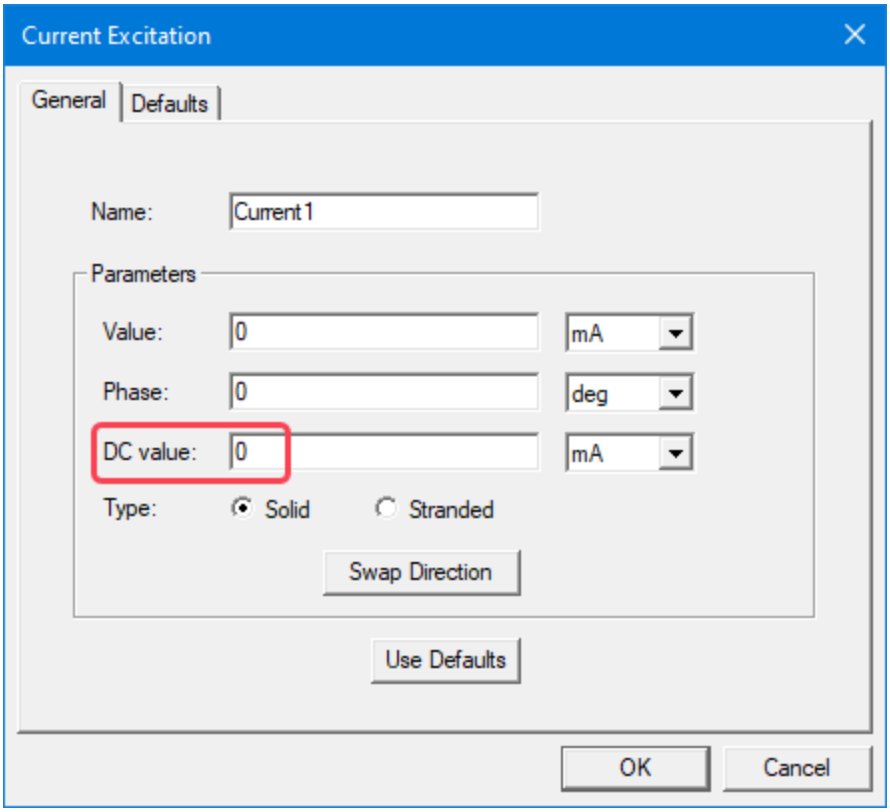
Relative Permeability BH Curve for a Nonlinear Permanent Magnet:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



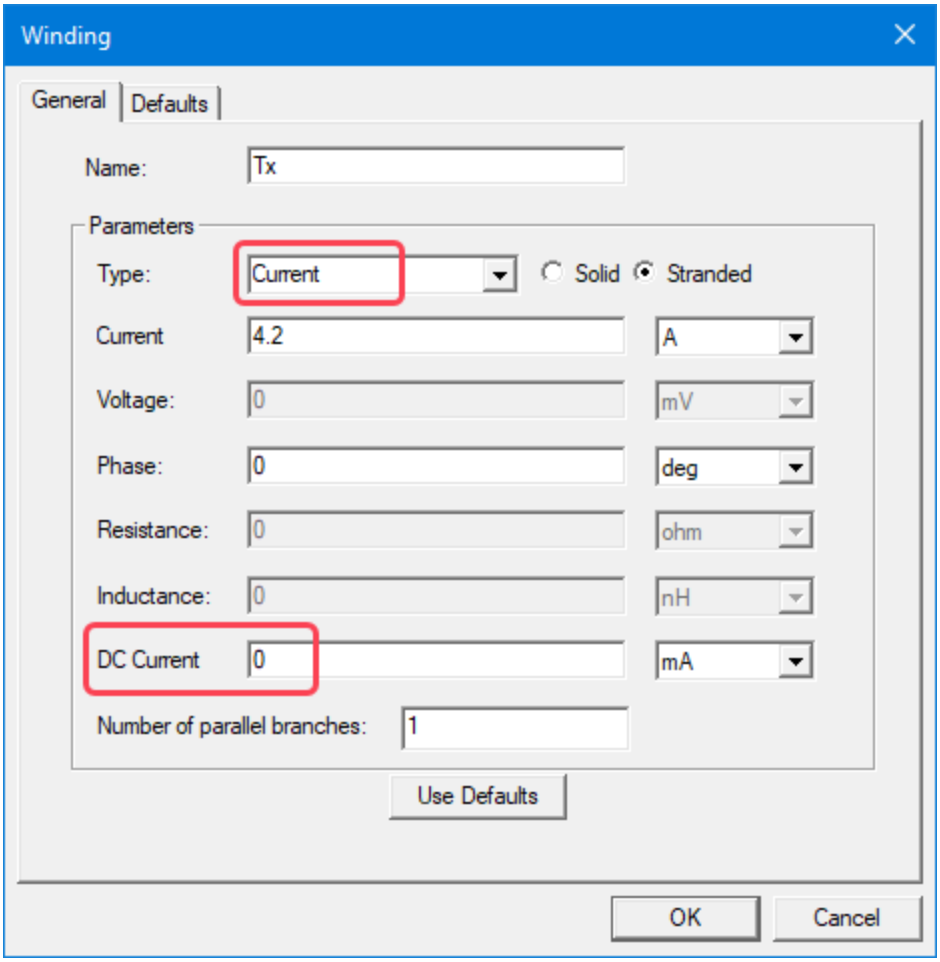
4. Define a DC input. The **Current** and **Winding** excitations will include a DC input field:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

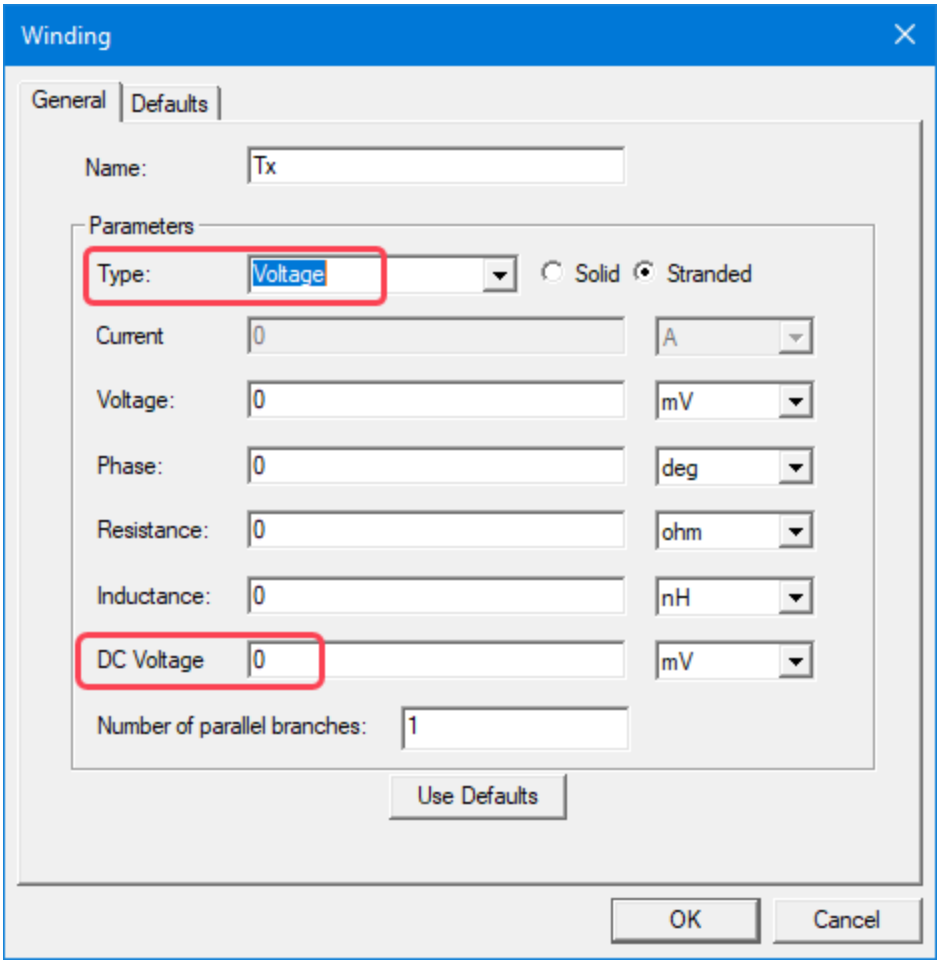


The DC input for a Winding excitation depends on the winding Type:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

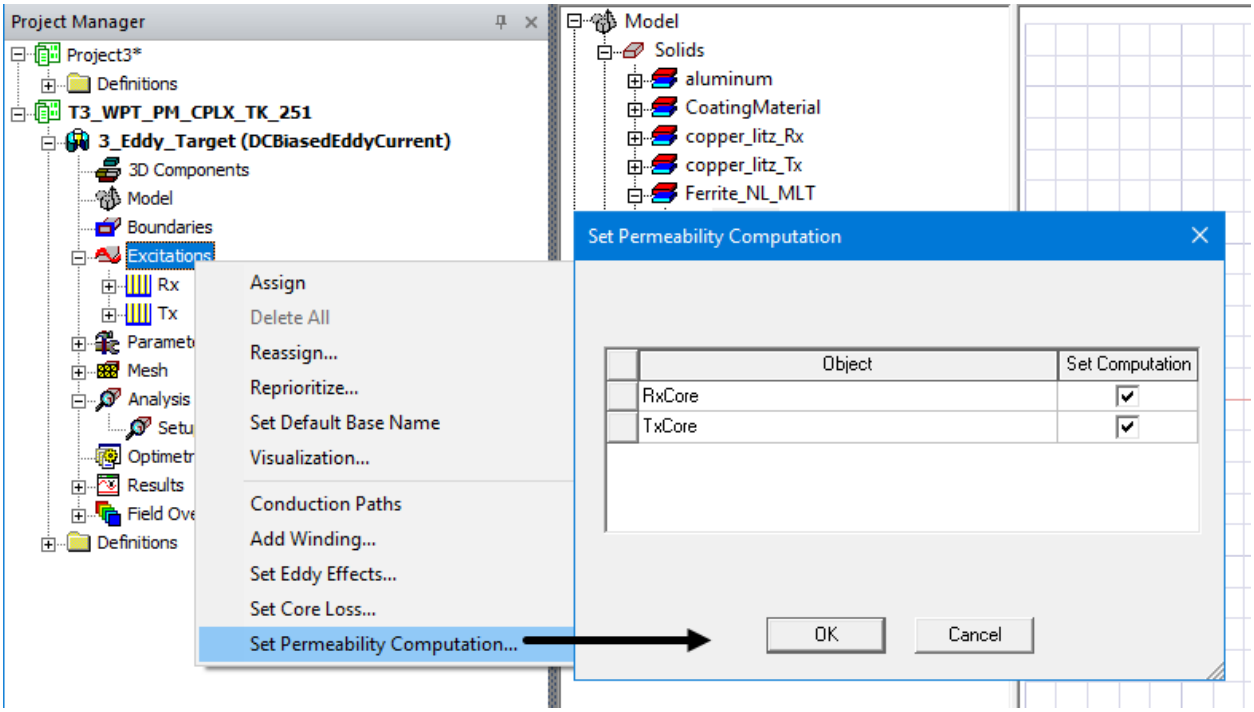


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



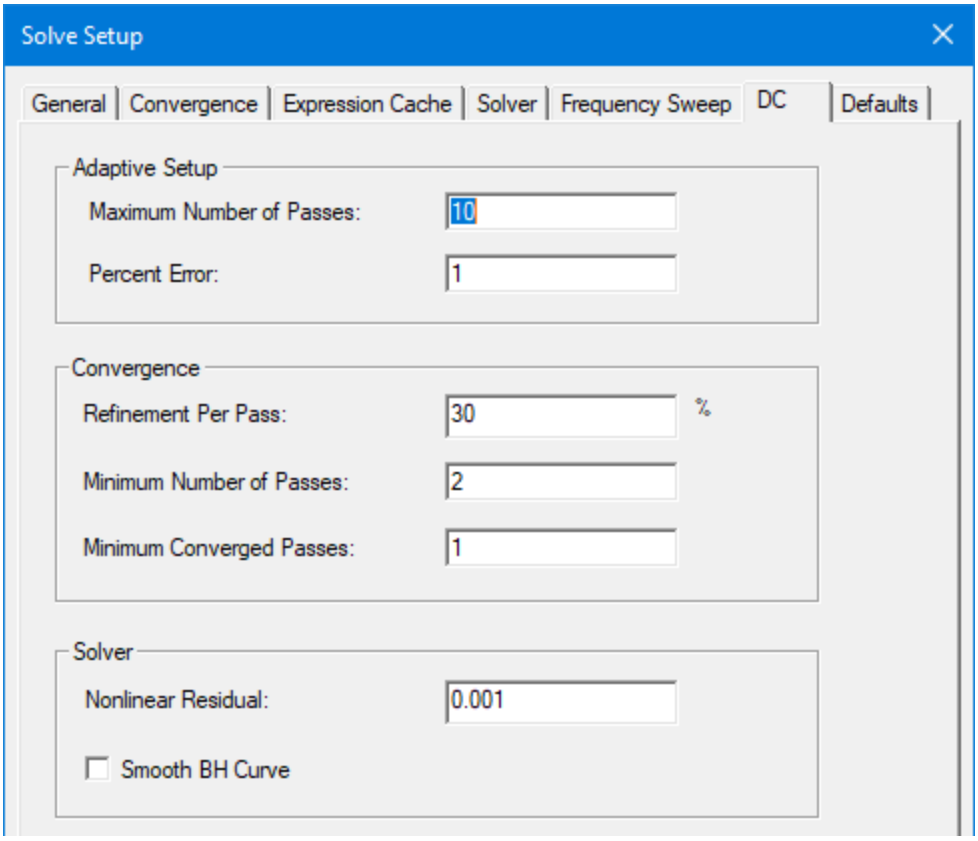
5. Right-click on **Excitations** > **Set Permeability Computation**. By default, all objects with nonlinear B-H curve material, including soft magnets and nonlinear permanent magnets, are selected. Click **OK** to add to the simulation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



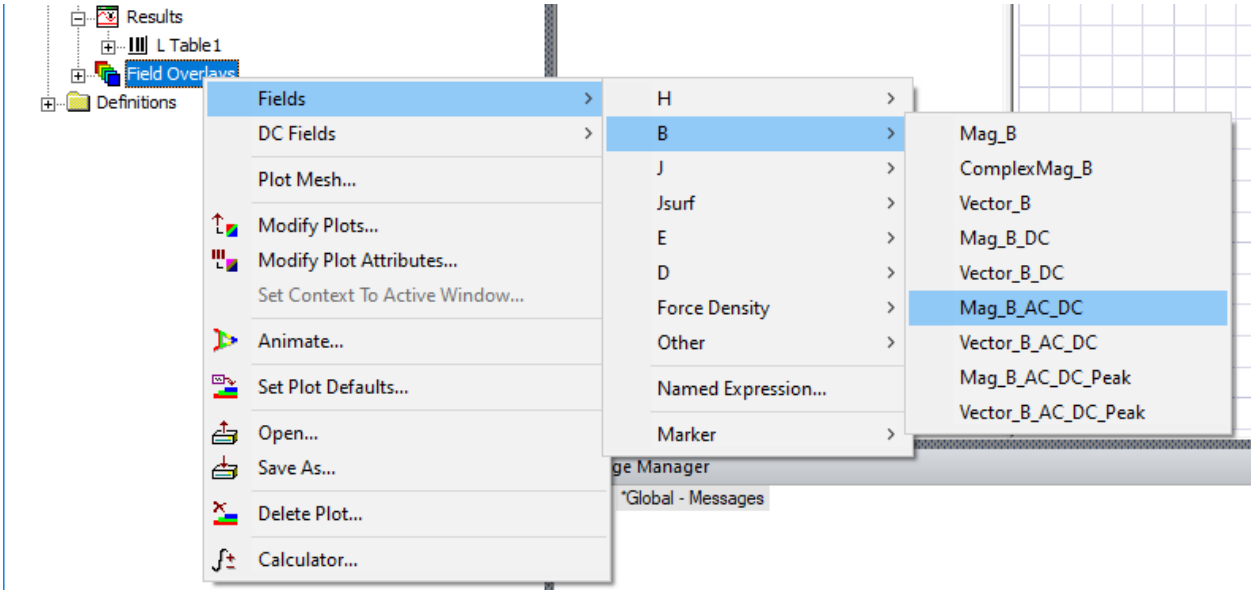
6. In the Solve Setup window, make sure to configure the **DC** tab:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

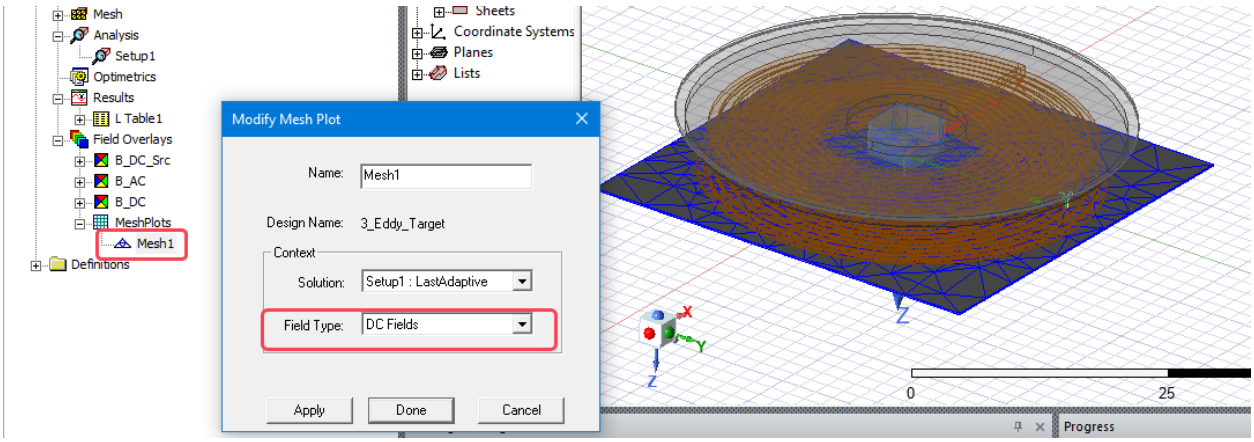


7. Solve the 3D AC Magnetic design.
8. Create B, H, and J field plots from the AC, AC+DC, and DC results.
    a. The Mag_B_DC, Vector_B_DC is the DC field mapped from magnetostatic solution.
    b. The Mag_B_AC_DC, Vector_B_AC_DC is the addition of AC field from eddy-current solution and DC field from magnetostatic solution.
    c. The Mag_B_AC_DC_Peak, Vector_B_AC_DC_Peak is the peak value of AC+DC field among all AC phases.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

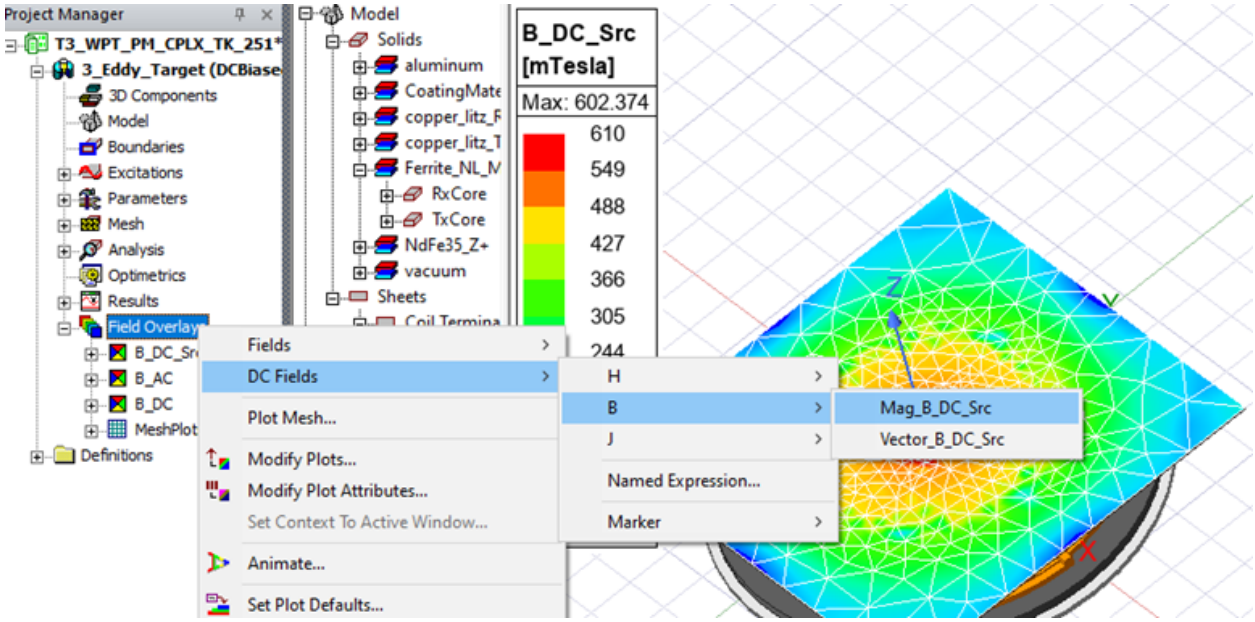


9. Plot the Magnetostatic solution DC mesh:

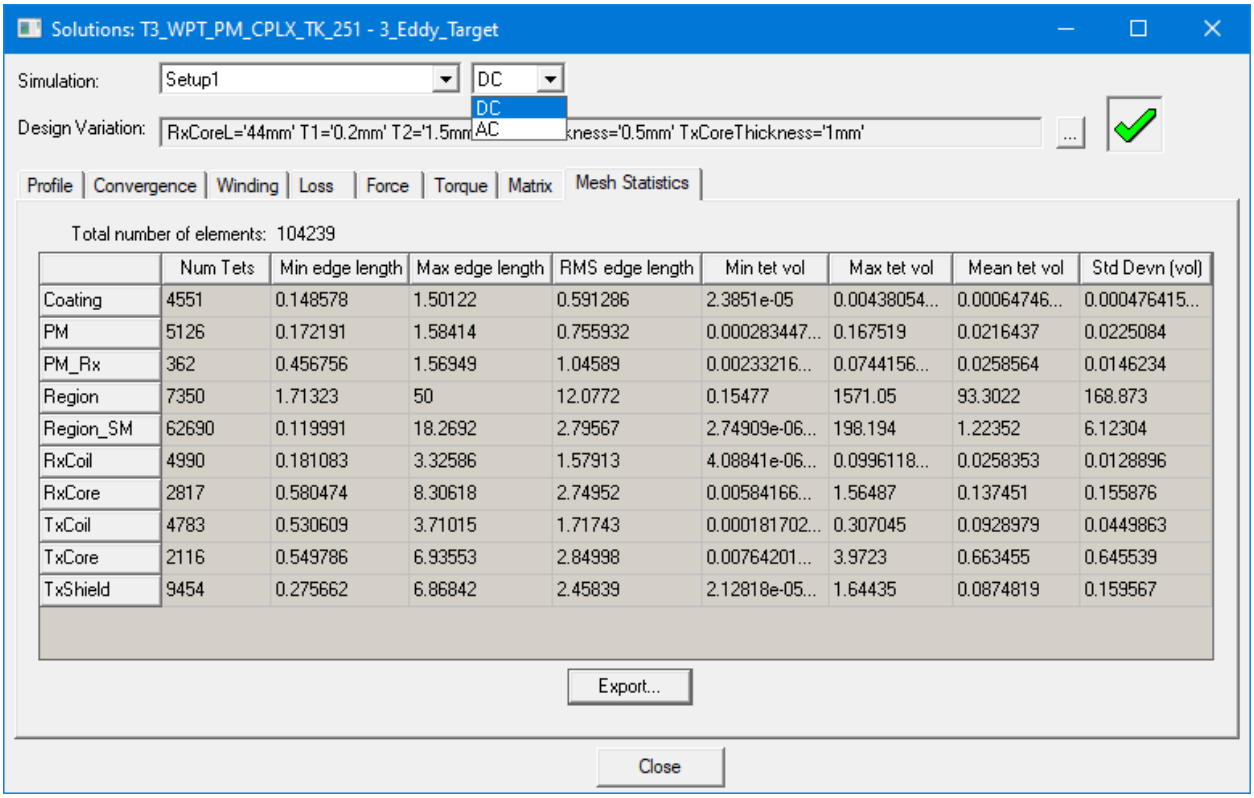


10. Plot the magnetostatic solution DC fields B, H, and J.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



11. Both the AC and DC Solution Data are available in the Solution Data window (right click on **Results** and select **Solution Data**).



	Num Tets	Min edge length	Max edge length	RMS edge length	Min tet vol	Max tet vol	Mean tet vol	Std Devn (vol)
Coating	4551	0.148578	1.50122	0.591286	2.3851e-05	0.00438054...	0.00064746...	0.000476415...
PM	5126	0.172191	1.58414	0.755932	0.000283447...	0.167519	0.0216437	0.0225084
PM_Rx	362	0.456756	1.56949	1.04589	0.00233216...	0.0744156...	0.0258564	0.0146234
Region	7350	1.71323	50	12.0772	0.15477	1571.05	93.3022	168.873
Region_SM	62690	0.119991	18.2692	2.79567	2.74909e-06...	198.194	1.22352	6.12304
RxCoil	4990	0.181083	3.32586	1.57913	4.08841e-06...	0.0996118...	0.0258353	0.0128896
RxCore	2817	0.580474	8.30618	2.74952	0.00584166...	1.56487	0.137451	0.155876
TxCoil	4783	0.530609	3.71015	1.71743	0.000181702...	0.307045	0.0928979	0.0449863
TxCore	2116	0.549786	6.93553	2.84998	0.00764201...	3.9723	0.663455	0.645539
TxShield	9454	0.275662	6.86842	2.45839	2.12818e-05...	1.64435	0.0874819	0.159567

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the **DC** option, only the **Profile**, **Convergence**, and **Mesh Statistics** tabs are applicable.

Electrostatic Solver Settings

For an electrostatic solution you can do the following:

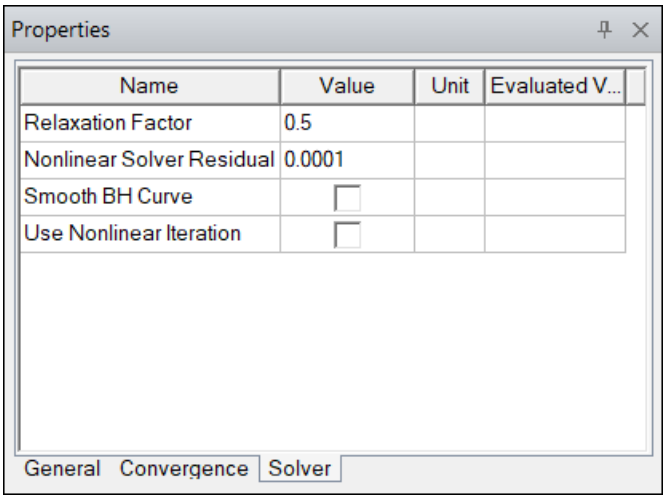
- For 3D designs, you can Enable an Iterative Solver.
- Import Mesh
- For **2D** designs, the **Thermal Two-Way Coupling\Relaxation** setting can be used to improve two-way coupling convergence by applying the relaxation factor to the temperature field. When the thermal modifier takes on a nonlinear form, the convergence of the solution field in Maxwell's equations may fail during two-way coupling iterations. With nonlinear iterations, using an under-relaxation factor can mitigate the risk of overshooting in the property updating process.

  The range of the relaxation factor is from 0 to 1. In general, a smaller relaxation value helps convergence, but may increase the total number of iterations. The default relaxation factor is 1.0.

  The **Thermal Two-Way Coupling\Relaxation** setting is applicable only when **Enable Feedback** is checked in Temperature of Objects window, which invokes two-way coupling instead of one-way coupling. The relaxation setting is

  - applicable to all Maxwell 2D designs,
  - applicable to both geometry modes (Cartesian XY and Cylindrical about Z),
  - does not support non-intrinsic and intrinsic variables.

  If activated in the Solve Setup window, the Relaxation Factor is also available in the Solve Properties window:



AC Conduction Solver Settings

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. For 2D and 3D designs, enter a frequency value in the **Adaptive Frequency** text box, and select the desired units. Use of variables and expressions are also supported.
2. For 3D designs, enter a value in the **Nonlinear Residual** text box. This value specifies how close each solution must come to satisfying the equations that are used to compute the electric field.
3. If desired, you can Import Mesh.
4. For 2D and 3D designs, selecting the **Number of Nonlinear** Iterations check box allows you to set both a **Minimum** iteration number to avoid non-converged solution, and a **Maximum** iteration number to avoid taking too much computation time. These values must be integers greater than 1. The default values for **Minimum** and **Maximum** are 2 and 50, respectively. The **Maximum** number should be greater than the **Minimum** number.
5. For **2D** designs, the **Thermal Two-Way Coupling\Relaxation** setting can be used to improve two-way coupling convergence by applying the relaxation factor to the temperature field. When the thermal modifier takes on a nonlinear form, the convergence of the solution field in Maxwell's equations may fail during two-way coupling iterations. With nonlinear iterations, using an under-relaxation factor can mitigate the risk of overshooting in the property updating process.

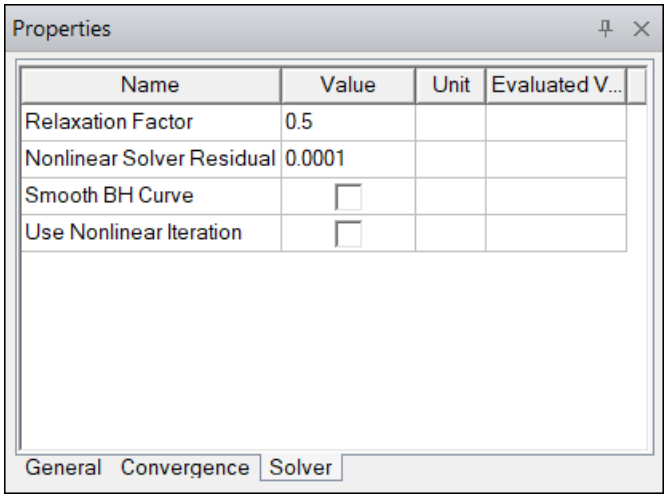
   The range of the relaxation factor is from 0 to 1. In general, a smaller relaxation value helps convergence, but may increase the total number of iterations. The default relaxation factor is 1.0.

   The **Thermal Two-Way Coupling\Relaxation** setting is applicable only when **Enable Feedback** is checked in Temperature of Objects window, which invokes two-way coupling instead of one-way coupling. The relaxation setting is

   - applicable to all Maxwell 2D designs,
   - applicable to both geometry modes (Cartesian XY and Cylindrical about Z),
   - does not support non-intrinsic and intrinsic variables.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If activated in the Solve Setup window, the Relaxation Factor is also available in the Solve Properties window:



DC Conduction Solver Settings

For a DC Conduction solution:

1. If desired, you can Import Mesh.
2. For 3D designs, optionally Enable an Iterative Solver.
3. For **2D** designs, the **Thermal Two-Way Coupling\Relaxation** setting can be used to improve two-way coupling convergence by applying the relaxation factor to the temperature field. When the thermal modifier takes on a nonlinear form, the convergence of the solution field in Maxwell's equations may fail during two-way coupling iterations. With nonlinear iterations, using an under-relaxation factor can mitigate the risk of overshooting in the property updating process.

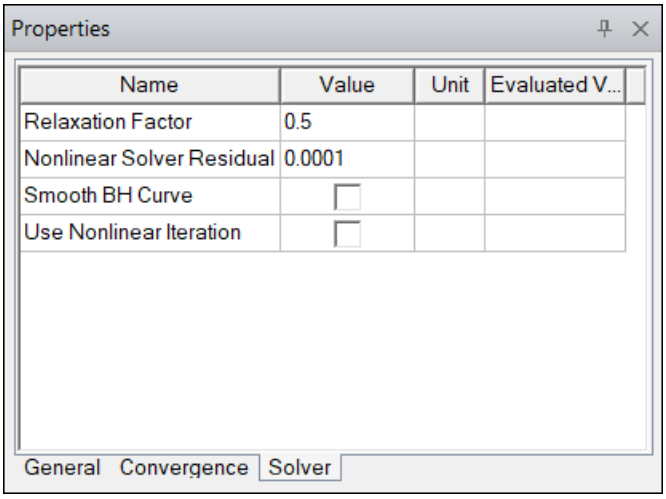
   The range of the relaxation factor is from 0 to 1. In general, a smaller relaxation value helps convergence, but may increase the total number of iterations. The default relaxation factor is 1.0.

   The **Thermal Two-Way Coupling\Relaxation** setting is applicable only when **Enable Feedback** is checked in Temperature of Objects window, which invokes two-way coupling instead of one-way coupling. The relaxation setting is

   - applicable to all Maxwell 2D designs,
   - applicable to both geometry modes (Cartesian XY and Cylindrical about Z),
   - does not support non-intrinsic and intrinsic variables.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If activated in the Solve Setup window, the Relaxation Factor is also available in the Solve Properties window:



Import Mesh for Non-Transient Solutions

The **Import mesh** check box can be selected in the “target” design if the mesh for the same geometry is to be used in the analysis. This setup is designed to be generally applicable to all solutions.

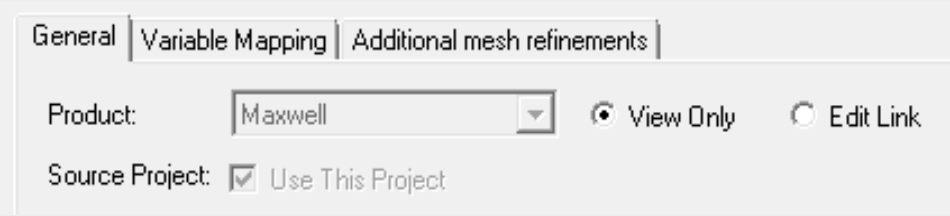
If the **Import mesh** feature is selected, the **Setup Link** button becomes active, the **Setup Link** window automatically appears, and the mesh will be imported from the source design to the target design.

> **Note:** The usual way to create geometrically equivalent designs is by copying all geometry objects from one design and then using **Import From Clipboard** in the other design.

1. By default, the **Setup Link** window opens on the **General** tab with only the **This project** radio button selected.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: If a link had previously been set up, the **Setup Link** window opens in **View Only** mode with all settings disabled.



To enable settings, you can select **Edit Link**.

2. To select a source project, do one of the following:
   - To use the current project as the source, check **Use This Project**. This disables the **Save source path to:** radio buttons and the ellipsis **[...]** button and its associated text field.
   - To specify a source project file other than the current project click the ellipsis **[...]** button to open a file browser window. When you have selected the project, click the **Open** button to accept the project file for the setup. You can use the check box to **Open as read only**.

     Use the radio buttons to specify whether to save the source path relative to **The project directory of the selected project** or **This project**.
3. When you select a source project file, the **Source Design** and the **Source Solution** fields are filled in with default values, and their drop-down menus contain any available designs and solutions. You can use the drop down menus to choose from the available designs and solutions.

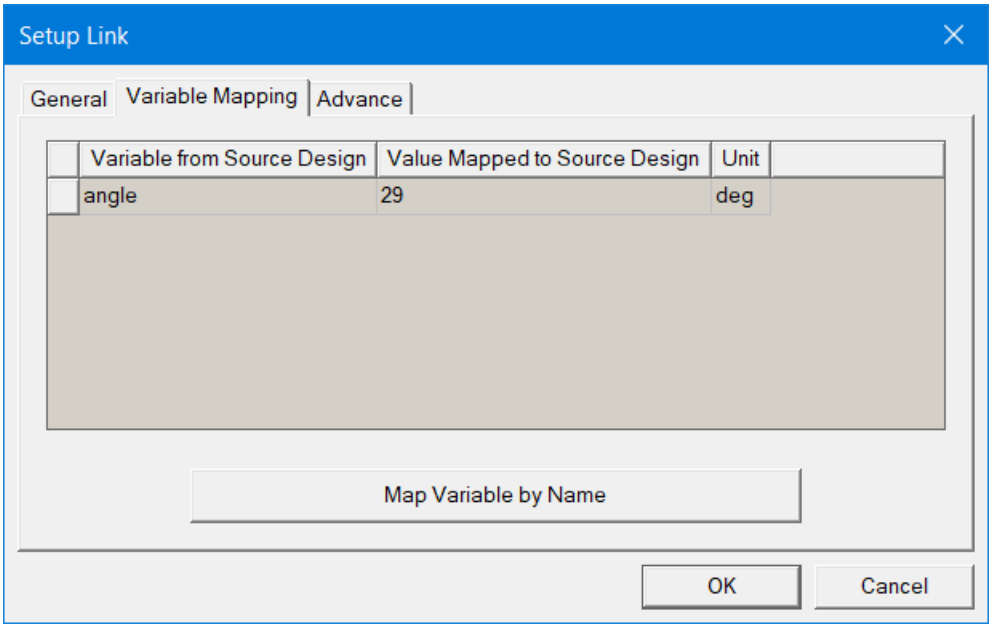
   The “Default” solution is the product-dependent solution of the first Setup. That is the setup listed first in the source design's project tree (alphanumerical order). A product-specific solution of this setup becomes the default solution. In most products, it is “LastAdaptive.” In a Transient solution type, it is “Transient.”
4. Use the check box to specify whether to **Simulate source design as needed**.
5. Use the check box to specify whether to **Preserve the source design solution**. Note that in the extractor mode, the source project will be saved upon exit. Extractor mode means that the software is opened during the link solely for the purpose of solving.
6. The **Variable Mapping** tab lets you view any variables contained in the selected Project.

   When there are variables in the source design, you can choose to “map” these variables to constant values, expressions or variables in the target designs. Variable mapping

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

becomes more important when the datalink type requires source and target design to be geometrically identical and source design is geometrically parameterized.

For linked designs with variables of the same name, you can click **Map Variables by Name** to automatically map same named variables. If a variable does not have a same named counterpart, it retains its value in the source design.



7. You can select the **Additional mesh refinements** tab to either **Apply mesh operation in target design on the imported mesh** (default), or **Ignore mesh operations in target design**.
8. Click **OK** to accept the setup and close the **Setup Link** window.

Setup Link for Non-Transient Solutions

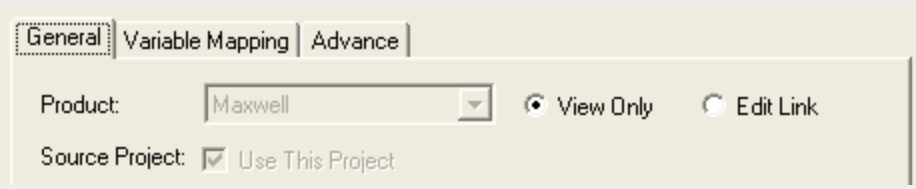
If you selected the **Use precomputed permeability data** option (refer to Permeability Options for Magnetostatic Solutions or Include DC Fields in 3D AC Magnetic T-Omega Solutions), or **Import mesh**, do the following to specify the source project, design, and solution containing the information you want to use. You can also view and map variables and, for Import Mesh, specify additional mesh refinements.

General Tab

1. By default, the **Setup Link** window opens on the **General** tab with only the **This project** radio button selected.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: If a link had previously been set up, the **Setup Link** dialog box opens in **View Only** mode with all settings disabled.



To enable settings, you can select **Edit Link**.

2. To select a source project do one of the following:
   - To use the current project as the source, check **Use This Project**. This disables the **Save source path to:** radio buttons and the ellipsis **[...]** button and its associated text field.
   - To specify a source project file other than the current project click the ellipsis **[...]** button to open a file browser window. When you have selected the project, click the **Open** button to accept the project file for the setup. You can use the check box to **Open as read only**.

   Use the radio buttons to specify whether to save the source path relative to **The project directory of the selected project** or **This project**.

3. When you select a source project file, the **Source Design** and the **Source Solution** fields are filled in with default values, and their drop down menus contain any available designs and solutions. You can use the drop down menus to choose from the available designs and solutions.

   The “Default” solution is the product-dependent solution of the first Setup. That is the setup listed first in the source design's project tree (alphanumerical order). A product-specific solution of this setup becomes the default solution. In most products, it is “LastAdaptive.” In a Transient solution type, it is “Transient.”

4. Use the check box to specify whether to **Simulate source design as needed**.
5. Use the check box to specify whether to **Preserve the source design solution**. Note that in the extractor mode, the source project will be saved upon exit. Extractor mode means that the software is opened during the link solely for the purpose of solving.

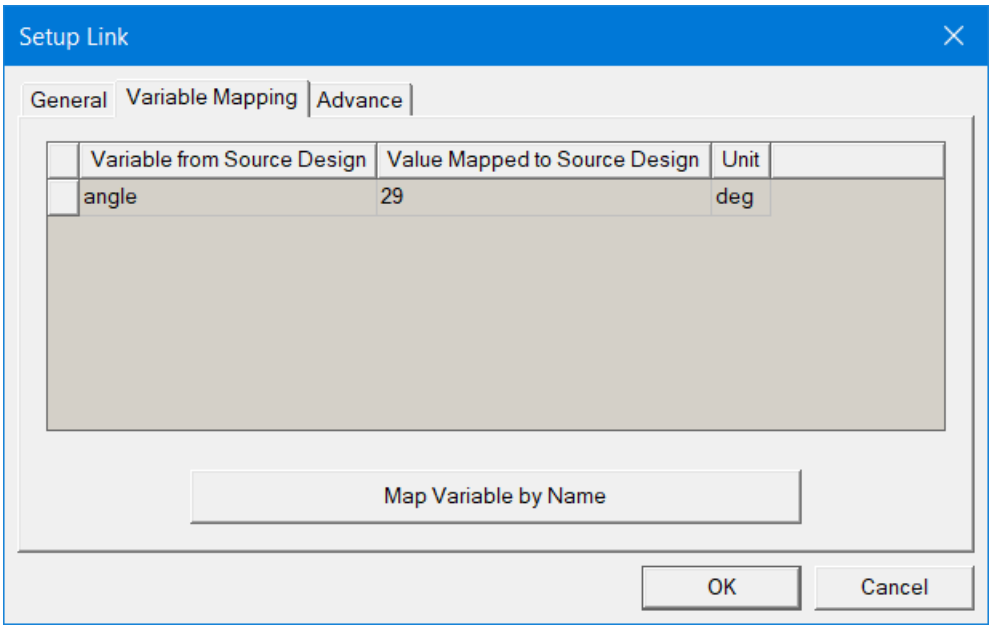
Variable Mapping Tab

The **Variable Mapping** tab lets you view any variables contained in the selected Project.

When there are variables in the source design, you can choose to “map” these variables to constant values, expressions or variables in the target designs. Variable mapping becomes

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

more important when the datalink type requires source and target design to be geometrically identical and source design is geometrically parameterized.



For linked designs with variables of the same name, you can click **Map Variables by Name** to automatically map same named variables. If a variable does not have a same named counterpart, it retains its value in the source design.

Additional Mesh Refinements Tab

> **Note:** This tab is present only for **Import mesh** link operations.

If you are setting up an Import mesh link you can use the radio buttons on the **Additional mesh refinements** tab to either **Apply mesh operation in target design on the imported mesh** (default), or **Ignore mesh operations in target design**.

Advance Tab

> **Note:** This tab is present only when using Permeability Options for Magnetostatic Solutions.

If you are setting up a link for the magnetostatic solver **Use precomputed permeability data** option, you can select the **Advance** tab to override the solved mu value from the source file with a user-specified control program. Refer to Permeability Options for Magnetostatic Solutions for details on using this feature:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enabling the Iterative Solver

The iterative solver provides an alternative to the direct solver. When you select the **Enable Iterative Solver** option, Maxwell invokes the iterative solver. Maxwell automatically switches to the direct solver if the iterative solver fails in convergence.

For more detail, see the technical notes for *Iterative Matrix Solver*.

To enable the iterative solver:

1. On the **Solution Setup** dialog box **Solver** tab, check the **Enable Iterative Solve** check box.

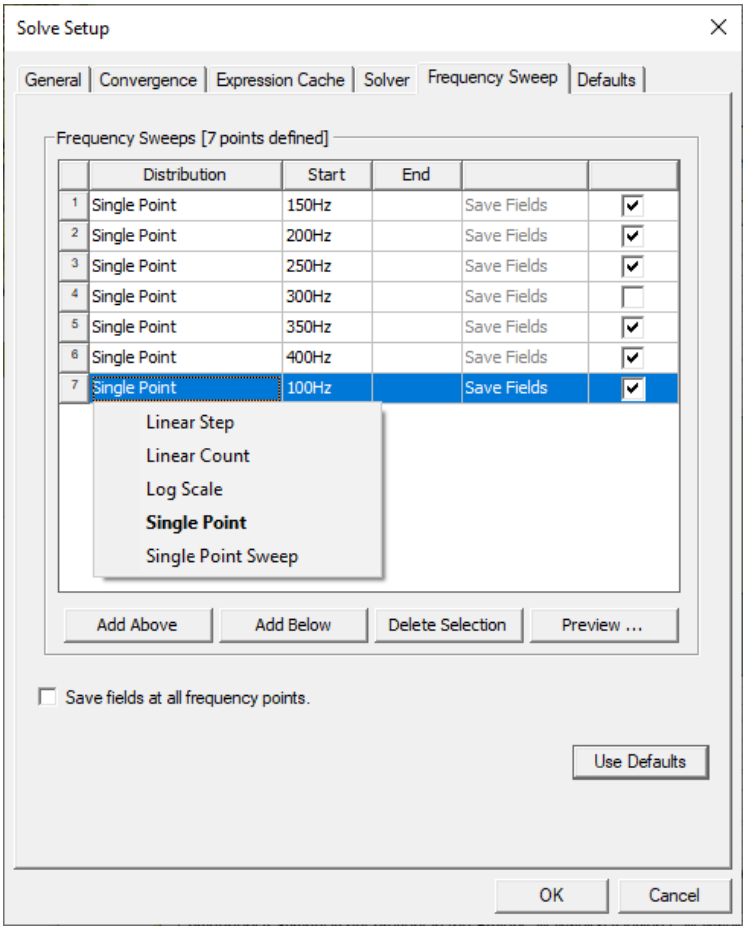
   This enables the **Relative Residual** input field.

2. Enter a value for the **Relative Residual**. The residual measures the convergence of the iterative solver. The default value is **1e-6** for the magnetostatic and electrostatic solvers and **1e-4** for the AC Magnetic solver.

Defining Settings on the Frequency Sweep Tab for AC Magnetic and AC Conduction Solutions

On the **Frequency Sweep** tab of the **Solve Setup** window:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



1. Use the **Add Above**, **Add Below**, and **Delete Selection** buttons to add or remove sweeps. Each row represents a separate sweep. Selecting a sweep's **Distribution** activates a drop-down menu allowing you to select the type of sweep. Information in the columns varies with the type of sweep.

    - **Linear Step** – A linear range of frequency points for which you specify a constant step size. Maxwell will solve the frequency point at each step in the specified frequency range, including the start and stop frequencies. For example, specifying 10 Hz for the **Start** frequency, 20 Hz for the **End** frequency, and 2.5 Hz for the step size instructs Maxwell to compute a solution for frequencies of 10, 12.5, 15, 17.5, and 20 Hz.
    - **Linear Count** – A linear range of frequency points for which you specify the number (count) of points within the frequency range defined by the **Start** and **End** frequencies. Maxwell will divide the frequency range, which includes the start and end frequencies, by the count you specify and solves each frequency point in the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

count.

- **Log Scale** – A range of points for which you specify the **Start**, **End**, and number of samples. Maxwell assigns the sampled points using intervals based on a logarithmic scale.
- **Single Point** – Individual frequency points for which you specify the **Start** value only. A check mark in the **Save Fields** column indicates that the fields for the point will be saved.
- **Single Point Sweep** – Converts the current selection to single points. This is convenient if you want to save (or not save) fields at specific frequency points.

2. Click the **Preview** button to display the currently defined sweep(s).
3. Optionally, you can check **Save fields at all frequency points** to save fields for the entire set of sweep ranges – except for **Single Point** entries that are unselected.

> **Note:** The AC Magnetic 3D Omega and A-Phi solvers can be used to solve for a single frequency; in that case no frequency sweep is allowed on the **Frequency Sweep** tab.

Defining Settings on the Defaults Tab for Non-Transient Solutions

Default analysis parameters are used to initialize the data for new setups.

To set up the default analysis parameters, define the following parameters under the **Defaults** tab of the **Solve Setup** dialog box:

1. Click **Save Defaults** if you want to save the defaults based on the data for the current setup.
2. Click **Revert to Standard Defaults** to clear the existing values.
3. Click **OK**.

Setting Analysis Parameters for Transient Solutions

When you set up an analysis, define the parameters on the following tabs of the **Solve Setup** dialog box (which appears when you click **Maxwell** > **Analysis Setup** > **Add Solution Setup**):

- General
- Save Fields
- Advanced
- Solver

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Expression Cache
- Defaults

Defining Settings on the General Tab for Transient Solutions

The following settings can be defined on the **General** tab of the **Solve Setup** dialog box for transient solutions:

- Setup Name
- Transient Setup (2D) or Transient Setup (3D)
- HPC and Analysis Options

Setup Name for Transient Solutions

By default, **Setup1** appears as the name of the first analysis you set up.

To change the name of the setup, type the new name in the **Name** text box on the **General** tab of the **Solve Setup** dialog box.

To disable the setup, uncheck the **Enabled** check box. Refer to Disabling and Enabling an Analysis Setup for additional information.

Transient Setup (2D)

To set up a 2D transient analysis, define the following parameters on the **General** tab of the **Solve Setup** window:

Adaptive Time Step

1. Determine whether you want the software to adaptively alter the time step during the simulation to optimize the solution for speed and accuracy. If so, check the **Adaptive Time Step** check box. If not, continue with step _ .
2. Enter the **Stop Time**, which is the value at which the solution will stop, and select the units.
3. Enter the **Initial Time Step**. This will be the time step used to start the simulation and begin the adaptive process.
4. Enter the **Maximum Time Step** and **Minimum Time Step** to set the allowable range the adaptive process use to alter the time step
5. Enter a value for the **Error Tolerance**. Generally this value should be smaller than the minimum time step specified by at least an order of magnitude.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Non-Adaptive Time Step

1. Enter the **Stop Time**, which is the value at which the solution will stop, and select the units.
2. Enter the **Time Step** for the analysis, which defines the time increments, and select the units. The time step can also be defined as the function of time, speed, and position.

> **Note:**
>
> When you have an external circuit, the Time Step is controlled from the circuit simulator, and the time step settings in the Solve Setup window are ignored. This can be controlled using the **Maxwell 2D** > **Excitations** > **External Circuit** > **Set Minimum Time Step command**.
>
> Time Step settings in the Solve Setup window are also ignored for transient-transient simulation in which case Twin Builder controls the time step.

Transient Setup (3D)

To set up a 3D transient analysis, define the following parameters on the **General** tab of the **Solve Setup** window:

1. Enter the **Stop Time**, which is the value at which the solution will stop, and select the units.
2. Enter the **Time Step** for the analysis, which defines the time increments, and select the units. The time step can also be defined as the function of time, speed, and position.

> **Note:** When you have an external circuit, the Time Step is controlled from the circuit simulator, and the time step settings in the Solve Setup window are ignored. This can be controlled using the **Maxwell 3D** > **Excitations** > **External Circuit** > **Set Minimum Time Step command**.
>
> Time Step settings in the Solve Setup window are also ignored for transient-transient simulation in which case Twin Builder controls the time step.

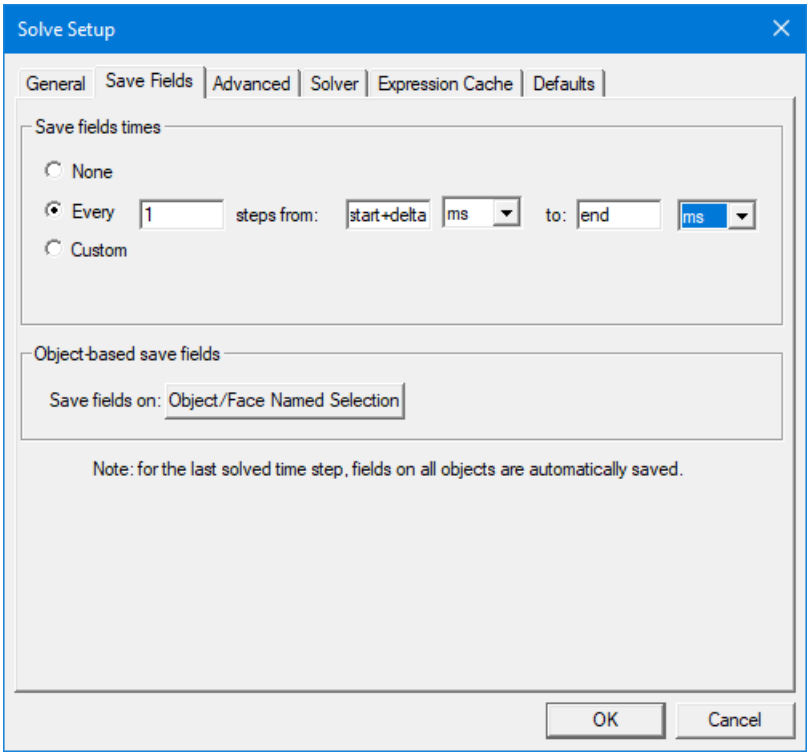
Defining Settings on the Save Fields Tab for Transient Solutions

Fields are automatically saved for the last completed time step for the transient analysis. The **Save Fields** tab can be used to specify additional time points at which to save fields and to limit the objects for which the fields are saved.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

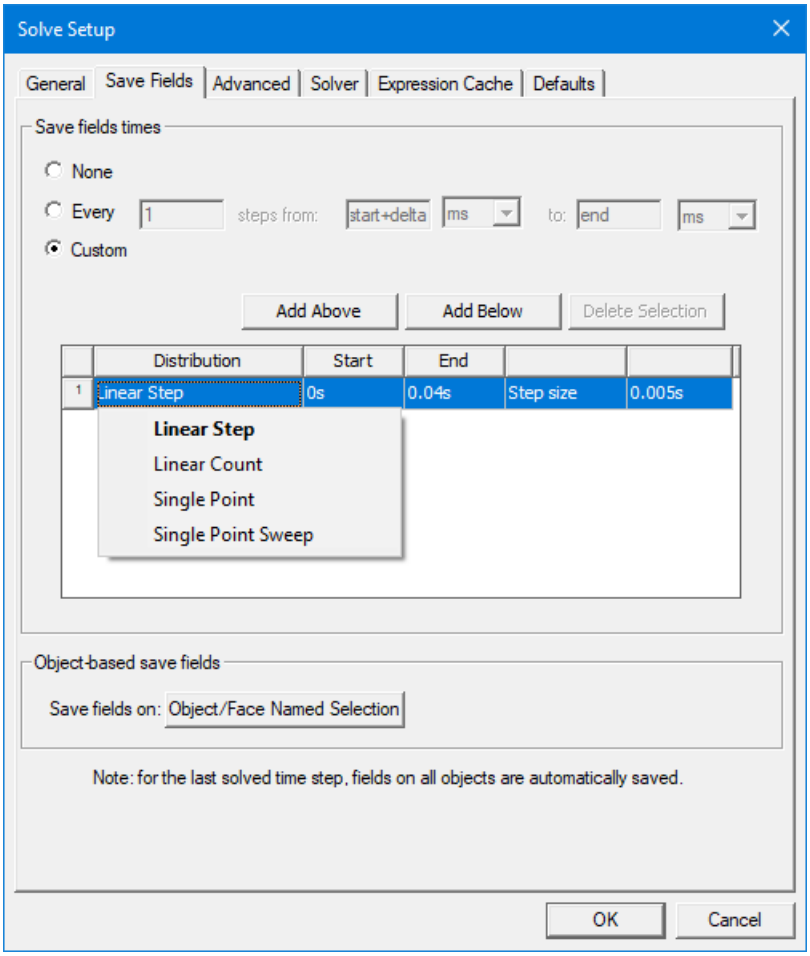
1. Specify the field save times:
   - **None** - only the last completed time step is saved. This is the default.
   - Every **N** timesteps - Enter the number of steps, start time, stop time and select the units. The values can be constants, variables or expressions. Note that the results for last completed time step will also be included.

     For example, if the variable start is 2 ms, delta is 1ms, and end is 12 ms, and the solution is calculated every ms, fields would be saved at 3 ms, 5 ms, 7 ms, 9 ms, 11 ms, and 12 ms.



   - **Custom** - select this option to add time points as desired.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. For **Custom**, click the **Preview** button to view the save field times. Some cases do not support a preview (for example, when variables are used to define the settings or when a user-control program is used). You can add and remove rows at your desired locations using the **Add Above**, or **Add Below**, and **Delete Selection** buttons.
3. Click **OK**.

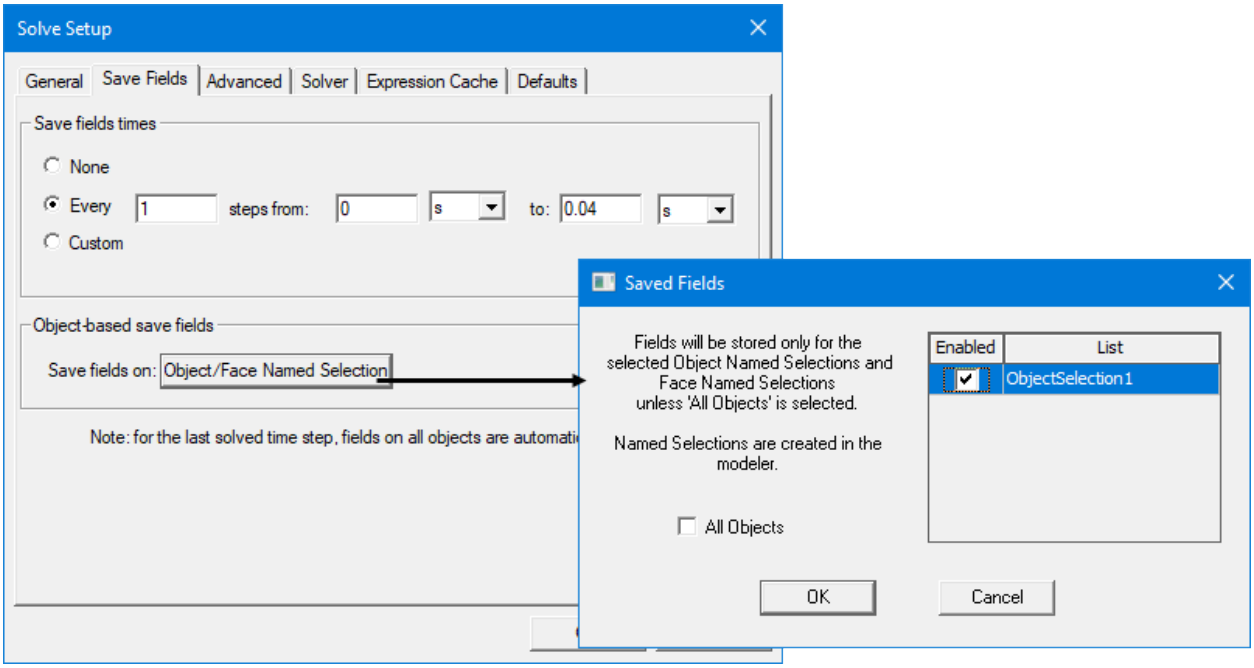
Save Fields on Object/Face Named Selection

By default, a transient solution saves fields on all objects in the design for the specified Save Field Times. To save simulation time and disk space, you can specify named selection of objects and/or faces that are most relevant to the simulation so that only the fields on those entities will be saved and be available for postprocessing.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

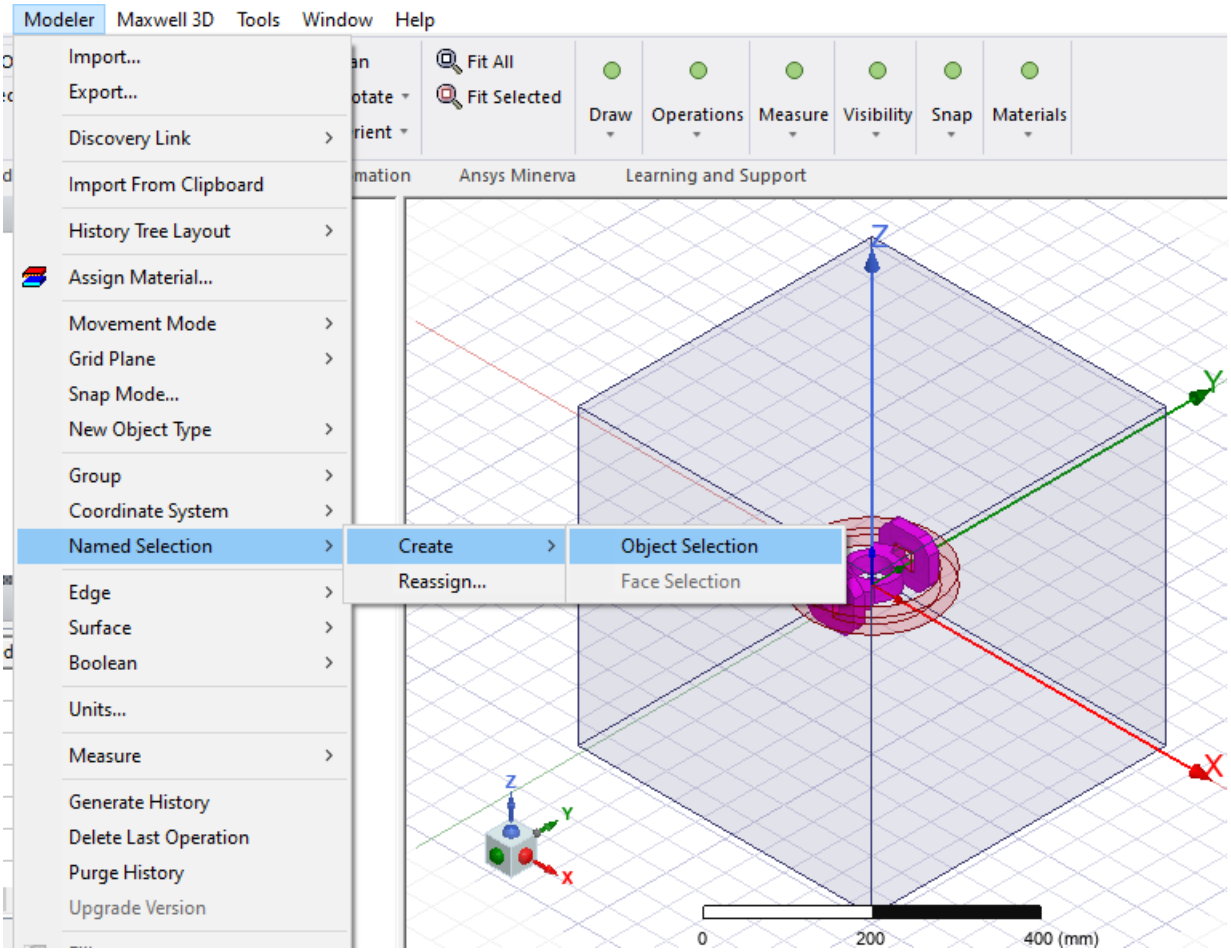
- For 2D designs, only sheet objects of object named selections will be considered.
- For 3D designs, only solid objects of object named selections will be considered.
- Field results/plots at the last completed timestep will automatically include fields for all objects.



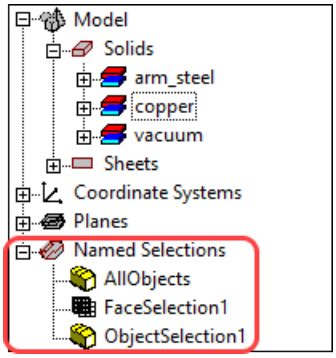
To create an object or face selection:

1. Select the objects or faces you would like to save fields for.
2. Select **Modeler** > **Named Selection** > **Create** > **Object Selection** or **Face Selection**. The example that follows shows the creation of an object selection.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The resulting list is added to the history window under the **Named Selections** entry. The selections are named numerically, but you can change its name in its Properties window.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining Settings on the Advanced Tab for Transient Solutions

The following settings can be defined on the **Advanced** tab of the **Solve Setup** window:

- Control Program
- Start/Continue from a previously solved setup
- Import mesh

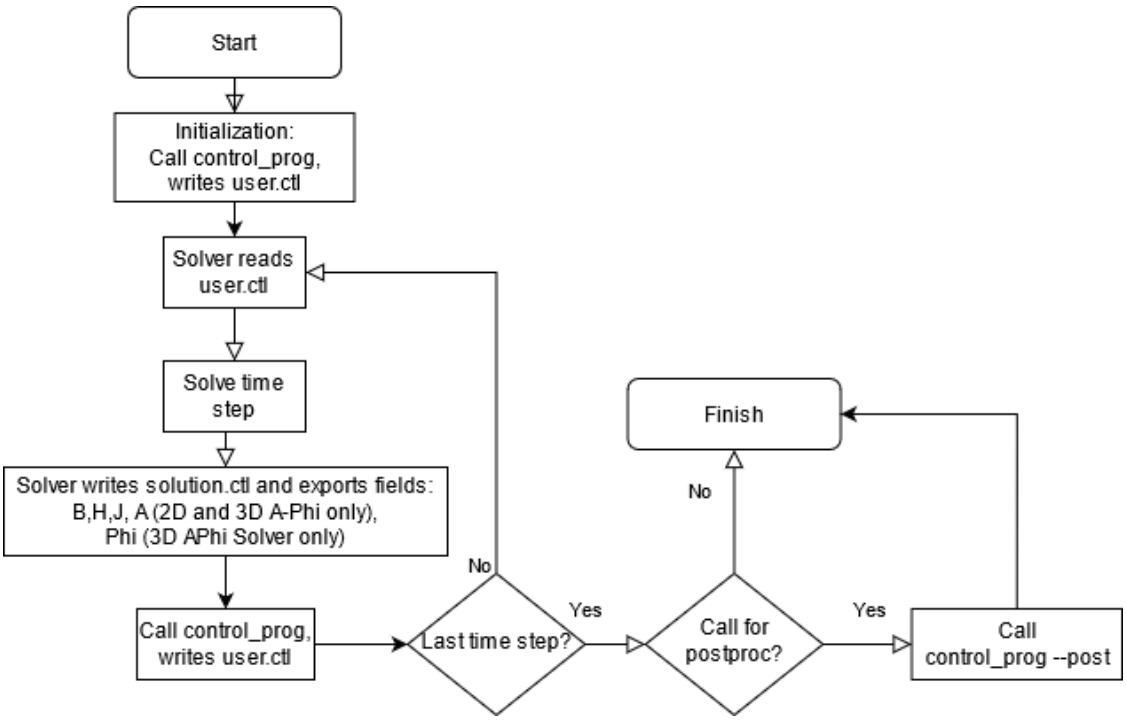
Using a Control Program in Maxwell 2D and 3D Transient Solutions

User control programs are externally created executable files, or Python (`*.py`) scripts that are called after each time step, and allow you to control the source input, circuit elements, mechanical quantities, time step, and stopping criteria, based on the updated solutions.

> **Note:** The control program is executed from a temp directory that Maxwell creates for every setup run (For parametric setup a temp folder is create for each task). The solver copies a control program file with name <**setup_name**>**.py** for Python, or <**setup_name**>**.ctrlprog** for compiled program to the temp directory.

The user control program uses the file with fixed name **user.ctl** to output control parameters to control the execution of the transient solver. Each file must use a predefined syntax which is flexible enough to cover a wide range of items.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The process shown in the flowchart can be summarized as:

1. The solver calls the user-control program on initialization.
2. The user-control program writes control information to file **user.ctl**.
3. The transient solver reads control parameters from **user.ctl**.
4. The transient solver solves the current time step.
5. The transient solver copies the previous solution to **previous.ctl** and writes out solution information to **solution.ctl**. If requested, the solver exports:
   - B, H, J fields (**bfield.ctl**, **hfield.ctl**, **jfield.ctl**, respectively)
   - A field **afield.ctl** (available in 2D and 3D A-Phi only)
   - Phi field **phifield.ctl** (available in 3D A-Phi only).
6. Return to the transient solver if the control program succeeds with exit status 0 or fails with exit status non-zero.
7. Return to step 1 for the next time step.
8. (Optional) Call script after last time step for post processing. If user selects the **Call after last time step for post processing** check box (refer to *Activating a Control Program in Transient Solutions*), then the control program is called after the simulation is completed.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you use a Python control program, the script is called with `--post` flag

(for example: `my_script.py --post`).

For a compiled control programs the script is called with `-post` flag

(for example: `my_tool.exe -post`).

> **Note:** By default a control program script will be called by the pre-installed Python interpreter:
> `<install_path>\ANSYS Inc\v252\AnsysEM\commonfiles\CPython\3_10\winx64\Release\python\python.exe`
> However, the user can specify a custom Python interpreter by setting the following environment variable:
> **EM_CTRL_PROG_PYTHON_PATH=<*path_to*\python.exe>**

Activating a Control Program in Transient Solutions

Use the following steps to access and invoke a user-control program to use with the Maxwell 2D or 3D transient solver. User control programs can be either externally created executable (`.exe`) files, or Python (`*.py`) scripts.

To specify a control program to use to generate the solution:

1. Select the **Use Control Program** check box.

   The control program field becomes active, allowing you to enter the name of the user control program.

2. Click the **...** button to select the user control program you want to use during the solution generation process.

   The **Open** window appears.

3. Select the control program, and click **Open** to return to the **Advanced** tab of the **Solve Setup** dialog box.

4. Enter the arguments to pass to the control program in the **Arguments** text box. The solver calls the program in the following format:

   ```
   program_name specified_arguments
   ```

5. If you select the **Call after last timestep for post processing** check box, then the control program is called after the simulation is completed.

   If you use a Python control program, the script is called with `--post` flag

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(for example: `my_script.py --post`).

For a compiled control programs the script is called with `-post` flag

(for example: `my_tool.exe -post`).

6. Click **OK** to accept the configuration or **Cancel** to ignore the settings. You return to the **Solve Setup** window.

Control Program File Formats in Transient Solutions

User control programs and related files have their own file formats that should be followed when creating control program files.

- User-Control File Format
- Solution and Previous Control File Formats
- User-Control Fields Extraction Format
- Output variable values are output to a file with a fixed name of **outputvar.ctl**. The file contents use the following format:

```
<output variable name> <output variable value>
<output variable name> <output variable value>
...
```

The following summarizes the files related to user control programs for transient solutions.

File Name	Functionality
user.ctl	Input data that controls Maxwell Solver
solution.ctl	Data for global quantities such torque, current, speed, etc.
previous.ctl	Same data as solution.ctl but for previous timestep
outputvar.ctl	Contains expression cache variables
var.data	Variations information from design and project variables
afield.ctl	Magnetic potential (available only in 2D and 3D A-Phi solver)
bfield.ctl	Magnetic flux density
hfield.ctl	Magnetic field strength
jfield.ctl	Electric current density
phifield.ctl	Electric Scalar Potential (available only in 3D A-Phi solver)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

User Control File Format

The **user.ctl** file is created by the user control program. Below are descriptions of the keywords that can be used in a **user.ctl** file, and indicating the solver types for which they are valid.

Keyword	Description	2D	3D	3D A-Phi
windingSrc <winding_name> <val>***	Voltage [V]/Current [A] value for Winding	X	X	X
windingR <winding_name> <val>***	Resistance value [Ohm] for Voltage Winding	X	X	X
windingL <winding_name> <val>***	Inductance value [H] for Voltage Winding	X	X	X
loadTorque <value>	Load Torque [N*m] for rotational motion only	X	X	
loadForce <value>	Load force in MotionSetup [N] for translational only	X	X	
damping <value>	Damping coefficient [translational: N*s/m; rotation: N*m*s/rad]	X	X	
loadInertia <value>	Load Inertia in MotionSetup [kg*m²] for rotational only	X	X	
mass <value>	Mass in MotionSetup [kg] for translational only	X	X	
speed <value>	Speed of the moving object [translation - m/s; rotation - rad/s]	X	X	
position <value>	Position of the moving object [[translation - user unit (from Motion Setup); rotation – rad]	X	X	
timeStep <value>	Time step size [s]	X	X	X
stop <0 or 1>	Stop solver if value is equal to 1	X	X	X
exportFieldAtMeshNodeOnAllObjects	Request to export fields on all objects in design[1]	X	X	X
exportFieldAtMeshNodeOnObject <objectID>*	Request to export fields on specific object	X	X	X

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Keyword	Description	2D	3D	3D A-Phi
NStep <value>	Export fields only every N steps (works only in conjunction with exportFieldAtMeshNode<...>)	X		
FieldSelect <value>**	Export only specific field (one of A,B,H,J) (works only in conjunction with exportFieldAtMeshNode<...>)	X		
Binary <0 or 1>	Export fields in binary format (works only in conjunction with exportFieldAtMeshNode<...>)	X		
voltageSrc <voltage_name> <source_value>***	Voltage source value [V]			X
currentSrc <current_name> <source_value>***	Current source value [A]			X
Full line could be repeated for *multiple object IDs; **multiple fields; ***multiple excitation names				

[1] Regarding a skewed, multi-slice model, for the first slice, the output file name is "FieldName.ctl", which is same as that in non-skew model, and for the other slices, the output file name is "FieldName_%d.ctl", where %d is a number from 1 to (numSlice-1).

The following **user.ctl** file example:

- Sets the time step to 5 ms.
- Sets Resistance to 1 μOhm and Inductance to 1 μH for Winding A and Winding B.
- Sets Voltage for Winding A to 10 V and for Winding B to -10 V.
- After time step is completed all fields for the object with ID 88 will be exported.

```
begin_data
 windingSrc WindingA 10
windingR WindingA 1e-6
windingL WindingA 1e-6
windingSrc WindingB -10
windingR WindingB 1e-6
windingL WindingB 1e-6
exportFieldAtMeshNodeOnObject 88
timeStep 5e-3
stop 0
end_data
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

User Control Fields Extraction Format

The **user.ctl** file is created by the user control program. Depending on the solver the following field types are available during export:

Field	2D	3D	3D A-Phi
B – Magnetic Flux Density	X	X	X
H – Magnetic Field Strength	X	X	X
J – Current Density	X	X	X
A – Magnetic Vector Potential	X		X
Phi - Electric Scalar Potential			X

To extract fields from Maxwell, the control program must create a **user.ctl** file with one of the two following syntax strings:

> **Note:** You can combine field export keywords with other keywords.

Extract fields on all objects

user.ctl

```
begin_data
exportFieldAtMeshNodeOnAllObjects
end_data
```

Extract fields by object ID

user.ctl

```
begin_data
exportFieldAtMeshNodeOnObject <objectID>
exportFieldAtMeshNodeOnObject <objectID>
end_data
```

You can get the desired object IDs as follows:

1. Select **Tools** > **Open Command Window**.
2. In the following example code snippet, replace "Stator" in the `object_name` with the name of the desired object.

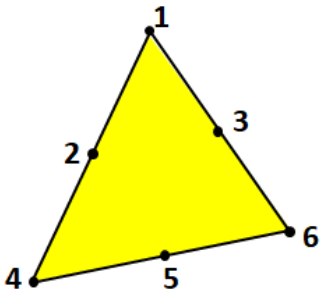
```
object_name = "Stator"
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
oProject = oDesktop.GetActiveProject()
oDesign = oProject.GetActiveDesign()
oEditor = oDesign.SetActiveEditor("3D Modeler")
object_id = oEditor.GetObjectIDByName(object_name)
print(object_id)
AddInfoMessage(str(object_id))
```

3. Copy and paste the code snippet into the Command Window and execute.
4. Repeat as needed for additional objects.

Data Structure and Nodes Order for 2D Fields Extraction



Node order in Maxwell 2D element

The Maxwell 2D solver writes the data and updates field files (**afield.ctl**, **bfield.ctl**, **hfield.ctl**, **jfield.ctl**) at each time step with the following structure:

Time
M
ID K x_1 y_1 x_4 y_4 x_6 y_6 B_{x1} B_{y1} B_{x2} B_{y2} B_{x3} B_{y3} B_{x4} B_{y4} B_{x5} B_{y5} B_{x6} B_{y6}

where:

- Time – is the time corresponding to the time-stepping FE solution
- M – is the total number of elements
- ID – is the object ID
- K – is the element number
- x_i, y_i – are the coordinates of node i, for i in [1, 4, 6]
- B_{xi} B_{yi} – is the field X, Y component value of node i, for i in range 1 to 6

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

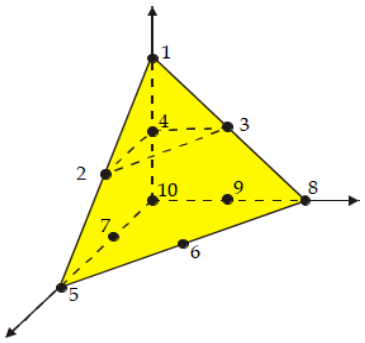
Note: For vector potential, the **afield.ctl** file has one component of the field, which is A_z.

Note: Coordinates of the element are given for the vertices (1, 4, 6) only. Middle points (2, 3, 5) can be calculated from the other given coordinates since they are located in the middle as shown above, for example: $x_2 = (x_1+x_4)/2$.

The following is an example for **bfield.ctl** data extracted in an object with ID 761 that consists of 2 elements at t = 0.04 sec.

```
0.040000
2
761 550 0.002 0.037 0.003 0.037 0.003 0.037 -2.220 0.488 -1.540 0.783 -1.799 0.950 -0.860 1.077 -1.119 1.245 -1.379 1.413
761 573 0.018 0.032 0.021 0.032 0.020 0.031 -0.695 1.266 -0.626 1.433 -0.810 1.378 -0.556 1.599 -0.740 1.544 -0.924 1.490
```

Data Structure and Nodes Order for 3D Fields Extraction



Node order in Maxwell 3D element

The Maxwell 3D solver writes the data and updates field files (**bfield.ctl**, **hfield.ctl**, **jfield.ctl**) at each time step with the following structure:

Time
M
ID K x_1 y_1 z_1 x_5 y_5 z_5 x_8 y_8 z_8 x_{10} y_{10} z_{10} B_{x1} B_{y1} B_{z1} B_{x2} B_{y2} B_{z2} … B_{xi} B_{yi} B_{zi} … B_{x10} B_{y10} B_{z10}

where:

- Time – is the time corresponding to the time-stepping FE solution
- M – is the total number of elements

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- ID – is the object ID
- K – is the element number
- x_i, y_i, and z_i– are the coordinates of node i, for i in [1, 5, 8, 10]
- B_{xi} B_{yi}, B_{zi} – is the field X, Y, Z component value of node i, for i in range 1 to 10

> **Note:** The coordinates of the 3D element are given for the vertices (1, 5, 8, 10) only. Middle points (2, 3, 4, 6, 7, 9) can be calculated from the other given coordinates since they are located in the middle as shown above, for example: $x_2 = (x_1+x_5)/2$

Related Topics

User-Control File Format

Using a Control Program in Maxwell 2D and 3D Transient Solutions

Activating a Control Program in Transient Solutions

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution and Previous Control File Formats

The **solution.ctl** and **previous.ctl** files are created by the transient solver. Below is a list of keywords and definitions applicable to these files.

Keyword	Description	2D	3D	3D A-Phi
time <value>	Current time in simulation [s]	X	X	X
position <value>	Position when motion setup is assigned[translation - user unit (from Motion Setup); rotation – rad]	X	X	
speed <value>	Speed when motion setup is assigned[translation - m/sec; rotation - rad/sec]	X	X	
torque moving_group <value>	Torque [N*m]	X	X	
force moving_group <value>	Force on moving objects [N]	X	X	
strandLoss <value>	Stranded loss for all objects [W]	X	X	X
solidLoss <value>	Solid loss for all objects [W]	X	X	X
coreLoss <value>	Core loss for all objects [W]	X	X	X
timeStep <value>	timestep size [s]	X	X	X
windingFlx <winding_name> <value>*	Winding Flux value [Wb]	X	X	X
windingEmf <winding_name> <value>*	Winding EMF value [V]	X	X	X
windingI <winding_name> <value>*	Winding Current value [A]	X	X	X
windingV <winding_name> <value>*	Winding Voltage value [A] (voltage windings only)	X	X	X
voltageV <voltage_name> <value>*	Voltage value on voltage excitation [V]			X
voltageI <voltage_name> <value>*	Current value on voltage excitation [A]			X
currentV <current_name> <value>*	Voltage value on current excitation [V]			X
currentI <current_name> <value>*	Current value on current excitation [A]			X
* Could be repeated for multiple excitation names				

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

An example `solution.ctl` file for 2D/3D solvers:

```
begin_data
 time <current_time>
 strandLoss <value>
 solidLoss <value>
 coreLoss <value>
 torque <torque>
 speed <speed>
 position <position>
 windingI <winding_name> <value>
 windingEMF <winding_name> <value>
 windingFlx <winding_name> <value>
end_data
```

An example `solution.ctl` file for the A-Phi solver:

```
begin_data
 time <current_time>
 strandLoss <value>
 solidLoss <value>
 coreLoss <value>
 windingI <winding_name> <value>
 windingEMF <winding_name> <value>
 windingFlx <winding_name> <value>
 voltageV <voltage_name> <value>
 voltageI <voltage_name> <value>
end_data
```

Start/Continue from a Previously Solved Setup

Enabling the **Start/Continue from a previously solved setup** option allows the source design solution to be linked to a set of different target designs, where the user can continue the solution based on a different parameter/circuit setup. For example, to speed up the process of either a 2D or 3D transient solution reaching a steady state, you can use this option to link a 2D (or 3D) AC magnetic solution as the initial condition for a corresponding transient solution.

Source and target designs linked through this workflow are required to have the same FEM degrees of freedom. Therefore, the excitations on the source and the target need to be of the same type. If an external circuit is used to excite a winding excitation, the same external circuit netlist needs to be coupled in both designs. However, numerical values (and parameters' values) defining the excitations can be different.

If the **Start/Continue from a previously solved setup** check box is selected, the **Setup Link** button becomes active, and the **Setup Link** dialog box automatically appears in which the user

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

can link the current design to a project solution set based on a different parameter/circuit setup. Refer to Setup Link for Transient Solutions for details on setting up this link.

> **Note: Power loss** and **Force density distribution** cannot be computed when **Start/Continue from a previously solved setup** is enabled.

Import Mesh for Transient Solutions

The **Import mesh** check box can be selected in the "target" design if the mesh for the same geometry is to be used in the analysis. This setup is designed to be generally applicable to all transient solutions. This feature is not necessarily related to the dynamic demagnetization distribution capability.

If the **Import mesh** feature is selected, the **Setup Link** button becomes active, the **Setup Link** dialog box, which includes an **Additional mesh refinements** tab automatically appears, and the mesh will be imported from the source design to the target design. Refer to Setup Link for Transient Solutions for details on setting up this link.

> **Note:** The usual way to create geometrically equivalent designs is by copying all geometry objects from one design and then using **Import From Clipboard** in the other design.

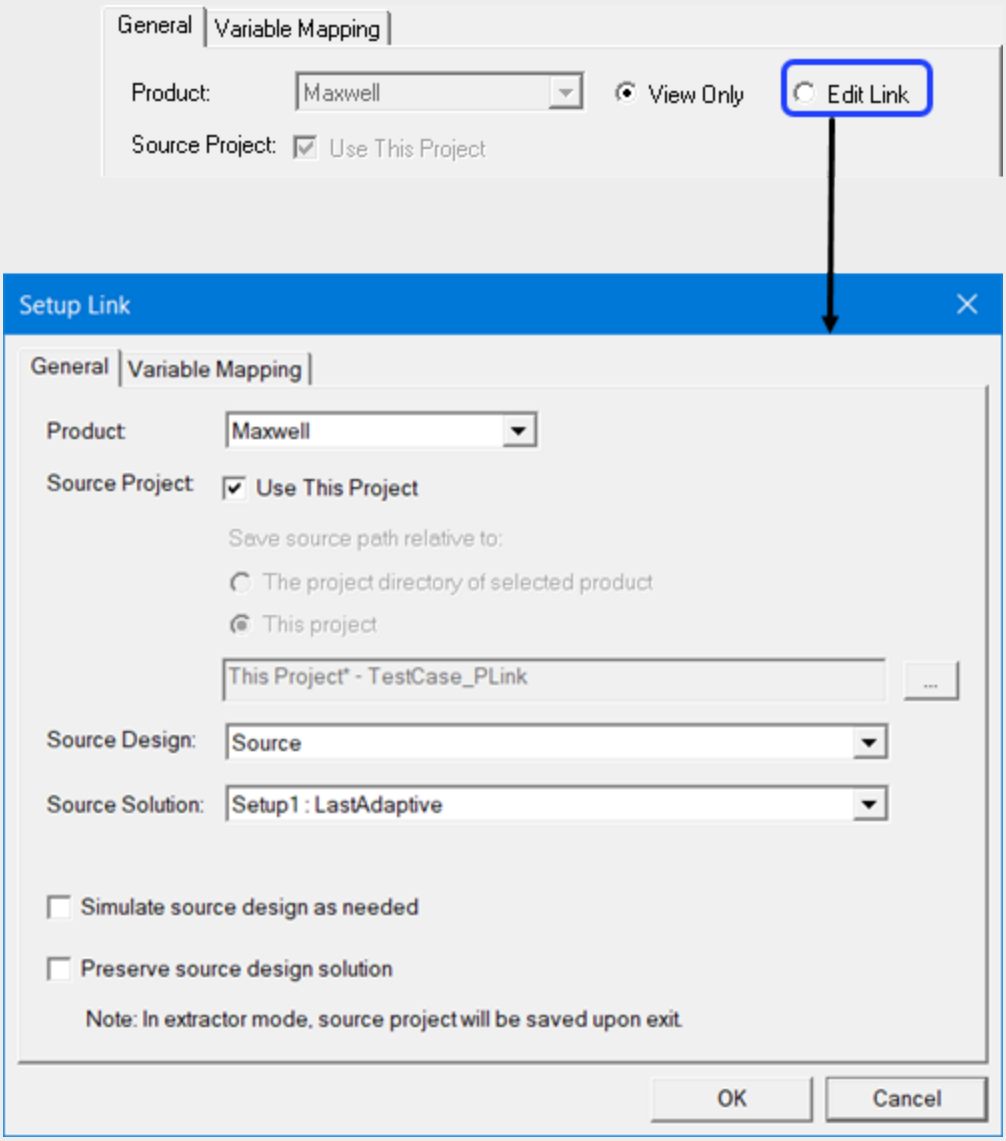
Setup Link for Transient Solutions

If you selected **Start/Continue from a previously solved setup**, or **Import mesh**, do the following to specify the source project, design, and solution containing the information you want to use. You can also view and map variables and, for Import Mesh, specify additional mesh refinements.

1. If the **Setup Link** window is not already open, click **Setup Link** (located at the bottom of the **Solve Setup** window's **Advanced** tab) to open the **Setup Link** window.
2. By default, the **Setup Link** window opens on the **General** tab with only the **Use this project** radio button selected.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: If a link had previously been set up, the **Setup Link** window opens in **View Only** mode with all settings disabled.



To enable settings, you can select **Edit Link**.

3. To select a source project do one of the following:
   - To use the current project as the source, check **Use This Project**. This disables the **Save source path relative to:** radio buttons and the ellipsis **[...]** button and its associated text field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To specify a source project file other than the current project click the ellipsis **[...]** button to open a file browser window. When you have selected the project, click the **Open** button to accept the project file for the setup. You can use the check box to **Open as read only**.

  Use the radio buttons to specify whether to save the source path relative to **The project directory of selected product** or **This project**.

4. When you select a source project file, the **Source Design** and the **Source Solution** fields are filled in with default values, and their drop down menus contain any available designs and solutions. You can use the drop down menus to choose from the available designs and solutions.

   The **Default** solution is the product-dependent solution of the first Setup. That is the setup listed first in the source design's project tree (alphanumerical order). A product-specific solution of this setup becomes the default solution. In most products, it is **LastAdaptive**.

5. Use the check box to specify whether to **Simulate source design as needed**.
6. Use the check box to specify whether to **Preserve source design solution**. Note that in the extractor mode, the source project will be saved upon exit. Extractor mode means that the software is opened during the link solely for the purpose of solving.
7. The **Variable Mapping** tab lets you view any variables contained in the selected Project.

   When there are variables in the source design, you can choose to “map” these variables to constant values, expressions or variables in the target designs. Variable mapping becomes more important when the datalink type requires source and target design to be geometrically identical and source design is geometrically parameterized.

   For linked designs with variables of the same name, you can click **Map Variable by Name** to automatically map same named variables.

8. The **Additional mesh refinements** tab, which is present only if you chose Import Mesh on the **Solve Setup** window's **Solver** tab, allows you to choose either to **Apply mesh operations in target design on the imported mesh** (default) or **Ignore mesh operations in target design**.
9. Click **OK** to close the **Setup Link** window and return to the **Solve Setup** window's **Advanced** tab.

Using an AC Magnetic Solution as the Initial Condition for a Transient Solution

In order to speed up the process of either a 2D or 3D transient solution reaching a steady state, you can set the solution from a 2D or 3D AC magnetic solver as the initial condition for the corresponding transient solver.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: Because the AC magnetic solution corresponds to a stationary operating condition (zero speed), in order to generate the initial condition for the transient solution with non-zero speed, a transformation of conductivity on the moving objects where eddy current is taken into account is required in the AC magnetic simulation. The transformation is

$$\sigma_0 = \sigma \cdot s_n$$

where σ_0 is the transformed conductivity from speed n to speed zero, which is to be set in the AC magnetic simulation, σ is the real conductivity in transient simulation, and s_n is the slip, defined as:

$$s_n = \frac{n_s - n}{n_s}$$

where n_s is the synchronous speed.

If there is an end connection, the end resistance R used in the transient simulation also has to be referred to the value R_0 associated with zero speed before being used in the AC magnetic solution by:

$$R_0 = \frac{R}{s_n}$$

Prerequisites

- Source design - Maxwell 2D (or 3D) AC Magnetic (Last Adaptive solution only)
- Target design - Maxwell 2D (or 3D) Transient
- The source and target designs must be identical (identical geometries, boundaries, excitations, windings, etc).

To setup and use a 2D (or 3D) AC magnetic solution as the initial condition for a corresponding 2D (or 3D) transient solution:

1. Create an AC magnetic source design, and ensure that **Solve Matrix: After last pass** is selected on the **General** tab of the AC magnetic **Solve Setup** window.
2. Solve the AC magnetic design.
3. Create a transient target design, whose geometry, boundary conditions, excitations, windings, etc., are identical to the the source AC magnetic design.
4. On the transient target design's **Solve Setup** dialog box, **Advanced** tab, click **Start/Continue from a previously solved setup** to open the **Setup Link** dialog.
5. On the **Setup Link** dialog **General** tab, select the source AC magnetic design and solution (Last Adaptive solution only). Refer to "Setup Link for Transient Solutions " on

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

page 15-65 for details on setting up this link.

6. Solve the target transient design.

Note:

- If the source and target designs are not identical, an error message displays.
- If the AC magnetic source design has not been solved, an error message displays.
- If a legacy AC magnetic design solved with a previous release of Maxwell is used as the source design, you must re-solve the AC magnetic design before solving the target transient design or an error message will be displayed.

Defining Expressions for Transient Solutions

You can specify additional evaluation criteria through the use of expressions and output variables.

To set expressions as evaluation criteria:

1. Right-click on the Analysis icon in the Project Manager tree to open the **Solve Setup** window.
2. Click the **Expression Cache** tab and click the **Add...** button to open the **Add to Expression Cache** dialog box.
3. Specify the Context for the expression, including **Report Type**, **Solution**, and **Parameter**.
4. Under the **Trace** tab, select from the **Category**, **Quantity**, and **Function** lists to create expressions.

   Selecting a listed category lists the Quantities and Functions available for each category. If you have defined one or more output variables, you can see them by selecting the **Output Variables** category. The **Output Variables** button opens a dialog box that lets you define output variables. See Specifying Output Variables.

   When you have created an expression, it appears in the **Expression** field of the **Trace** tab. If desired, you can use the **Range Function** button to select range functions to apply to the expression.

   Under the **Calculation Range** tab, you can view the values of available sweep variables. Clicking the ellipsis **[...]** button in the **Edit** column opens a list of values from which you can select.

5. When finished defining the expression that you want to add to the cache, click the **Add Calculation** button.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This adds the selected expression and the associated context to a table in the **Expression Cache** tab. You can define additional expressions with contexts and add them in the same way.

6. When finished adding expressions, click **Done** to close the **Add to Expression Cache** dialog box.

   The **Expression Cache** tab lists the expressions you have added in a table:

   - The **Title** field is editable, by default showing the name as built from the expression (with underscores removed).
   - The **Expression** field shows the full expression. If necessary, you can resize the **Solve Setup** window. You can also resize each columns in the table.
   - The **Context** column for non-field types lists the parameter type specifying the force, torque or matrix setup. For these types the context also can be given as part of the expression. For example, the expression *Force1.Force_x* with a context of *None* is equivalent to an expression of *Force_x* with a context of *Force1*.
     For field quantities, the **Context** can be the geometry on which to evaluate the field quantity - a point or line. If you use a line you need to use some range function to reduce the dimensionality to a single value (or use the Calculation Range tab to specify a single point along the line). Note that, if you create a named expression in the field calculator that already evaluates to a single value, then the **Context** here will likely be *None*.
   - The **Intrinsics** column shows a clickable button that opens an **Edit Calculation Range** dialog box. If the **Intrinsics** column button shows **None** you cannot edit the value. If the button shows variables, click the ellipsis **[...]** in the **Edit** column to display a list of the variable values from which you can select. Click **OK** to close the **Edit Calculation Range** dialog box and apply your selections to the **Expression Cache**.
     The **Intrinsics** field lists the values you may have set in the **Calculation Range** tab when you added the expression. This is used to set the values of “other” sweeps. Usually there are no other sweeps because the values of the variables are set in the definition of each variation. In some cases you may need to specify the values of “other” sweeps. For field values with a line for the context, you can specify the (normalized) distance along the line For eddy current field quantities you can also specify the phase for the calculation.

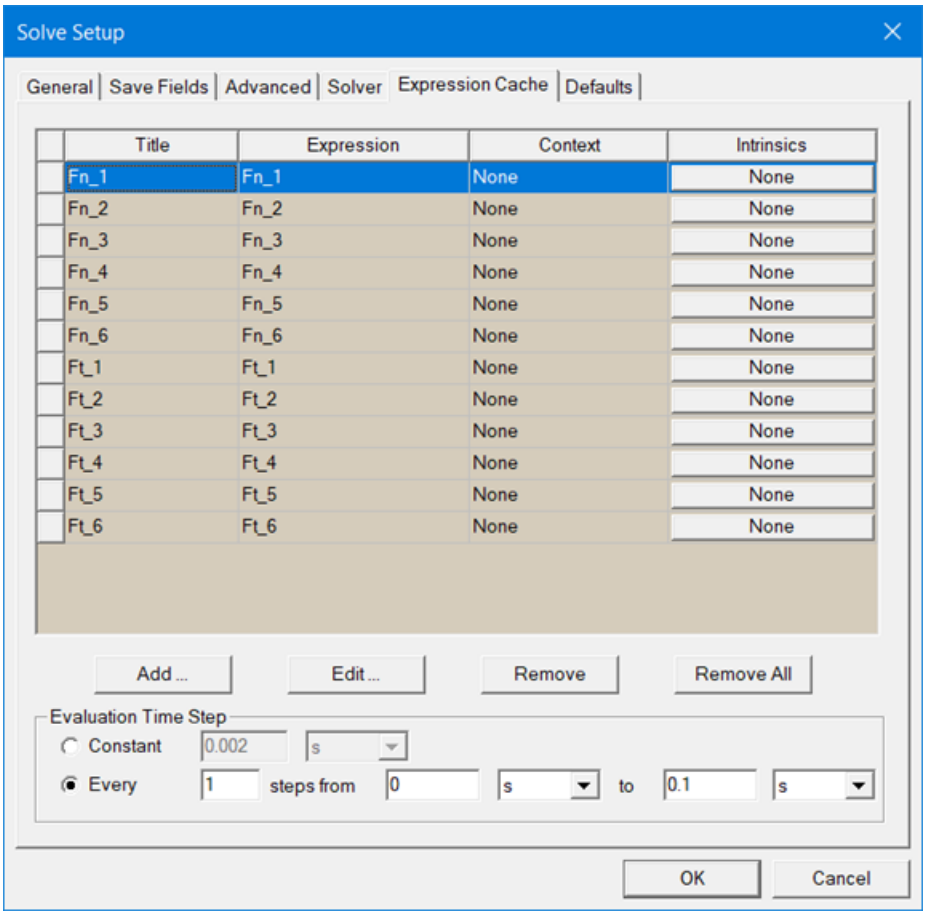
7. In the **Evaluation Timestep** panel, set how often the solver is to output field solutions. Select **Constant** to output solutions at a preset time interval (e.g., every 0.1 s); or **Number of solve steps** to output solutions each preset number of timesteps (for example, every three timesteps).
8. Optionally, buttons allow you to **Edit** and **Remove** selected expressions, and to **Remove All** expressions in the table.
9. When finished defining expressions, click **OK** to close the **Solve Setup** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Speeding Up a Maxwell 2D Parametric Transient Design that Uses Expression Cache

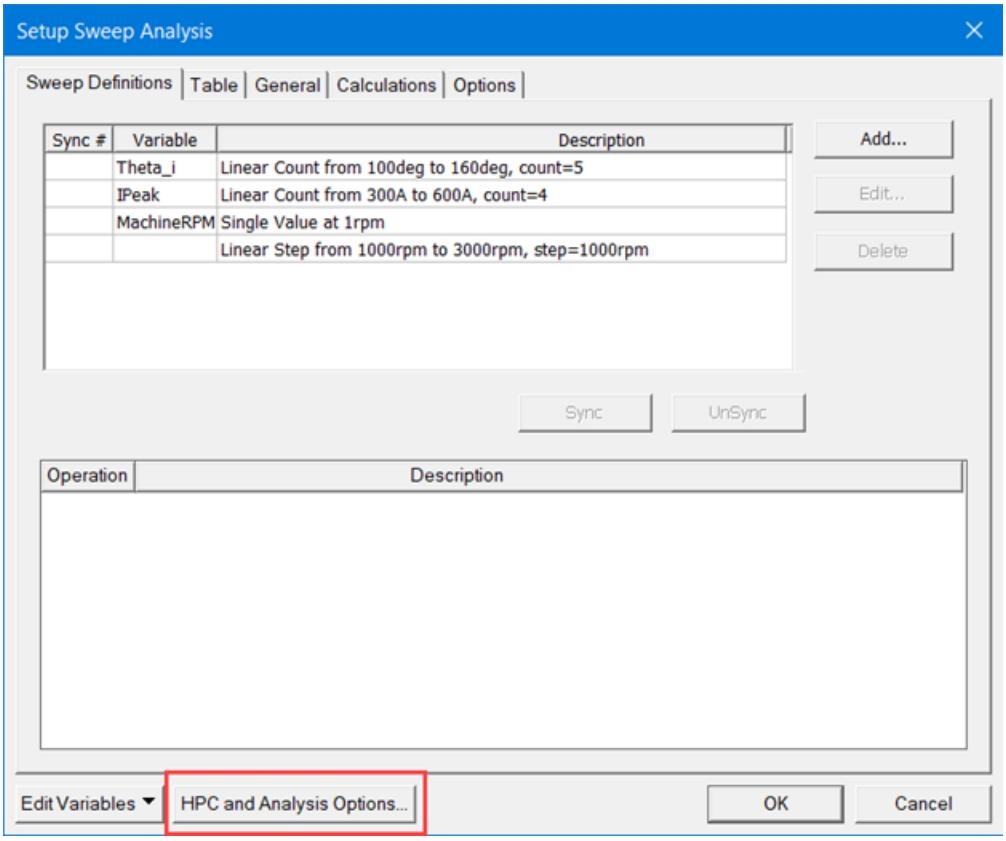
A Optimetrics\Parametric analysis of a Maxwell 2D transient design that uses expression cache can require substantial simulation time. To cut down on simulation time for this type of analysis, use the **Optimetrics Variations** and **Single level only** HPC settings.

The following example demonstrates how to configure these settings. The Solve Setup **Expression Cache** tab has been configured to have several expressions:



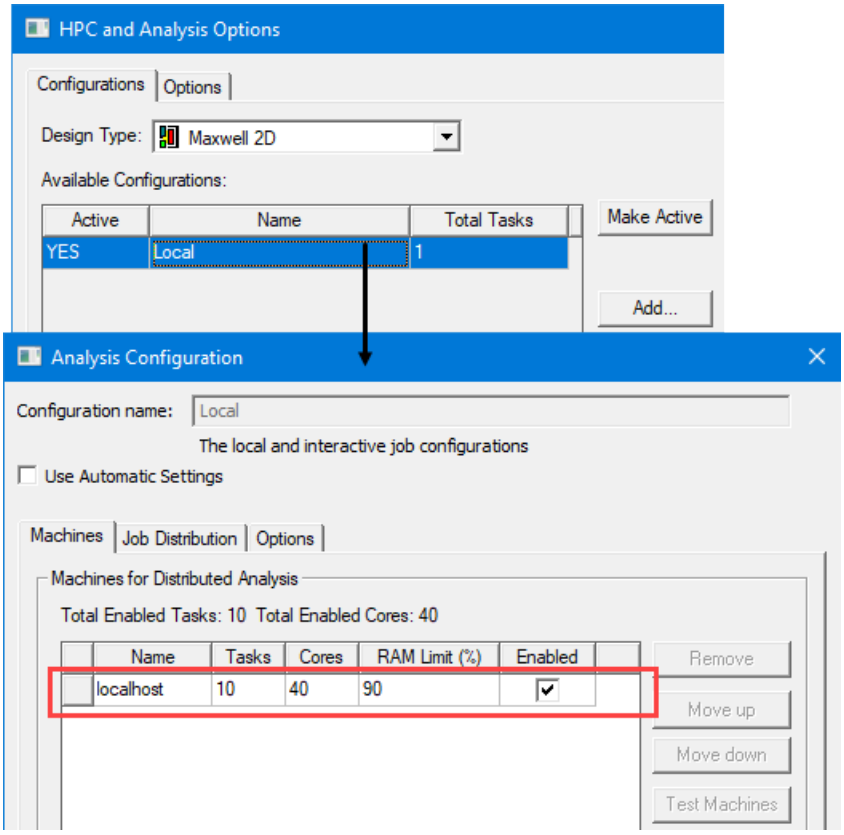
1. Open the **Optimetrics**\ParametricSetup window, and click **HPC and Analysis Options**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



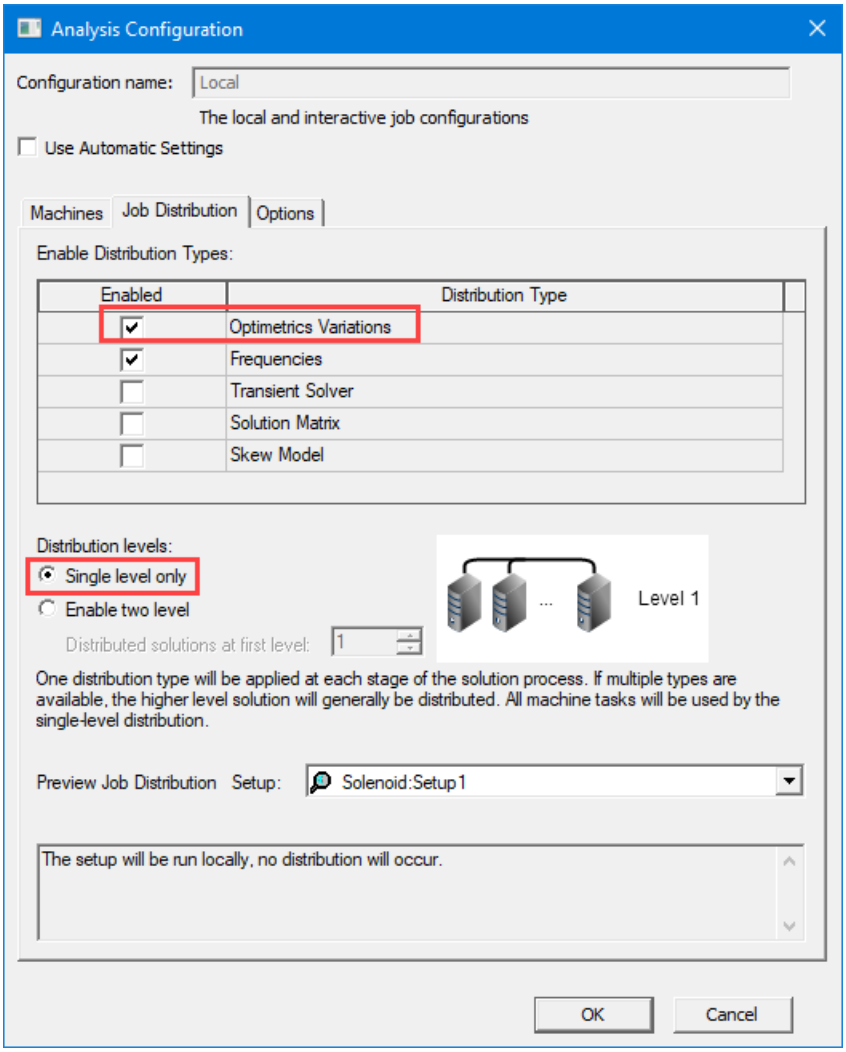
2. In the window that opens, double-click an active configuration to open the Analysis Configuration.
3. From the **Machines** tab, specify multiple tasks to run.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Click the **Job Distribution** tab.
5. From the Enable Distribution Types list, select **Optimetrics Variations**. And for Distribution, select **Single level only**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. In the Project Manager tree, right-click the Optimetrics\ParametricStudy, and select **Analyze**.

Defining Settings on the Solver Tab for Transient Solutions

To define solver settings on the **Solver** tab of the **Solve Setup** dialog box for transient solutions:

1. Enter a residual value in the **Nonlinear Residual** text box.

   To specify a time-dependent nonlinear residual, type in a function of TIME, such as **sin (TIME)**, or enter an expression that uses a dataset, such as **pwlx(*<name of dataset>*, TIME)**. A time-dependent variable can also be used.

   You can save the time-dependent expression as the default for the nonlinear residual by clicking the **Save Defaults** button on the **Defaults** tab of the **Solve Setup** dialog box. A

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

new solve setup will then use the user-saved default if the expression can be evaluated. A value of “0” will be used when the expression cannot be evaluated. For example, if the expression depends on a variable that is not yet defined in the design/project.

Changing the definition of the residual will NOT invalidate any existing solution; and the next simulation will continue from the stop time of the last simulation.

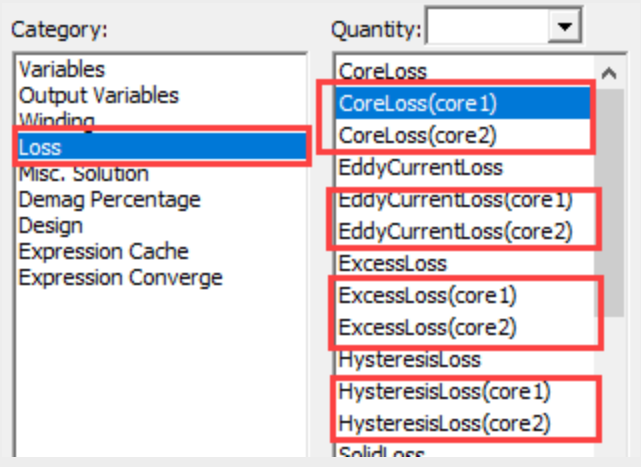
Changing the definition of a *variable* used for the residual value will NOT invalidate any existing solution. The next simulation will be for a different “design variation”.

Changing the definition of a *dataset* used for the residual will NOT invalidate any existing solution. The next simulation will continue from the stop time of the last simulation.

If a variable is used, you can choose to plot a family of curves based on the solved variations of the variable.

2. For both 2D and 3D Transient designs, you can select the **Smooth BH Curve** check box. This allows the solver to smoothen the list of points to make a refined B-H curve for better solving. If this option is not selected, the solver just uses the discrete points.
3. Select or clear **Output per object core loss**. When this check box is selected:
   - For steel material, the solver will output core loss for each object, as well as the individual components (hysteresis loss, eddy current loss, and excess loss).
   - For ferrite material, the solver will output only core loss.

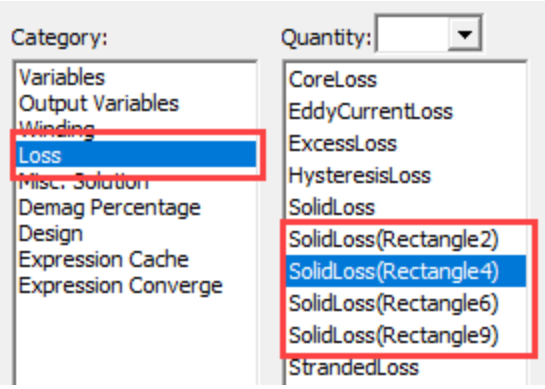
Note: To report core loss on objects, the objects must be selected in the **Set Core Loss** dialog box. The loss quantities for the core loss objects will show in the reporter.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

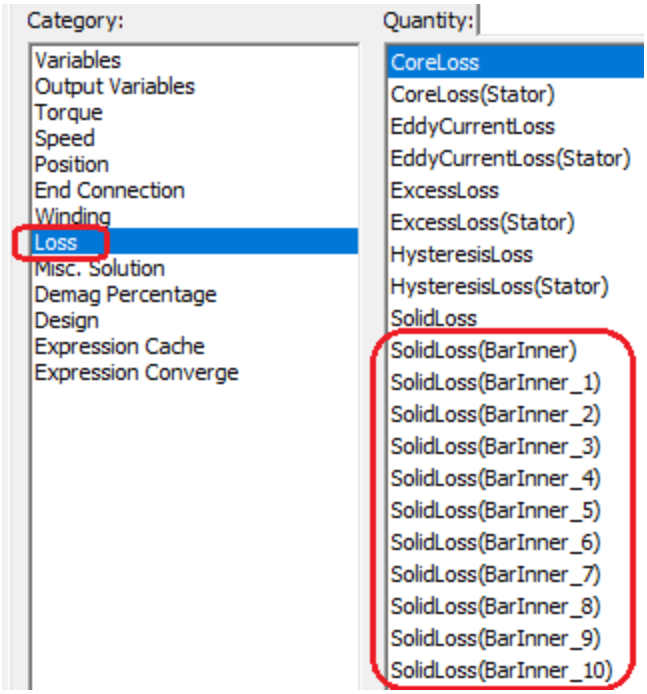
4. Select or clear **Output per object solid loss**. When this check box is selected, solid loss will be output for :

    - The solid loss quantities of the objects selected in the Set Eddy Effects dialog will show in the reporter. (Objects selected should not belong to stranded winding or stranded conductor.)
    - Coil objects in solid windings.
    - For 2D designs:
        - Solid loss quantities of the coil objects assigned with the solid Current Excitation will show in the reporter.

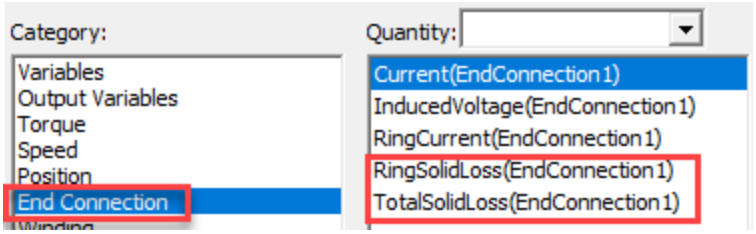


        - Each bar in an end connection will be considered as a separate object for solid loss output.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- All ring segments in one end-connection will be combined together as one virtual object for solid loss output.



If there are N end-connections (such as double cages), there will be N virtual objects.

5. Select or clear the **Output error** check box. When this check box is selected, the transient solver generates the output error data, which is then sent back to the desktop. The data can be viewed or used in the post processor, just as other transient solution data (such as power loss or winding).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** Output error is the percentage error in the total energy of the solution. This is a measure of the accuracy of the finite element approximation and can be used to judge if the mesh is good enough. Output error is calculated by finding the residual energy (the left-over energy calculated from the field after subtracting out the sources) divided by the total energy including the sources. Computing output error uses significant computation time.

6. For 3D Transient analysis only, select the desired **Scalar Potential** shape function: either **Second Order** (default), or **First Order** from the drop-down menu. You can also select the desired **Scalar Potential** on the **Solver** tab in the Properties window.

   Use of the First Order scalar shape function, rather than the Second Order shape function, may be desirable for any of the following cases:

   - If computational time is a main concern and solution accuracy is secondary. This might be useful for the initial design stage, before switching to using the more time consuming, and more accurate, second order scalar shape function.
   - If the contour of the geometry is very complicated, it may not be possible to use the higher second order shape function due to the inability of relatively larger element size to properly represent the original geometry shape
   - If the gradient of the magnetic field changes sharply, it may not be possible to use the higher second order shape function with large element size. Using a larger element size may not be able to adequately model the large change of material property with large element size, and nonlinear convergence may be difficult to achieve.

7. For 2D and 3D Transient designs, you can select the **Number of Nonlinear Iterations** check box. This provides an option for you to set both a **Minimum** iteration number to avoid non-converged solution, and a **Maximum** iteration number to avoid taking too much computation time. These values must be integers greater than 0. By default the values for **Minimum** and **Maximum** are 1 and 100, respectively. Maximum number should be greater than the Minimum number.

8. For 2D Transient analysis only, select the algorithm for the **Time Integration Method** using the drop-down menu.

   - **Backward Euler** – Default
   - **Runge-Kutta** – third-order algorithm for greater accuracy

9. Select or clear the **Fast Reach** check box.

   For transient simulations whose winding excitations are an AC voltage source, the DC flux linkage component may take a very long time to decay, especially for a device with a large time constant. Selecting **Fast Reach** adds an additional voltage component to the original

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

voltage definition during the first half-cycle to quickly eliminate the DC flux linkage, thus speeding up the solution process. You must also provide a **Frequency of Added Voltage Source** value that is greater than zero. The default value for **Frequency of Added Voltage Source** is 60 Hz. This frequency value is applied to all winding voltage sources.

> **Note:**
>
> - This speedup capability is only applicable to voltage sources whose integration of voltage quantity over one period is zero. This means the DC component of all voltage sources is zero.
> - This feature is intended to be used in conjunction with windings of the type "voltage". If the winding is fed by an external circuit, a similar functionality is available using the alternating flux model defined in the properties of sinusoidal voltage sources. Alternatively, the dedicated **AC_Model** component can be applied if an external circuit is feeding the winding.

10. Select or clear the **Auto Detect** check box.

    This option enables automatic detection of when steady state is reached based on a check of the flux linkage. When this option is selected, you must provide both a **Frequency of Added Voltage Source** value that is greater than zero (default value is 60 Hz), and a **Stop criterion** value that is greater than 0 and less than 1 (default value is 0.005). The smaller the **Stop criterion** value, the longer it will take to auto detect when steady-state is reached. For additional information on auto detect operation, refer to Automatic Detection of Reaching Steady State for Transient Simulations.

11. As needed for your design, select or clear settings in the **Time Decomposition Method Option** panel.

    The Time Decomposition Method (TDM) is an HPC distributed analysis type based on domain decomposition along the time-axis, (rather than the normal geometry division) to parallelize the transient solution. Instead of solving Maxwell transient problems sequentially for each timestep, TDM enables timesteps to be solved simultaneously in parallel. Thus, TDM can be implemented on distributed memory parallel platforms based on MPI. TDM has very good scalability, resulting in significant speedup for both 2D and 3D transient solutions.

    > **Note:** Maxwell 3D A-Phi transient solver supports only General Transient TDM.

    If you have enabled the **Use Automatic Settings** option in the **Analysis Configuration** dialog box, or if you have enabled the **Transient Solver** distribution type in the **Analysis Configuration** dialog box, you can select one of the following:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Transient	For general transient (Non-Periodic) TDM applications, select **General Transient**. Using this method, TDM solves a collection of many timesteps simultaneously.
Periodic	The **Periodic** TDM method intended for use in steady-state simulation. In such a case, all the setup and solutions at the first timestep will be the same as those at the last timestep of one period. Setup includes source, boundary, and position in electrical degrees. The solutions include all electrical and magnetic quantities. To use this method, all current in windings and all induced eddy currents in the conducting region should have the same time period. For example, an induction machine cannot be solved using the Periodic method because the time period of the currents in the stator winding is different from the time period of induced eddy current in the rotor bars. Periodical TDM is computationally very efficient if a problem is applicable. But because all timesteps over one period have to be solved simultaneously (cannot be divided into subdivision as the case for general TDM transient), the memory requirement may prevent users from using periodical TDM. To this end, the support of half-periodical TDM is able to effectively cut memory usage to roughly half. You can also select a **Half-periodic** option. This TDM option can cut your computer's memory usage when solving Maxwell transient problems. If all physical quantities satisfy anti-periodic condition along Time axis, a problem can be solved using this option so that memory usage can be cut to almost half. Also, since the total timesteps to be solved are halved, the total computation time is also cut roughly by half.

12. For **2D** designs, the **Thermal Two-Way Coupling\Relaxation** setting can be used to improve two-way coupling convergence by applying the relaxation factor to the temperature field. When the thermal modifier takes on a nonlinear form, the convergence of the solution field in Maxwell's equations may fail during two-way coupling iterations. With nonlinear iterations, using an under-relaxation factor can mitigate the risk of overshooting in the property updating process.

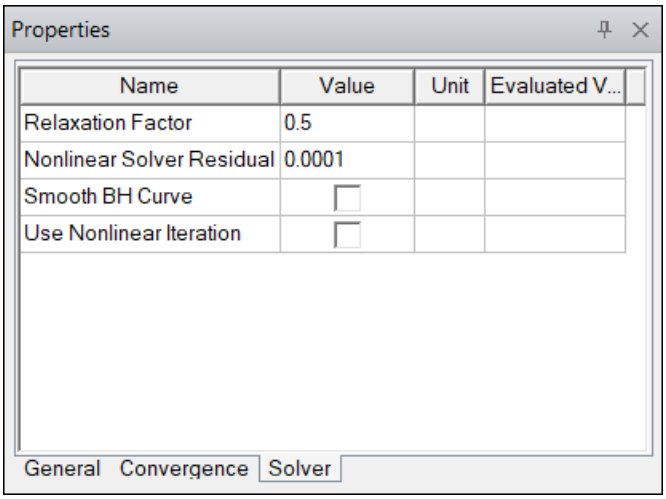
    The range of the relaxation factor is from 0 to 1. In general, a smaller relaxation value helps convergence, but may increase the total number of iterations. The default relaxation factor is 1.0.

    The **Thermal Two-Way Coupling\Relaxation** setting is applicable only when **Enable Feedback** is checked in Temperature of Objects window, which invokes two-way coupling instead of one-way coupling. The relaxation setting is

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- applicable to all Maxwell 2D designs,
- applicable to both geometry modes (Cartesian XY and Cylindrical about Z),
- does not support non-intrinsic and intrinsic variables.

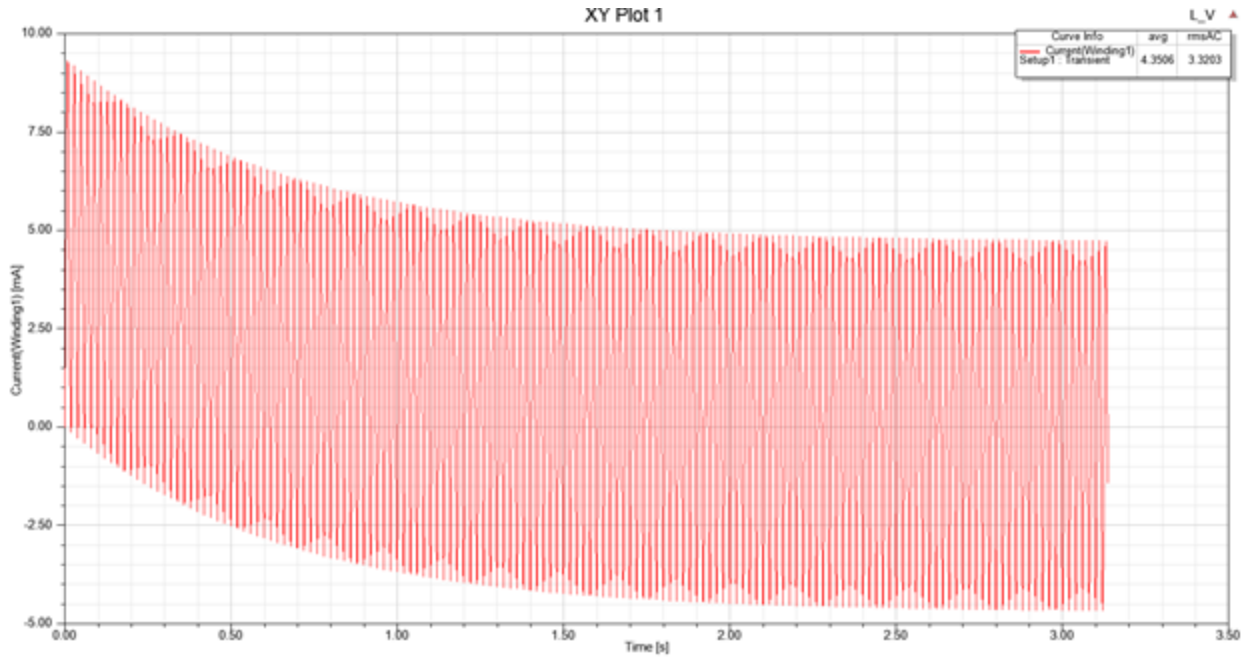
If activated in the Solve Setup window, the Relaxation Factor is also available in the Solve Properties window:



Automatic Detection of Reaching Steady State for Transient Simulations

For transient simulations, when the time constant of the design is large, many cycles may be needed to reach steady state.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Because it is often difficult to predict how many cycles are needed to reach the steady state, the user typically must preset a very long simulation time range (a large stop time) to ensure that steady state can be reached before the end of the simulation. This can result in the waste of significant simulation time for cases in which steady state is reached long before the set stop time.

Enabling the **Auto Detect** setting using a preset **Stop Criterion** causes the solver to check whether steady state has been reached during the simulation, automatically stopping the simulation when steady state is detected. This allows the user to set a very large stop time without concern for wasting needless simulation time, thus improving the efficiency of the simulation without losing the accuracy of the solutions.

Design Considerations

All windings in a project are divided to three categories: AC winding, DC winding, Non-applicable winding.

- For an AC winding, the winding source is periodic with the same frequency as the user-set frequency (**Frequency of Added Voltage Source**) and it has no DC component. A winding which is connected to an external circuit is treated as AC winding.
- For a DC winding the value of the winding source doesn't change throughout the simulation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- A non-applicable winding is a winding which cannot be included in either of the first two categories, such as the case when the frequency of the source is different from the frequency set by users.

If a project only includes AC windings and DC windings, automatic detection will be conducted by checking the flux linkages of all AC windings. If a project includes a Non-applicable winding, the automatic detection feature will be disabled.

How Auto Detection Works

Winding information is checked during simulation to determine whether steady state has been reached. If rotational motion is included in the project, the torque is also checked.

Of the three winding parameters that can be used to check whether steady state has been reached during the simulation: *current*, *induced voltage*, and *flux linkage*, **Auto Detect** uses flux linkage for the following reasons:

- Compared with the induced voltage, which is the time derivative of the flux linkage, the flux linkage itself is less sensitive to numerical noise.
- Similarly, current calculations are not as smooth as flux linkage. Also, when the winding excitation is current, the current is not applicable for detecting steady state.

The automatic check for steady state is carried out at the end of each cycle. The cycle time, T is calculated based on the input frequency (**Frequency of Added Voltage Source**) supplied by the user: $T=1/f$.

For example, if the check is carried out at time instant t_c, the flux linkage values over the last cycle $[t_c–T, t_c]$ and the second last cycle $[t_c–2T, t_c–T]$ will be used. The first checkpoint, t_c, is at $3T$ since, in general, the first cycle is unsuitable for checking for steady state – especially when **Fast Reach Steady State** is enabled (in which case the first cycle is not even a physical result).

The DC component and the fundamental component of the flux linkage of the two cycles are compared to check for steady state. These components are calculated as follows:

DC component:

$$D = \frac{1}{T}\int_0^T f(t)dt$$

Fundamental component:

$$F = \frac{1}{T}\int_0^T f\left(t\right)e^{-2\pi j\frac{t}{T}}dt$$

Assuming the DC component of the flux linkage at the last cycle is D_1, and the fundamental components are denoted by F_1 and F_2 at the last and the second last cycle, the following two criteria are implemented to check for steady state:

$$|D_1| < r$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$|F_1 - F_2| < r$$

where the reference value r is calculated by

$$r = |F_1| \times c$$

Coefficient c is the **Stop Criterion** for reaching steady state and is set by the user on the solver tab for transient solutions and must be greater than 0 but less than 1 (default = 0.005). Thus the smaller the coefficient is, the stricter the stop criterion for reaching steady state is, and the longer it will take to auto detect when steady state is reached.

If rotational motion is involved, after the flux linkage has converged, the solver will further check whether the torque has reached steady state. In steady-state, the torque should be constant with small deviation. Such deviation can come from mesh noise, cogging torque, slot effects, etc. This deviation should be small compared to the rms value of the torque. To check whether the torque has reached steady state, the solver compares the difference of the average torques in the last two cycles. The difference should be smaller than the tolerance value. (The tolerance value is normalized by the rms value of the torque in one cycle.)

The average value of the torque and the rms value of the torque in one cycle are calculated as follows:

The average value of the torque is

$$DT = \frac{1}{T}\int_0^T Tor(t)dt$$

The rms value of the torque is

$$RT = \frac{1}{T}\int_0^T Tor(t)^2 dt$$

The following criterion is implemented in the solver to check for steady state for torque:

$$|DT_1 - DT_2| < RT \times 10c$$

As above, *c* is the user-supplied **Stop Criterion**.

> **Note:** Because torque is usually noisier than flux linkage, the tolerance of the torque check is looser than the tolerance of flux linkage.

Time Decomposition Method for Maxwell Transient Designs

If a Maxwell 2D or 3D transient design problem is too large to solve efficiently on one machine, Maxwell can use the **Time Decomposition Method** (**TDM**). The Time Decomposition Method (TDM) is an HPC distributed analysis type based on domain decomposition along the time axis (rather than the normal geometry division) to parallelize the transient solution. Instead of solving Maxwell transient problems sequentially for each timestep, TDM enables timesteps to be solved

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

simultaneously in parallel. Thus, TDM can be implemented on distributed memory parallel platforms based on MPI. TDM has very good scalability, resulting in significant speed up for both 2D and 3D transient solutions.

> **Note:** Maxwell 3D A-Phi transient solver supports only General Transient TDM.

Maxwell supports two time decomposition methods:

- **General Transient (Non-Periodic)** – This method is intended for general transient applications.

  For this method, TDM solves a collection of many timesteps simultaneously – a collection of many timesteps being defined as one time subdivision. For example, if the end time of the entire transient simulation is 4 seconds, and if we want to solve all time-step solutions together for every 1 second, then this 1 second is defined as one time subdivision, and the total time subdivisions is 4. For the same problem, we can also define every 2 seconds as one time subdivision. Accordingly, the total time subdivisions becomes 2.

- **Periodic** – This method is intended for use in steady-state simulation. In such a case, all the setup and solutions at the first timestep will be the same as those at the last timestep of one system period. Setup includes source, boundary, and position in electrical degrees. The solutions include all electrical and magnetic quantities. To use this method, all current in windings and all induced eddy currents in the conducting region should have the same time period. For example, an induction machine cannot be solved using this method because the time period of the currents in the stator winding is different from the time period of induced eddy current in the rotor bars.

  You can also select a **Half-periodic** option if all physical quantities satisfy anti-periodic condition along Time axis. In this case, the memory usage can be cut to almost half.

Periodic TDM has the advantage that the problem just needs to be solved over one period of time, while **General Transient** TDM uses many cycles to reach steady state. But to enforce periodical condition along the Time axis, all timesteps over one period of time must be solved together, which may require extensive memory usage.

Using TDM

To use TDM, you must do the following:

- Have and use the HPC license Workgroup option (Refer to Setting HPC and Analysis Options).
- Set the total number of Tasks, N, (at least three Tasks) for the solve workgroup. The actual number of Tasks for parallel computation is N-1 (dependent tasks) because the first Task is used as the independent task, which is responsible for task assembly, dependent task control, synchronization, etc.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Enable the **Transient Solver** distribution type in the **Analysis Configuration** dialog box (Refer to Distributed Analysis Configuration).

Note: In Maxwell 2D, for **General Transient** TDM, the expression cache will be updated every 10 subdivisions, that is to say, every (task number - 1)*10 timesteps. For **Periodic** TDM, the expression cache is updated after the entire solution is completed.

Note: If the model contains nonlinear material definitions, the default nonlinear residual setting in the Solution Setup window may need to be reduced to achieve a similar BH curve convergence as the sequential computation of time steps. The reduction factor required can depend on the excitations defined in the model. Reduction factors between 1e-1 (for smooth excitations) and 1e-6 (for complex excitations, such as PWM excitations) are commonly used.

Limitations to Using TDM

- Mechanical transient is not supported. This means all moving positions for every timestep have to be known in advance so that FEA will take place at that predetermined position.
- The solution at a timestep cannot depend on the arbitrary history of previous timesteps (allowed for previous couple of timestep solutions required due to the time integration scheme itself). Traditional core loss computation is supported because core loss computation is done in post-processing.

  Strictly speaking, TDM cannot support hysteresis modeling. But for soft hysteresis materials, especially for lamination core loss computation, the hysteresis loop, or the coercivity Hc, is very small. In such a case, the impact of hysteresis on the nonlinear operating point can be ignored. This means we can decouple the nonlinear iteration process with consideration of hysteresis impact. For the first step of the nonlinear iteration process, distributed parallel TDM can still be applied. After the nonlinear iteration has converged, that is, the nonlinear problem has been linearized for all timesteps (or all timesteps in the sub-division), the second step is applied to take into account the impact of hysteresis. In such a case, the solution process is sequential – to consider the impact of hysteresis behavior from the previous time-step solution on the solution at current timestep. It can be expected that the results with TDM enabled and with TDM disabled might differ slightly due to ignoring the impact of hysteresis on the nonlinear operating point. However, from a practical application point of view, the solutions can be considered as sufficiently accurate –especially for lamination core loss computation.
- The purpose of the demagnetization process in a transient simulation is to find the worst operating point during the entire transient simulation. After a new worst operating point is

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

discovered, a linearized representation of the permanent magnet characteristic will be used to construct a new recoil line, which is applied to the simulation for subsequent timesteps. Thus, strictly speaking, TDM cannot support demagnetization modeling because all timesteps (or all timesteps in a subdivision) are solved simultaneously. However, as a reasonable approximation, we can solve all timesteps in one subdivision simultaneously based on distributed parallel TDM – and then search for the worst operating point among the solutions of these just-solved timesteps. Thereafter, this new worst operating point is used to construct a new recoil line, which is applied to all timesteps in the *next* subdivision. This means the impact of previous solutions on subsequent searches for the worst operating point is taken into account subdivision-by-subdivision, rather than timestep-by-timestep.

It can be expected that the results of the discovered *final* worst operating point might differ slightly between TDM- enabled solutions and TDM-disabled solutions. However, from a practical application point of view, the solutions can be considered as sufficiently accurate. This error can be reduced with the use of few timesteps in one subdivision.

Defining Settings for Output Variables for Transient Solutions

Any output quantity or expression within either the **Transient Report** or **Field Report** type can be defined as an output variable in the **Output Variables** dialog box, accessed via **Results** in the Project Manager tree.

To add an output variable to output with time:

- Click **Add** to select the output variables to be added to the list for output with time. The newly defined variable is added to the list.

> **Note:** The **Add** button will not be enabled if there are no variables defined in the project.

To delete an output variable:

- Select the variable from the list, and click **Delete**.

Do one of the following in the **Evaluation Timestep** section on the **Output Variables** tab of the **Solve Setup** window:

- Select **Using Constant**, type a value, and select the units. Select this option if you want to use a constant value for when you calculate output variable values.
- Select **Evaluate every Nth timestep**, and type a value. Select this option if you want to evaluate the output variable after a certain number of solution timesteps, that is, instead of calculating the output variable every fixed time interval, you want to calculate it every

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

certain number of timesteps. For example, if you want to evaluate the output variable every third timestep, type **3** in this text box.

Defining Settings on the Defaults Tab for Transient Solutions

Default analysis parameters are used to initialize the data for new setups.

To set up the default analysis parameters, define the following parameters under the **Defaults** tab of the **Solve Setup** window:

1. Click **Save Defaults** if you want to save the defaults based on the data for the current setup.
2. Click **Revert to Standard Defaults** to clear the existing values.
3. Click **OK**.

Setting Analysis Parameters for Electric Transient Solutions

When you set up an electric transient analysis, define the parameters on the following tabs of the **Solve Setup** window (which appears when you click **Maxwell** > **Analysis Setup** > **Add Solution Setup**):

- General (Name, Transient Setup)
- Expression Cache
- Solver
- Defaults

Defining Settings on the General Tab for Electric Transient Solutions

The following settings can be defined on the **General** tab of the **Solve Setup** dialog box for transient solutions:

- Setup Name
- Timesteps Setup

Setup Name for Electric Transient Solutions

By default, **Setup1** appears as the name of the first analysis you set up.

To change the name of the setup, type the new name in the **Name** text box on the **General** tab of the **Solve Setup** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To disable the setup, uncheck the **Enabled** check box. Refer to Disabling and Enabling an Analysis Setup for additional information.

Timestep Setup for Electric Transient Solutions

To set up a transient analysis, define the following parameters on the **General** tab of the **Solve Setup** window:

1. Enter the **Stop Time**, which is the value at which the solution will stop, and select the units.
2. Enter the **Initial Timestep**, which is the time step used to start the simulation and begin the adaptive process.

   > **Note:** The solver calculates the initial time step based on relaxation time Tau=Eps/Sigma of each material and then compares the results with the initial time step defined by the user. The smaller value is used.

3. Also enter the **Maximum Timestep** to set the allowable range the adaptive process use to alter the timestep.
4. Check **Save Fields** to enable saving the fields data.

Defining Expressions for Electric Transient Solutions

You can specify additional evaluation criteria through the use of expressions and output variables.

To set expressions as evaluation criteria:

1. Right-click on the Analysis icon in the Project Manager tree to open the **Solve Setup** window.
2. Click the **Expression Cache** tab and click the **Add...** button to open the **Add to Expression Cache** dialog box.
3. Specify the Context for the expression, including **Report Type**, and **Solution**.
4. Under the **Trace** tab, select from the **Category**, **Quantity**, and **Function**lists to create expressions.

   Selecting a listed category lists the Quantities and Functions available for each category. If you have defined one or more output variables, you can see them by selecting the **Output Variables** category. The **Output Variables** button opens a dialog box that lets you define output variables. See Specifying Output Variables.

   When you have created an expression, it appears in the **Expression** field of the **Trace** tab. If desired, you can use the **Range Function** button to select range functions to apply to the expression.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Under the **Calculation Range** tab, you can view the values of available sweep variables. Clicking the ellipsis **[...]** button in the **Edit** column opens a list of values from which you can select.

5. When finished defining the expression that you want to add to the cache, click the **Add Calculation** button.

   This adds the selected expression and the associated context to a table in the **Expression Cache** tab. You can define additional expressions with contexts and add them in the same way.

6. When finished adding expressions, click **Done** to close the **Add to Expression Cache** dialog box.

   The **Expression Cache** tab lists the expressions you have added in a table:

   - The **Title** field is editable, by default showing the name as built from the expression (with underscores removed).
   - The **Expression** field shows the full expression. You can resize the **Solve Setup** window, and also each column in the table, to view the contents of the fields.
   - The **Context** column for non-field types lists the parameter type specifying the force, torque or matrix setup. For these types the context also can be given as part of the expression. For example, the expression *Force1.Force_x* with a context of *None* is equivalent to an expression of *Force_x* with a context of *Force1*.

     For field quantities the **Context** can be the geometry on which to evaluate the field quantity - a point or line. If you use a line you need to use some range function to reduce the dimensionality to a single value (or use the Calculation Range tab to specify a single point along the line). Note that, if you create a named expression in the field calculator that already evaluates to a single value, then the **Context** here will likely be *None*.

   - The **Intrinsics** column shows a clickable button that opens an **Edit Calculation Range** dialog box. If the **Intrinsics** column button shows **None** you cannot edit the value. If the button shows variables, click the ellipsis **[...]** in the **Edit** column to display a list of the variable values from which you can select. Click **OK** to close the **Edit Calculation Range** dialog box and apply your selections to the **Expression Cache**.

     The **Intrinsics** field lists the values you may have set in the **Calculation Range** tab when you added the expression. This is used to set the values of “other” sweeps. Usually there are no other sweeps because the values of the variables are set in the definition of each variation. In some cases you may need to specify the values of “other” sweeps. For field values with a line for the context, you can specify the (normalized) distance along the line For eddy current field quantities you can also specify the phase for the calculation.

7. In the **Evaluation timestep** panel, set how often the solver is to output field solutions. Select **Constant** to output solutions at a preset time interval (e.g., every 0.1 s); or **Number**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of solve steps to output solutions each preset number of timesteps (*e.g.*, every 3 timesteps).

8. Optionally, buttons allow you to **Edit** and **Remove** selected expressions, and to **Remove All** expressions in the table.
9. When finished defining expressions, click **OK** to close the **Solve Setup** window.

Defining Settings on the Solver Tab for Electric Transient Solutions

To define solver settings on the **Solver** tab of the **Solve Setup** window for transient solutions:

1. Enter a value in the **Temporal Tolerance** text box.
2. For the Initial Condition, select either a preset **Value** or a **Static Field**.

   If **Static Field** is selected, the **Setup Link** button becomes active, and the **Setup Link** dialog box automatically appears, and the field will be imported from the source design to the target design.

3. The **Import mesh** check box can be selected in the "target" design if the mesh for the same geometry is to be used in the analysis.

   If the **Import mesh** feature is selected, the **Setup Link** button becomes active, the **Setup Link** dialog box automatically appears, and the mesh will be imported from the source design to the target design.

   > **Note:** The usual way to create geometrically equivalent designs is by copying all geometry objects from one design and then using **Import From Clipboard** in the other design.

Setup Link for Electric Transient Solutions

If you selected **Import mesh** or **Static Field**, do the following to specify the project, design, and solution containing the mesh information you want to use:

1. Click **Setup Link** in the **Solve Setup** window.

   The **Setup Link** window appears.

2. Click the **General** tab.
3. Select the project from the **Project File** drop-down menu.
4. Select the saved design from the **Design** drop-down menu.
5. Select the specific solution you want to link to from the **Solution** drop-down menu.
6. Click the **Parameters** tab.
7. Optionally, edit the **Value** and **Unit** columns for any of the parameters listed
8. Click **OK** to close the **Setup Link** window and return to the **Solve Setup** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining Settings on the Defaults Tab for Electric Transient Solutions

Default analysis parameters are used to initialize the data for new setups.

To set up the default analysis parameters, define the following parameters under the **Defaults** tab of the **Solve Setup** window:

1. Click **Save Defaults** if you want to save the defaults based on the data for the current setup.
2. Click **Revert to Standard Defaults** to clear the existing values.
3. Click **OK**.

Resetting Analysis to Time Zero

Before rerunning a transient simulation that has data associated with it, you need to reset the analysis to time zero; otherwise, Maxwell will not overwrite the data that is already present because it will not rerun the simulation. Resetting the analysis to time zero invalidates the current solution and will instruct the software to start over from the beginning when you rerun the simulation.

If you have changed simulation parameters, the transient analysis will rerun without the user having to select **Reset Time Zero**.

To reset the analysis, do one of the following:

- Click **Maxwell** > **Analysis Setup** > **Reset to Time Zero**.
- In the Project Manager tree, right-click **Setup** under **Analysis**, and select **Reset to Time Zero** from the shortcut menu.

Changing Memory Settings

The Maxwell solver has to make trade-offs between the solution time and the amount of memory used in the solution process.

The software automatically determines the amount of RAM installed on the computer, assuming that it can use up to 75% of this during the solution process. You can override this choice by changing the following two memory settings:

- Desired RAM Limit
- Maximum RAM Limit

Generally, set the RAM value as high as possible to obtain the fastest solutions.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These settings are not an absolute limit on the amount of RAM to be used. When solving small problems, Maxwell tries to respect this limit by choosing to run more slowly with a smaller amount of RAM. When solving a large problem, Maxwell may be unable to use less RAM than the specified limit. In this case, the amount of memory used exceeds the specified limit but tries to minimize how far it goes beyond.

Specifying the Desired RAM Limit

The **Desired RAM Limit** setting requests a restriction on the amount of physical random access memory (RAM) the MPS solver used in Maxwell may use before it must stop solving *on-core - solving processes entirely in RAM* - and start solving off-core. In *off-core mode*, Maxwell creates temporary solution files to which it spills, or shifts, data from RAM, instead of relying on the operating system to start disk swapping. The location of these temporary files is specified in the General Options (**Tools** > **Options** > **General Options** > **Directories**). The MPS solver is finely tuned at handling its own memory, and can optimize loading only those blocks of memory required for its immediate needs.

Using this option may help to keep the entire solver from being swapped out in the normal course of process management on your computer. This kind of control may be especially important when multiple solvers are running on the same machine. Of course, if the total memory requirement of all processes grows large enough, the operating system will be forced into disk swapping.

> **Note:** The Iterative solver doesn't go off-core for efficiency reasons. Because the Desired RAM Limit (the soft memory limit) is used for off-core run, it is not appropriate for the iterative solver. Instead, the iterative solver uses the Maximum RAM Limit (the hard memory limit). Once the limit is exceeded, the iterative solver will issue an "Out of memory" error message.

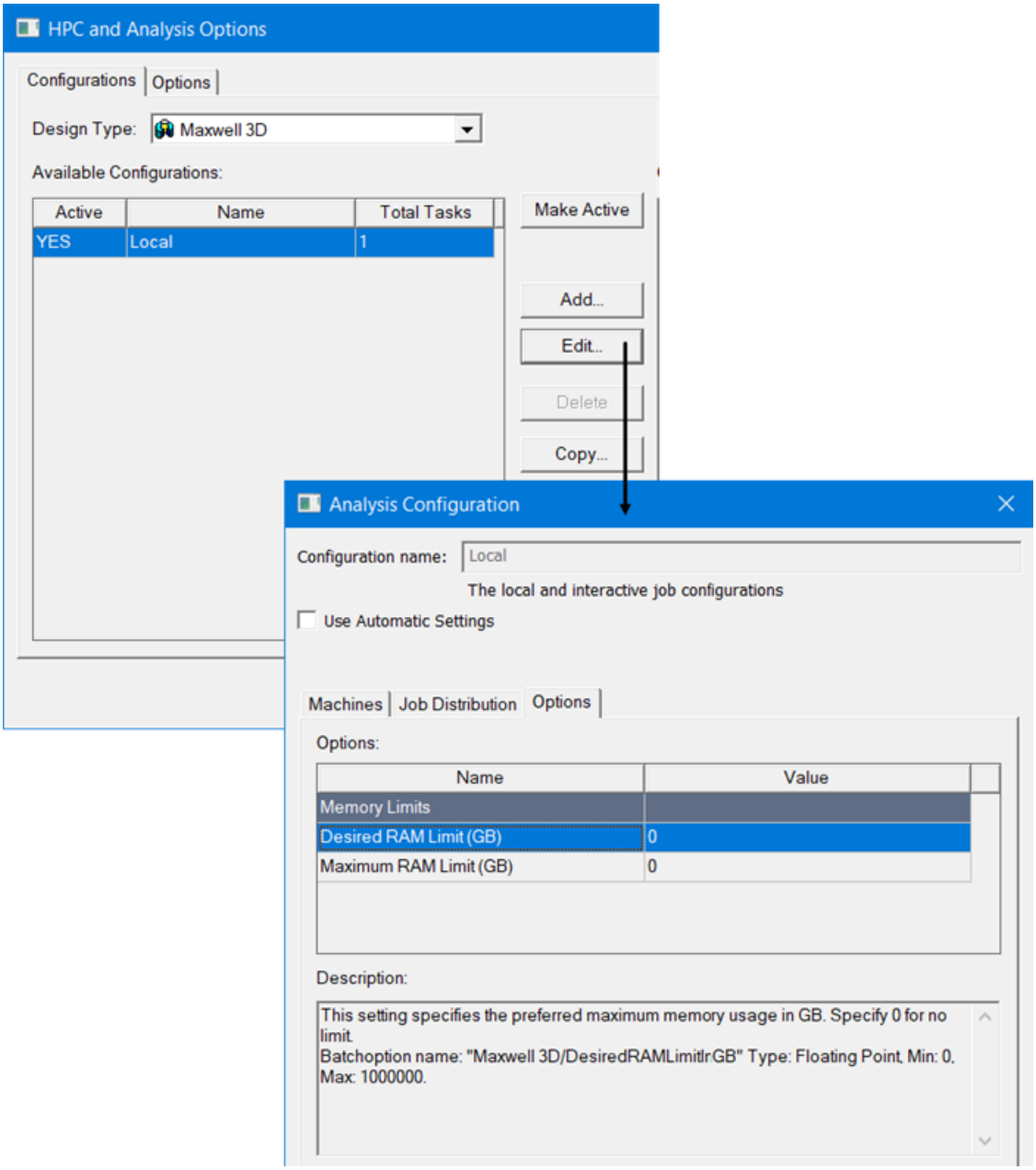
Regardless of this setting, processes are limited to 4TB of address space on 64 bit operating systems - no matter how much physical memory is installed.

> **Note:** Regardless of the **Desired RAM Limit** setting, if allocation fails, the Maxwell solver automatically switches to off-core mode.

In case you receive an error message regarding insufficient memory on a 64-bit operating system, you may have reached a point where the sum of physical RAM plus available swap space exceeds the minimum amount of RAM needed by the off-core solver. Even for the off-core solver, the RAM usage cannot be made arbitrarily small. In that case you can consider increasing the swap space (the virtual memory) in the settings of your system.

To specify the Desired RAM Limit of the machine on which Maxwell is installed, select **Tools** > **Options** > **HPC and Analysis Options**, then click **Edit**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



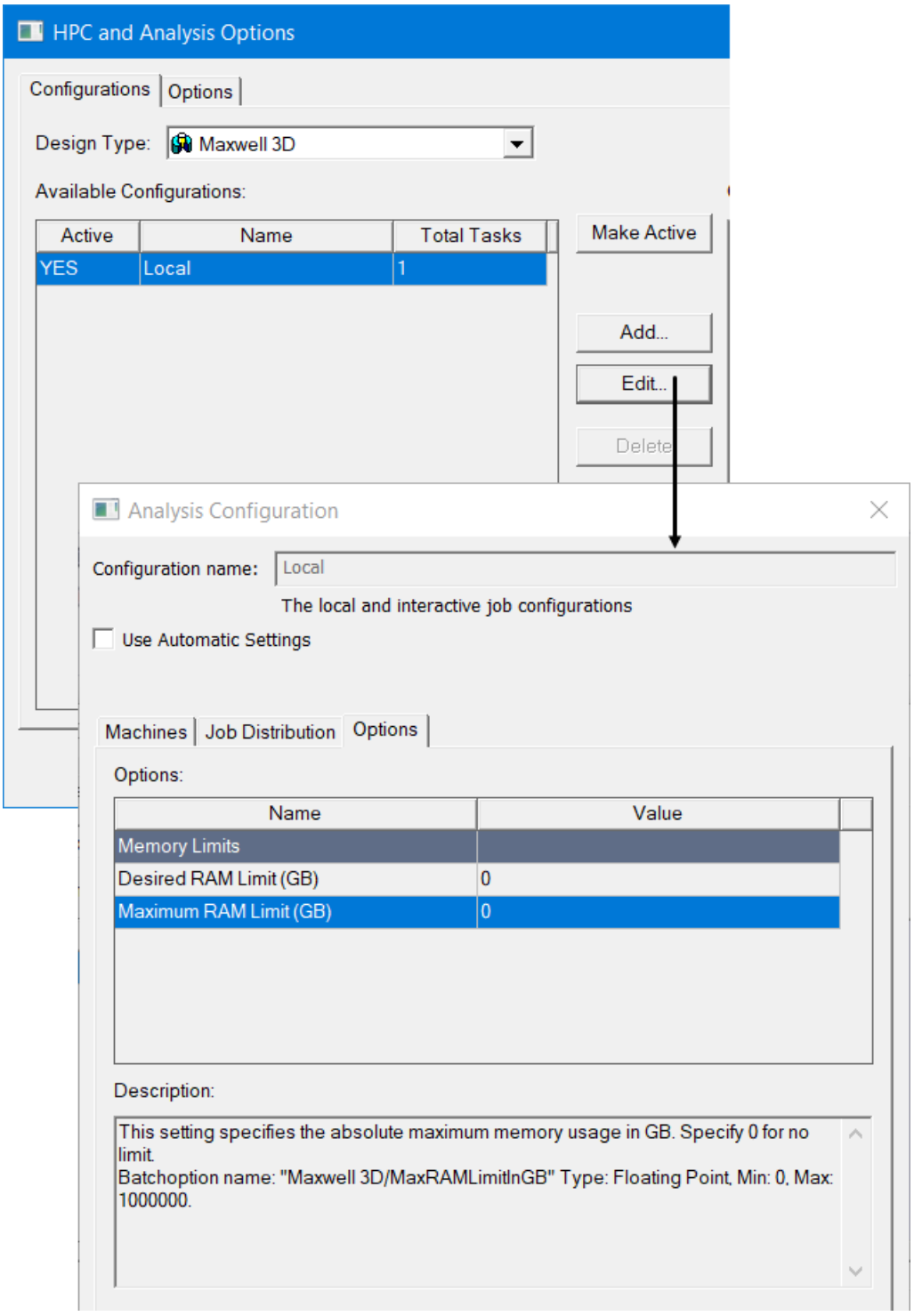
Refer to the *HPC and Analysis Options* section for more details.

Specifying the Hard Memory Limit

The **Maximum RAM Limit** setting specifies the absolute limit on the amount of physical memory that the solver can use. If the solver attempts to allocate more memory than this setting, the solution process terminates abnormally, and incorrect error messages may appear.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To specify the **Maximum RAM Limit** for the current user and machine, select **Tools** > **Options** > **HPC and Analysis Options**, then click **Edit**.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

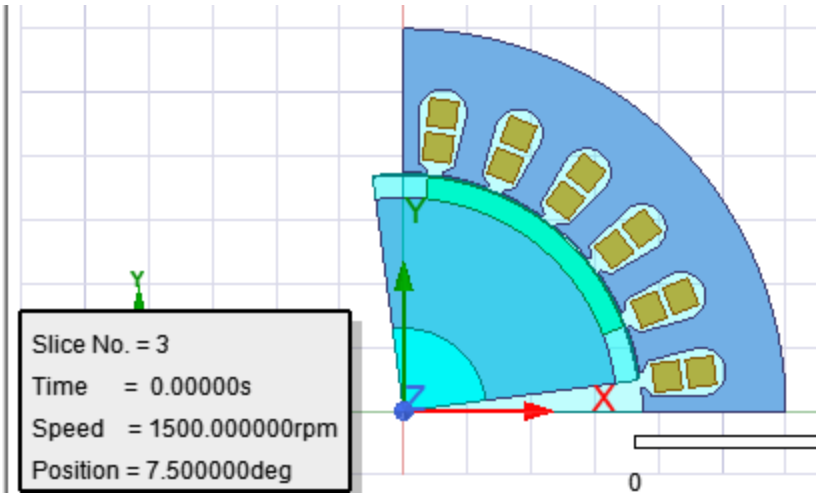
Refer to the *HPC and Analysis Options* section for more details.

> **Note:** Ansys recommends that you use the **Desired RAM Limit** setting if you want to limit the RAM the solver may allocate.

Setting View Context for Transient Solutions

To set the view context:

1. Click **View** > **Set Solution Context**.
2. In the **Set View Context** dialog box that appears, select a solution from the **Solution Name** drop-down menu.
3. Select a time instance from the **Time** drop-down menu.
4. For Maxwell 2D transient solutions only, in which **Use Skew Model is enabled on the Model Settings tab**, an additional **Slice** field displays, which allows you to select the skew model slice to be displayed in the modeler window.



5. In the **View** area of the dialog, you can choose the location to display the **Solution Context** within the modeler window. **Lower left** was selected for the image above.
6. The position and speed values may be displayed in the **Solution Context** by selecting the **Display Speed and Position values** check box.
7. Click **OK**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** You can also change the view context by double-clicking the solution context text label in the bottom left of the model window.
> **Example of text label showing solution view context:** Time='0s'

Speed and Position Display Settings

To set the display setting for speed and position in the view context:

1. Click **View** > **Set Solution Context**.
   The **Set View Context** window appears.
2. Select the **Motion View Format** tab.
3. In the **Format Speed Unit** section:
   a. Specify the **Unit Type** from the drop-down menu.
   b. Specify the **Field Width** for the display and the number of digits of **Precision** to display.
   c. If you prefer **Scientific Notation**, select the check box.
4. Click **OK**.

> **Note:** You can also change the view context by double-clicking the solution context text label in the bottom left of the model window.
> Example of text label showing solution view context: Time='0s'

Renaming a Solution Setup

Do the following to rename a solution setup:

1. In the Project Manager tree, under **Analysis**, right-click the setup you want to rename, and select **Rename** from the shortcut menu.

   The setup name text is highlighted in the Project Manager tree.

2. Type the new name for the setup, and press **Enter**.

You can also rename the solution setup by changing the text in the **Name** text box of the **Solve Setup** window.

Copying a Solution Setup

Solution setups may be copied and pasted within a design or across designs of the same type. This is beneficial for setups having a large number of parameters to specify such as an RMxprt design, or where minor changes to a setup are being evaluated.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** Setups cannot be pasted across solution types, across geometry modes such as from 3D to 2D, or across machine types in RMxprt.

To copy a solution setup:

1. In the Project Manager tree, under **Analysis**, right-click the setup you want to copy.

   A shortcut menu appears.

2. Select **Copy** from the shortcut menu.

   The setup parameters are copied to the clipboard.

3. In the Project Manager tree, right-click on the **Analysis folder to receive the copied setup**.

   A shortcut menu appears.

4. Select **Paste** from the shortcut menu.

The setup parameters are copied to the **Analysis** folder as a new setup.

Disabling and Enabling an Analysis Setup

To disable an **Analysis** setup definition without deleting it:

1. Expand the tree hierarchy under the Analysis icon in the Project Manager tree.
2. Right-click on the icon for the setup definition. In the shortcut menu that appears, click **Disable Setup**.

You can also disable an analysis setup by selecting **Properties** from the shortcut menu to open the **Edit Properties** window. In the upper right corner of the Edit Properties window, uncheck the **Enabled** box. Then click **OK**.

To reactivate an **Analysis** setup:

1. Expand the tree hierarchy under the **Analysis** icon in the Project Manager tree.
2. Right-click on the icon for the setup definition. In the shortcut menu that appears, click **Enable Setup**.

You can also reactivate the analysis setup by selecting **Properties** from the shortcut menu for the sweep to open the **Edit Properties** window. Check the **Enabled** box in the upper right corner, and click **OK** to apply the change and close the window.

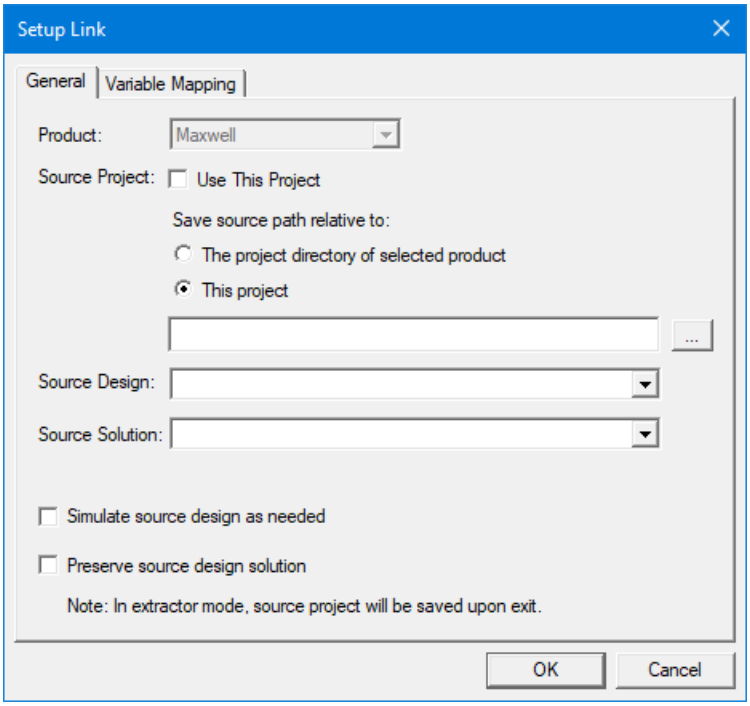
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup Link Window

The **Setup Link** window permits you to link the current project to another as described in the following topics:

- Setup Link for Non-Transient Solutions
- Import Mesh for Non-Transient Solutions
- Setup Link for Transient Solutions
- Import Mesh for Transient Solutions
- Setup Link for Electric Transient Solutions

The **General** and **Variable Mapping** tabs are common to all **Setup Link** windows.



Depending on the type of link being set up, other tabs such as **Additional Mesh Refinements**, and **Advance** may also appear. Details of the use of the various tab settings may be found in the topics linked above.

Clearing Linked Data

If you have previously set up links, the **Maxwell** > **Analysis Setup** menu contains an option to **Clear Linked Data**. This removes the linked data for all links in a target design, therefore invalidating the solutions. Clearing linked data for some link types requires Maxwell to revert to

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the initial mesh. Thus, in some cases, this command removes the current mesh of the target design.

Setting the Report Updating During Analysis Options

Updating numerous reports may take a significant amount of time. Updating reports during the analysis process can impact the overall time to solution. You may want to vary the times when your reports get updated relative to the impact on overall solve time.

Five options exist for updating reports during solutions:

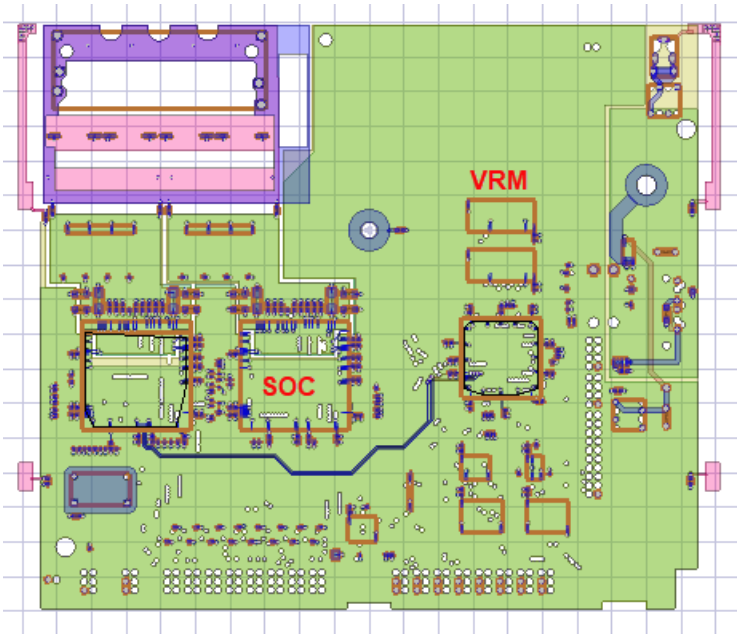
- **After each variation** – when performing an Optimetric or parametric analysis, all reports are updated after analysis of each variation has been completed.
- **Automatically** – the default. It means update most things immediately.
  For **AdaptivePass** plot context, plots are updated at the end of each solution pass. For **LastAdaptive** or **Transient**, the plot is updated at the end of the transient or adaptive solution.
  This option balances report and field plot updating with solution time. For example, reports may be updated after each adaptive pass but field plots will not be updated until the solution is complete.
- **Immediately** – update reports and plots as soon as data comes from the solver. This option will have the greatest impact on the overall solution time but will have the most rapid updating of reports and field plots. Caution should be used in selecting this option. Some types of reports and field plots may take a long time to update, especially as the mesh size increases.
- **Never** – only manual intervention updates reports. This option will prevent updates from impacting the solution time.
- **On Completion** – as with **Never**, but a single update is done when the solve completes. This option should be considered when using dynamic links to other products to provide best solution time with the final data available for the link.

To change the setting for the current project, see Desktop Performance options.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16 - Using a 3D Layout Component in Maxwell

HFSS 3D Layout designs can be imported into Maxwell as a component for simulation in a Maxwell 3D transient or AC magnetic A-Phi analysis. You can have a complex layout component interact with the surrounding 3D geometries without the burden of using a 3D geometry exported from HFSS 3D Layout. This feature allows great scalability in having a complex layout geometry sit in an EMI/EMC 3D environment.



The layout component can be imported from EDB to a Maxwell 3D design with a terminal solution type.

Supported functionalities for Layout Component in Maxwell 3D include

- Normal simulation workflow: Run simulation and view matrix and field overlay
- Layout geometry visualization and option to change object visualization attributes.
- Flex PCB Component support
- Visualize RLC elements in layout components the same way as they do in 3D layout, including RLC boundaries and components that are categorized as circuit elements.
- Parametric Analysis: Using design variable to control component
- Import of custom coordinate systems from layout component design to Maxwell 3D.

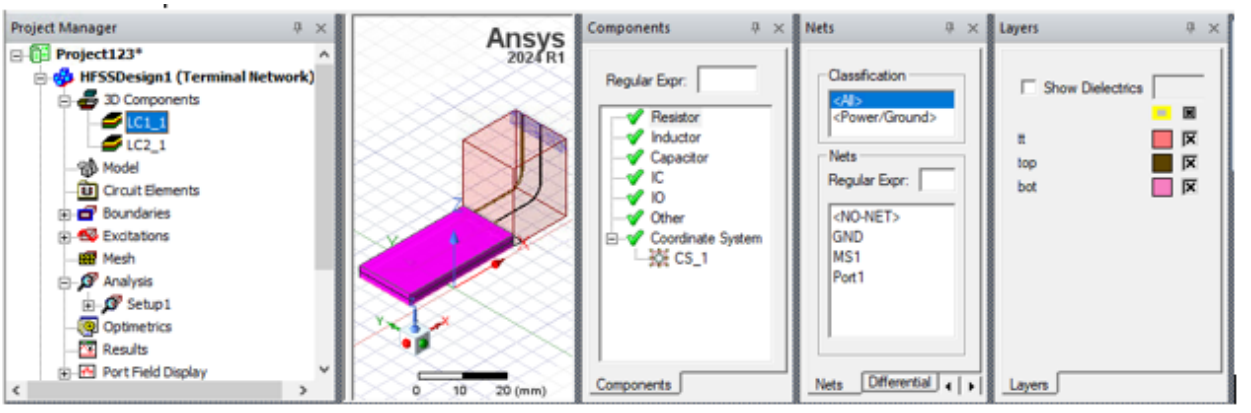
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Intersection Validation
  - Between MCAD geometry and layout component
  - Between different layout components
- Show Nets for Layout Components
  - Only MCAD nets are shown now, which may connect through ECAD conductors
  - Net attributes can be view through component object attributes
- Port Display
- Mesh Link
- Can use non-model sheet or cut plane to plot in layout component

Limitations:

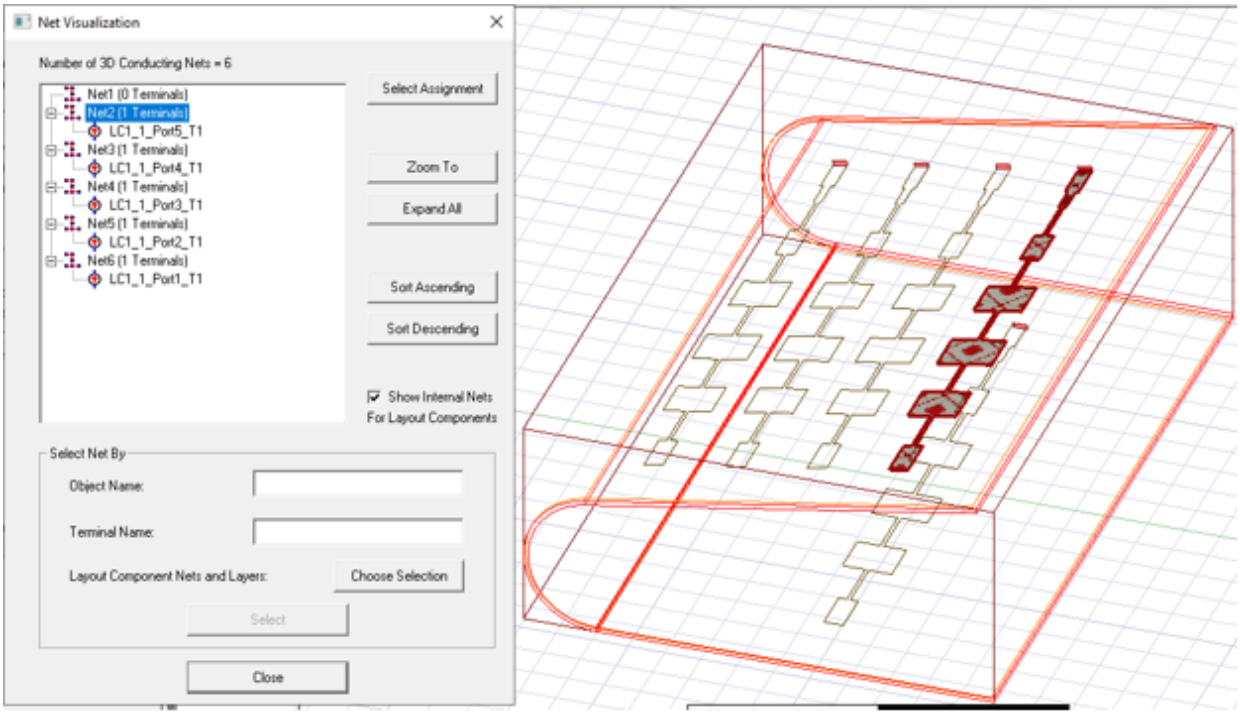
- Single top-design EDB only
- No Derivative support
- No Encryption

When a layout component is selected in the Project Manager tree or history tree, the Components/Nets/Layers tabs will display the information about the component in the same way as they do in a 3D layout design for that component.



Finally, in the **Net Visualization** dialog box, edge ports are correctly identified as part of the nets formed by ECAD and MCAD geometries.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Adding a 3D Layout Component

Use the following procedure to add a layout component to a Maxwell 3D design. These instructions assume that an HFSS 3D Layout of the PCB to be analyzed already exists.

> **Warning:** Make sure the 3D Layout file you are importing only has one design; otherwise, the import will fail.

1. In the Project Manager, right-click **3D Components** and from the shortcut menu, select **Browse Layout Component** > one of the following options:

   > **Note:** The Browse Layout Component is only available for a Transient or AC Magnetic A-Phi solution.

   - **Import .aedbcomp file**: Choose this option if you have exported an HFSS 3D Layout as a single-file Layout Component (*.aedbcomp). A *Select layout component file to import* dialog box appears in which you can browse to the file location and select and open it.
   - **Browse to project aedb folder**: HFSS 3D Layout design files are stored in a folder named <Project_Name>.aedb. If the source project contains more than one design, the design name will also be included (<Project_Name><Design_Name.aedb).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you select this option, the **Select aedb folder to import** dialog box appears. Navigate to the desired folder, double-click it to display its top-level contents, and click **Open** to select the folder.

- **Insert Existing EDB Definition:** This option will only be available if you have already imported at least one layout into the currently active mechanical design. Any such layouts will appear in a flyout menu, and you can click the one you wish to insert into the model.

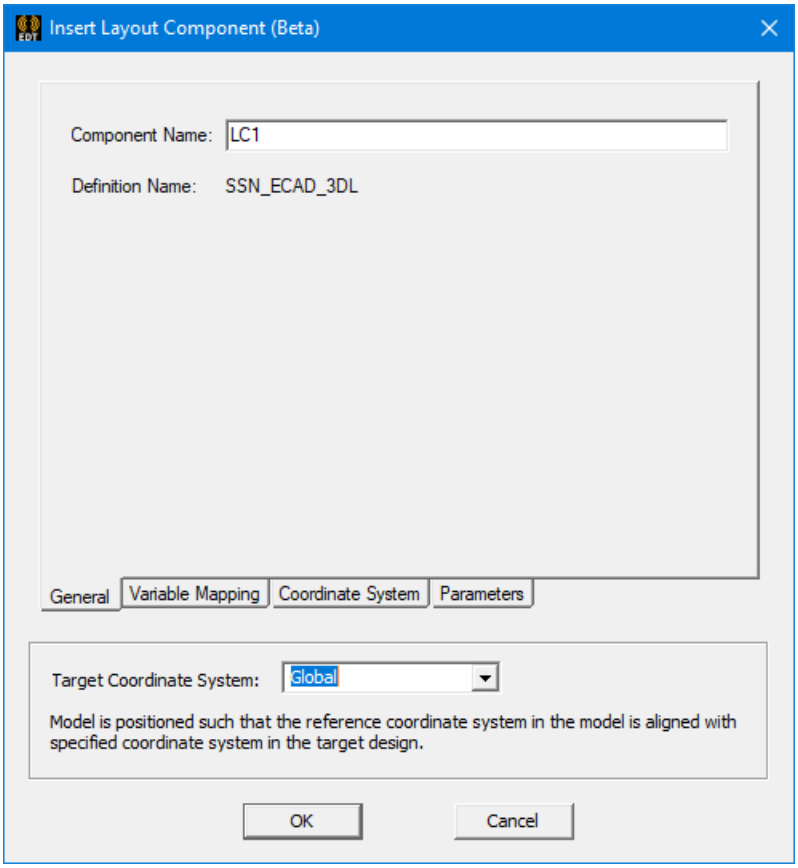
If a layout component has previously been added to the active mechanical design, the project name will be listed in the flyout menu, even if the component has since been deleted. You can select the project name from this flyout menu to re-add the component to the design. If you were to select it while the component already exists in the design, a duplicate layout component, overlapping the original one, would be added. Therefore, avoid doing so unless you want a duplicate component and plan on moving it to a separate location. You can also define a relative coordinate system (RCS) and choose to place the new component according to the RCS instead of dropping it on top of the previous layout component and then moving it.

> **Note:** Encrypted layout components cannot be imported into Maxwell.

> **Note:**
>
> Unused layout components that remain in the flyout menu can be removed by editing the *Components Library*. To do so, right click **Components** under *Definitions* in the Project Manager and choose **Edit Library** from the shortcut menu. Select one or more rows of the *Components* table and click **Remove Component(s)**. Click **Yes** to confirm removal for each selected component entry. Finally, click **OK** to exit the *Edit Library* dialog box.

The *Insert Layout Component* window appears with its **General** tab displayed:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Perform the following steps within the **General** tab:
    a. Optionally, change the **Component Name** from its default value (LC*x*, where *x* is an automatically incremented serial number).
    b. Choose the **Target Coordinate System** on which to base the layout component placement. Relative coordinate systems, defined in the target design prior to starting the *Browse Layout Component* process, appear in the drop-down menu in addition to the *Global* CS.

       The *Target Coordinate System* setting is visible regardless of which window tab is currently selected.

3. Select the **Variable Mapping** tab.

   What appears at this point depends on whether variables are defined in the source layout and are applicable to the current solution type. Choose one of the following two actions:

   - If there are no applicable variables in the source layout, no variables will be listed in this tab, and you can **skip to step 4**.
   - When applicable variables are present in the source layout, they are listed on this tab, and the following actions are possible:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

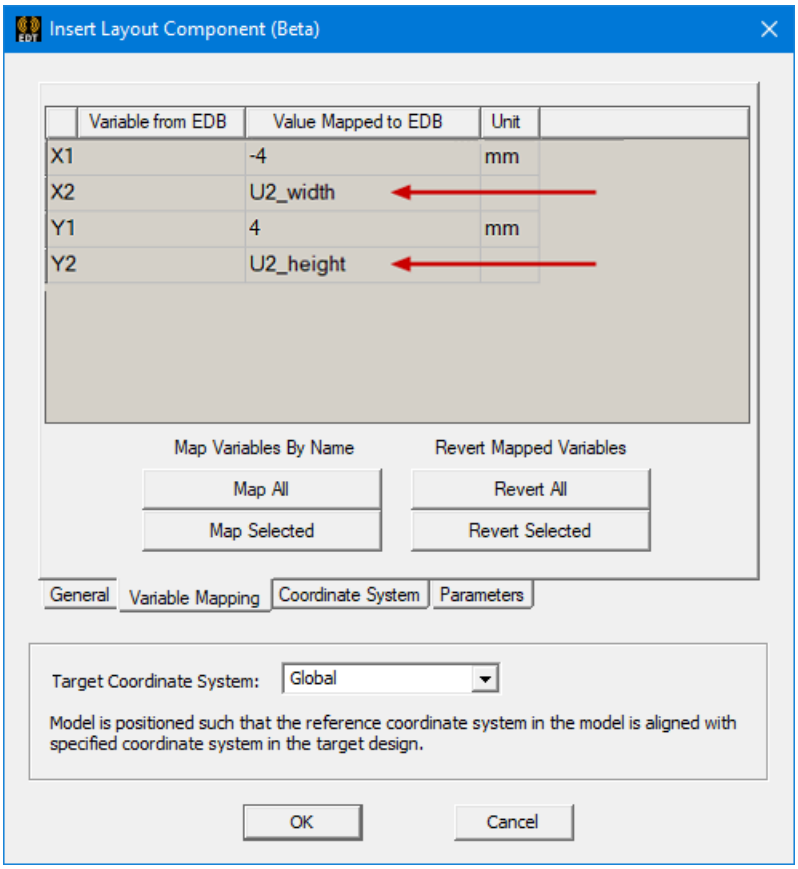
a. Optionally, you can **substitute a different numerical value** or **substitute variables** (instance parameter names) for the numeric values in the *Values Mapped to EDB* column. This substitution is particularly useful if you wish to include multiple instances of the layout component in a single design and have geometry variations between the instances. See Instance Variation for more information. Additionally, you can use variables to manipulate the geometry during an optimization or parametric analysis.

   In-place definition of new variables is supported. If you specify a previously undefined variable name, the *Add Variable* dialog box will appear when you press **Enter** or advance to a different table cell so that you can define its unit type, units, and initial value.

b. Alternatively, you can use the following buttons to automate the variable mapping task or to revert to the original values:
   - **Map All:** For all source layout variables, this command creates instance parameters, each with the same name and value as the source variable.
   - **Map Selected:** For selected layout variables, this command creates instance parameters, each with the same name and value as the source variable. Before clicking the *Map Selected* button, click a table row to select a variable to map. Ctrl+click to select multiple variables or to toggle the selection state of a row.
   - **Revert All:** For all variables, this command reverts the **Value Mapped to EDT** entry to the original source layout value.
   - **Revert Selected:** For selected table rows, this command reverts the **Value Mapped to EDT** entries to the original source layout values. Before clicking the *Revert Selected* button, click a table row to select a variable to revert. Ctrl+click to select multiple variables or to toggle the selection state of a row.

   In the following example, two of the four variables in the source ECAD database (X2 and Y2) have had instance parameters substituted for the original numeric values:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

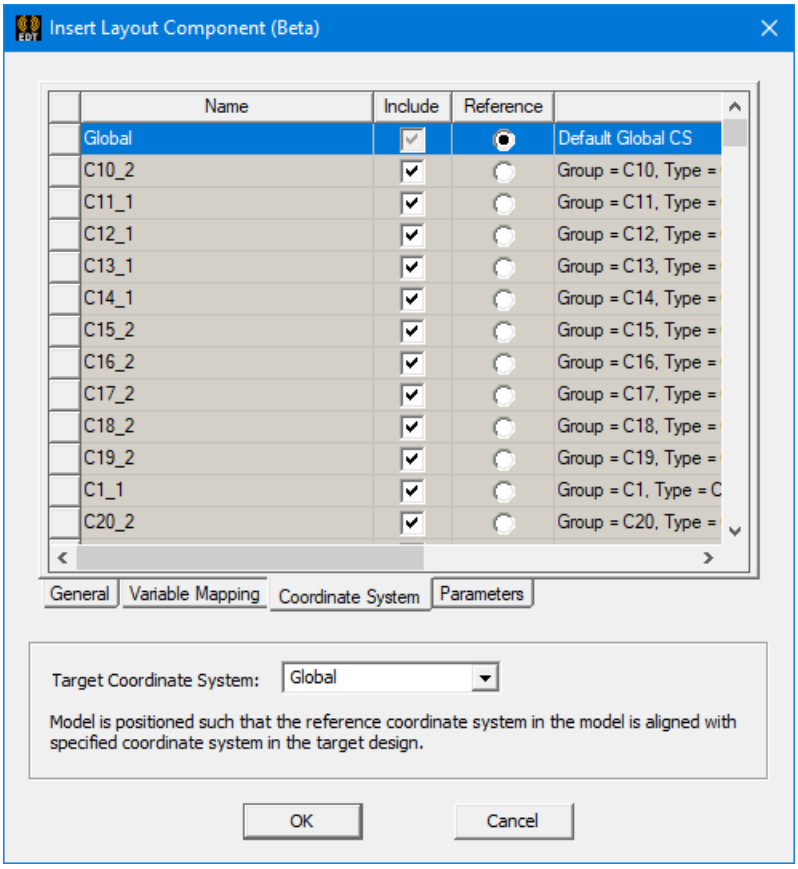


Note:

In the *Parameters* tab (next step), you can choose whether to create local *Design* or *Project* variables from instance parameters. For the latter type, the name will be automatically prefixed with a dollar sign ($) when the local variable is created. Therefore, you do *not* have to manually include a dollar sign at the beginning of the name you type.

4. In the Coordinate System tab, select a coordinate system from layout to serve as reference for component placement. You can allow include or exclude parts of the layout.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Layout component import supports relative CS placement, move, rotate, *etc*. Once imported, the layout ports will be created in the Project Manager tree. The bounding box and ports have actual Maxwell 3D geometry, but any visible object will have snapping locations when in snap mode or when creating a CS.

If the original layout design includes additional CS, these are included as properties to the layout component in Maxwell 3D. As you select the original layout design to import to Maxwell, you can choose a custom coordinate system as the reference CS. After creation in Maxwell, you can choose a target CS for the layout component, and the component will be moved to a place where its reference CS snaps to the target CS or you can move the layout component using the snapping locations; see Editing the Placement of a Layout Component for more details.

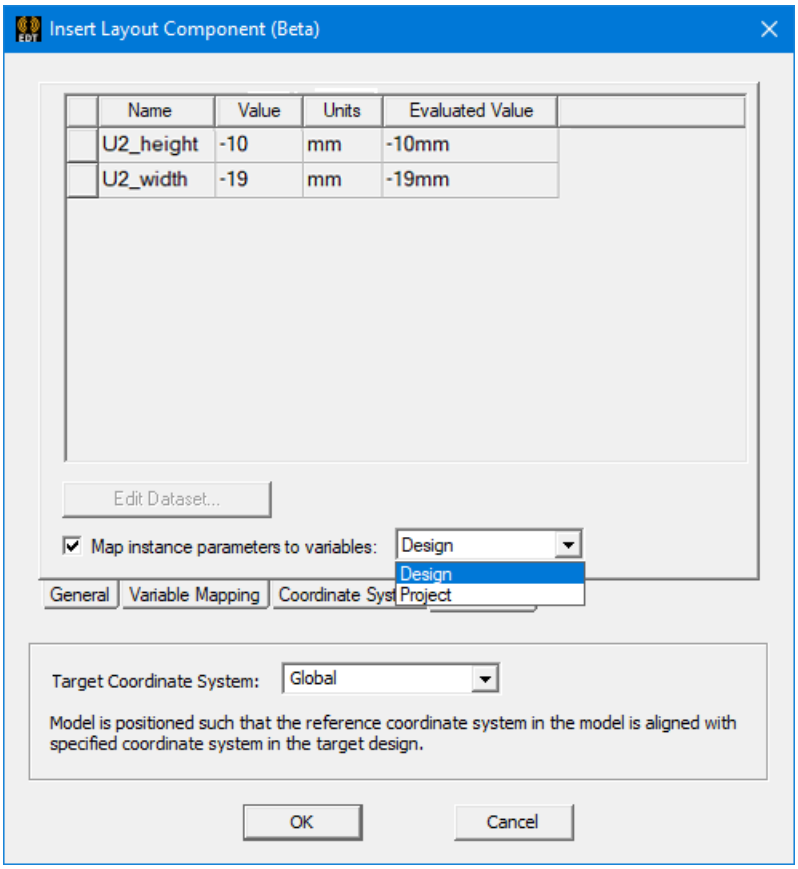
5. Select the **Parameters** tab.

   If you chose not to define local variables in the Variable Mapping tab, or if the source design contains no variables, none will be listed in the Parameters tab. **Skip to step 6**.

   Otherwise, when you have defined local variables in the Variable Mapping tab, they will be listed in the Parameters tab along with their values and units.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following figure shows the parameters with two local variables substituted for numeric values (as shown in the preceding figure):



You can choose whether to keep these mapped variables only as instance parameters or to also create local Design variables or Project variables, as follows:

a. Optionally, toggle the selection of **Map instance parameters to variables** as preferred.

> **Note:** If this option is cleared, mapped geometry variables are exposed in the Maxwell design only as layout component parameters, and you can vary the values between multiple instances of the component in the same design.
>
> Alternatively, if you also wish to include the mapped variables as Maxwell design or AEDT project variables, select this option and choose between the two types in the next step.

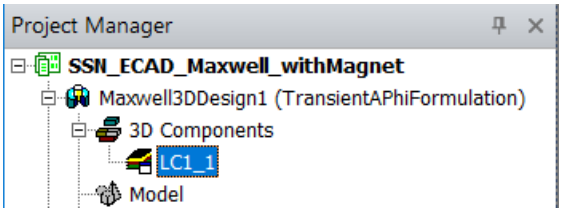
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. If the **Map instance parameters to variables** option is selected, choose either **Design** or **Project** from the adjacent drop-down menu to specify the type of local variable to create.

   As previously stated, the variable name will be prefixed with a dollar sign ($) automatically when you choose to create local Project variables.

6. Click **OK**.

   The layout component name appears under 3D Components in the Project Manager. An instance number ("_1" in this case) has been appended to the component name:



7. Optionally, you can change the visualization characteristics of the layout component; see Visualization for Layout Components for more details.

8. In object selection mode, select all layers of the layout component and assign a length-based mesh operation on the selection. Set the maximum element length to be three to four times the width or height of the grid cells, whichever is the lesser.

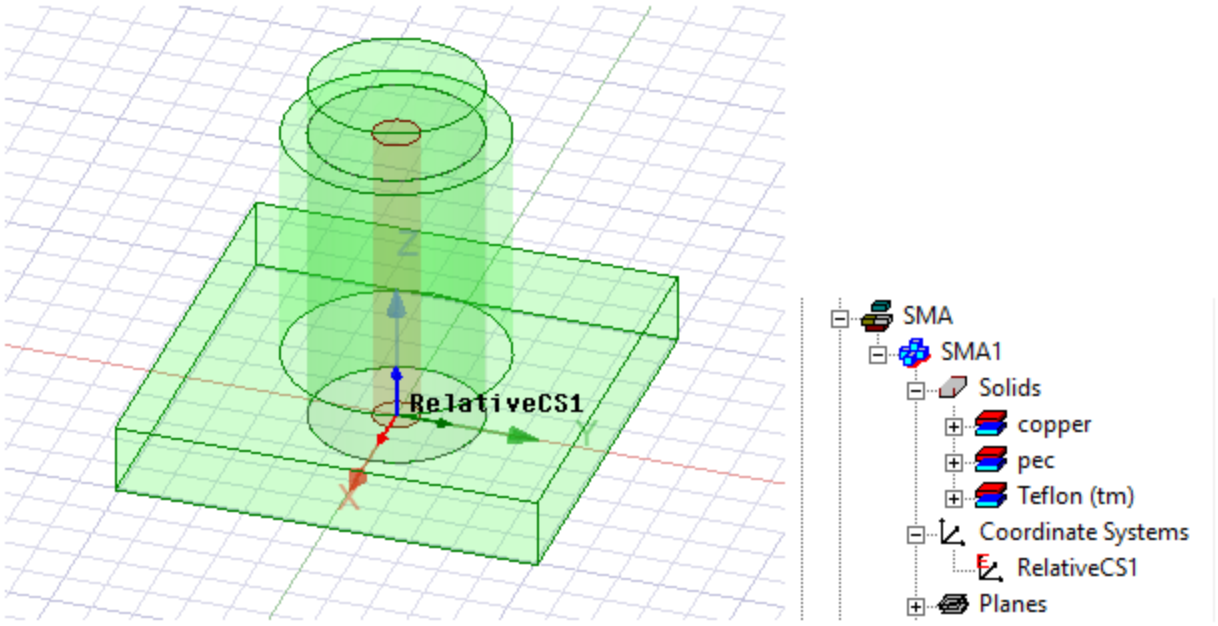
   Typically, you need not enforce the maximum element length inside the selection. For the majority of printed circuit boards, there will only be one element through the thickness of any layer due to their thinness. For more information on meshing layout components, see Meshing Recommendations for a Layout Component.

Using the Layout Component's Reference CS for Placement

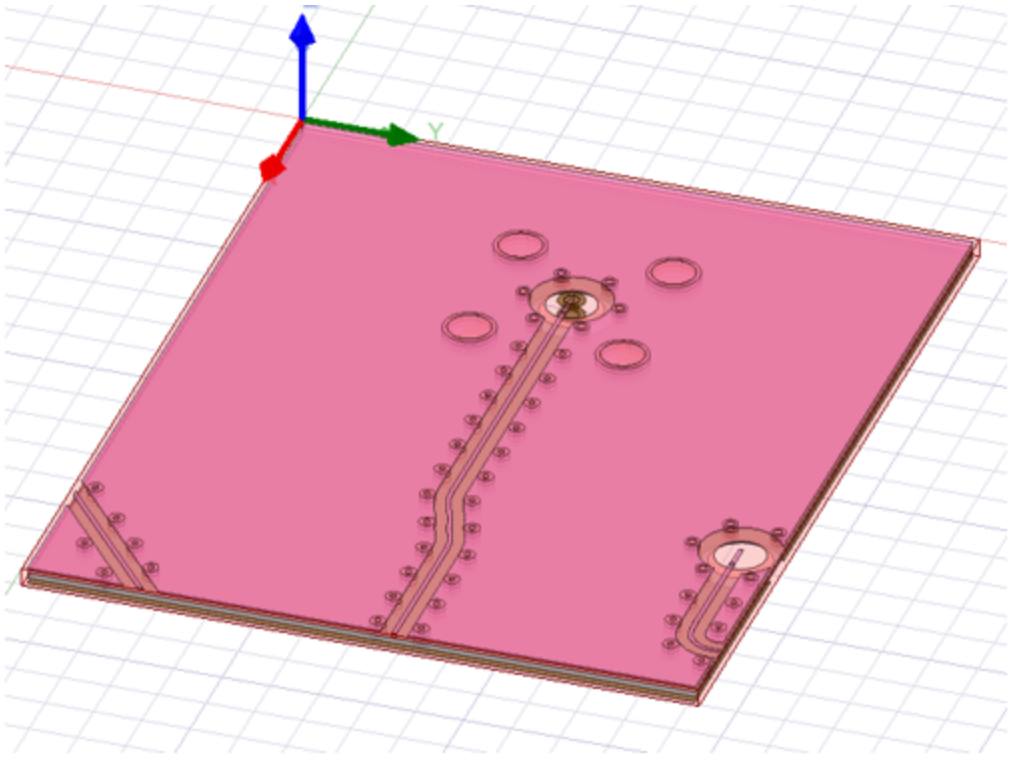
When importing the layout component, it is important to consider which CS to set as the reference coordinate system. If you do not carefully consider the layout's reference CS in relation to the target CS, you will not get the result you expected. This is best explained with the following example.

A 3D component (shown in green) comes with a CS called "RelativeCS1". This CS is located at the bottom of the component and at the center of the internal cylinder structure.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

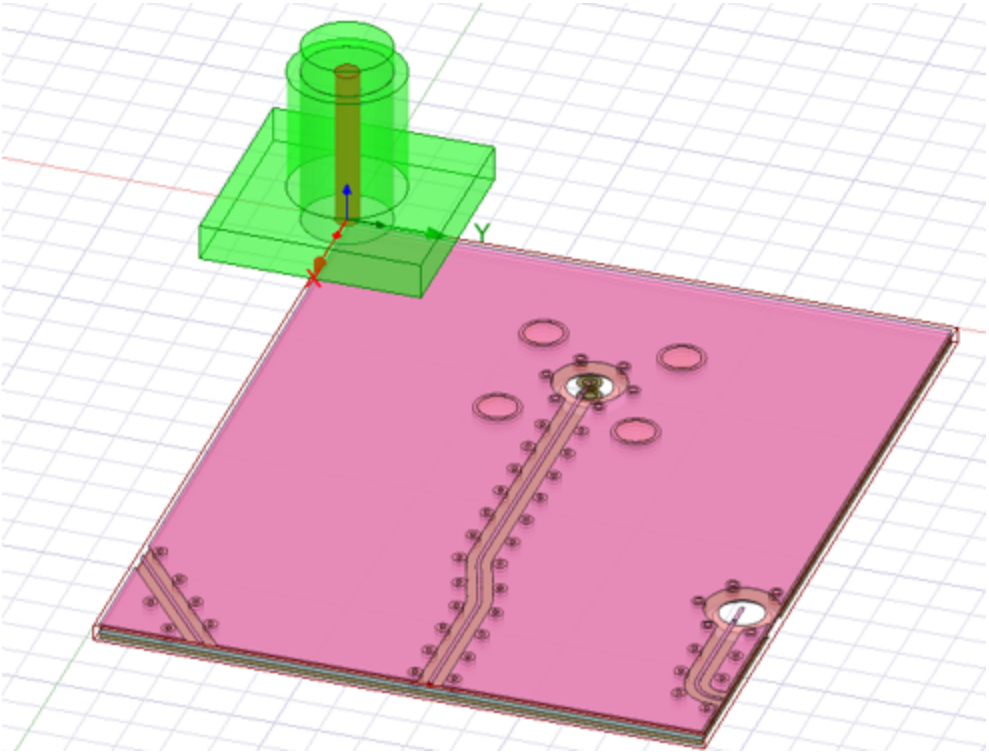


We also have a layout component, shown in pink. In the original layout design of this component, the global CS is located at one of the corners of the entire structure.



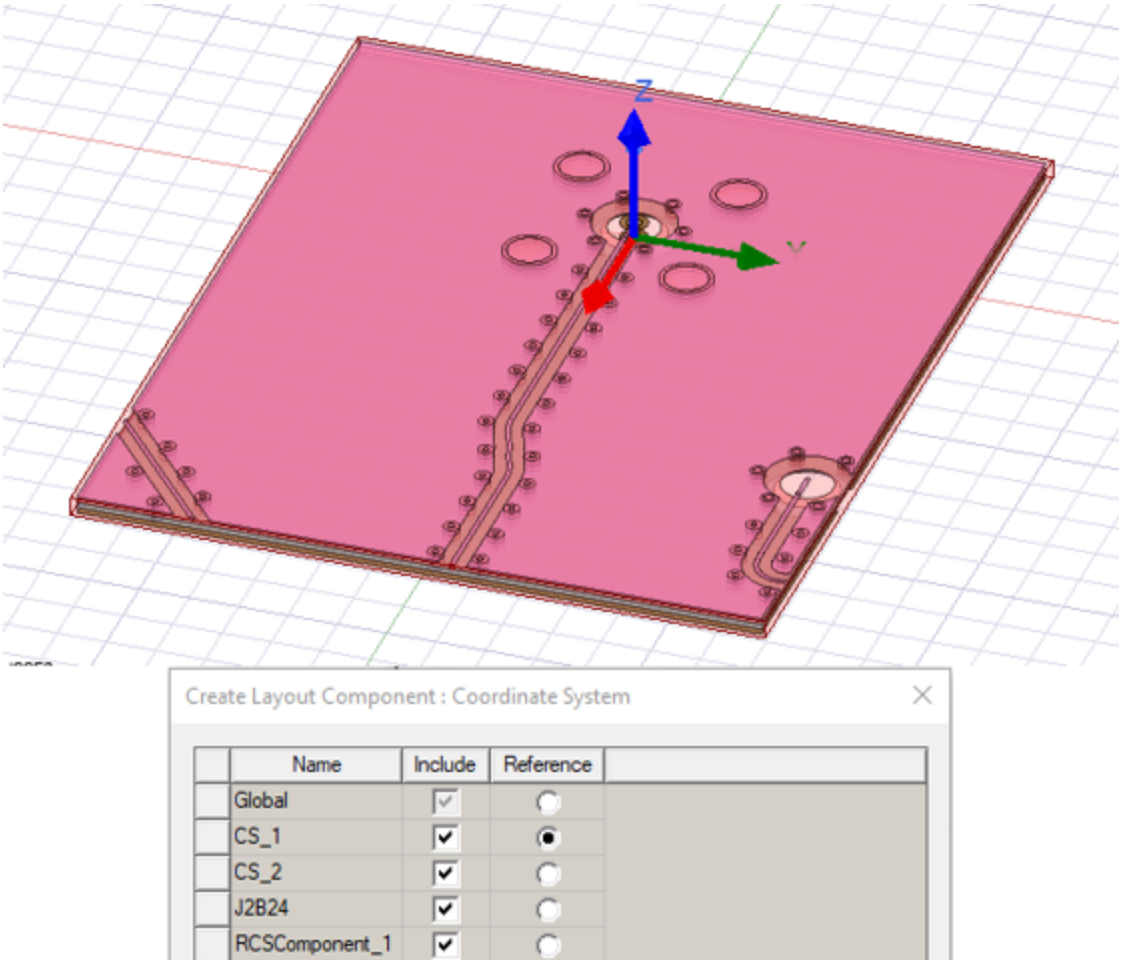
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this example, you want to place the layout component under the 3D component so that its holes (in the middle of the component) are aligned with the hole from the 3D component. When you import the layout component in Maxwell using its global CS as the reference CS and “RelativeCS1” as the layout component’s target CS, it snaps the layout component’s global CS to the target CS, which will look like the figure below.



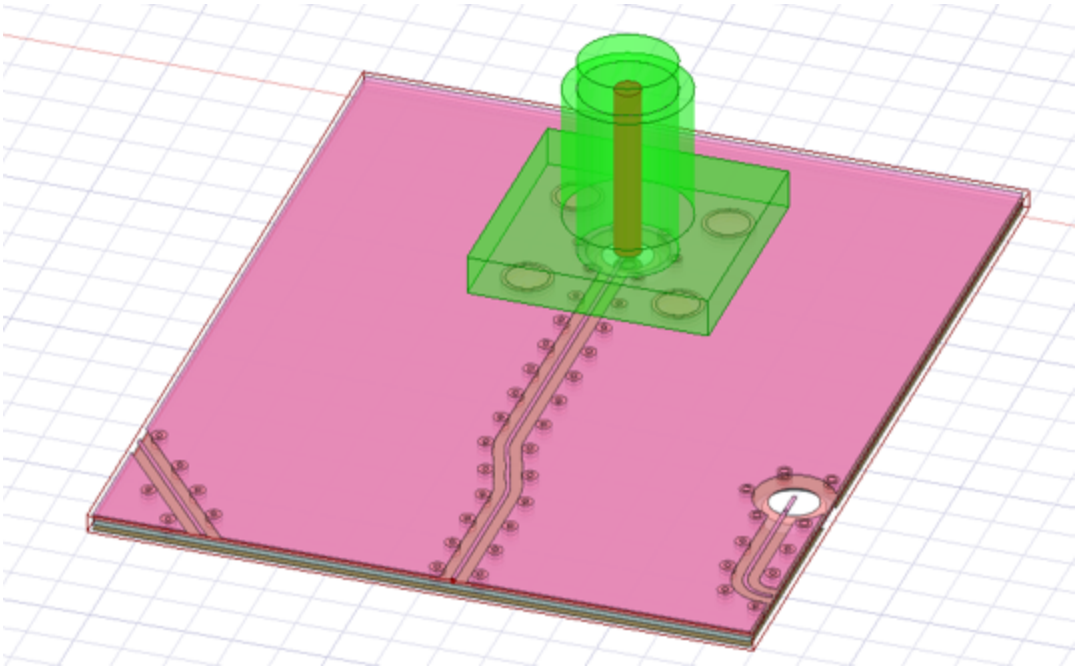
In the original layout design, there is another CS at the center of the holes called “CS_1." When you import the layout component, select this new CS as the reference CS.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The layout component is now in the right place where its reference CS “LC1_CS_1” snaps to the target CS “RelativeCS1.”

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



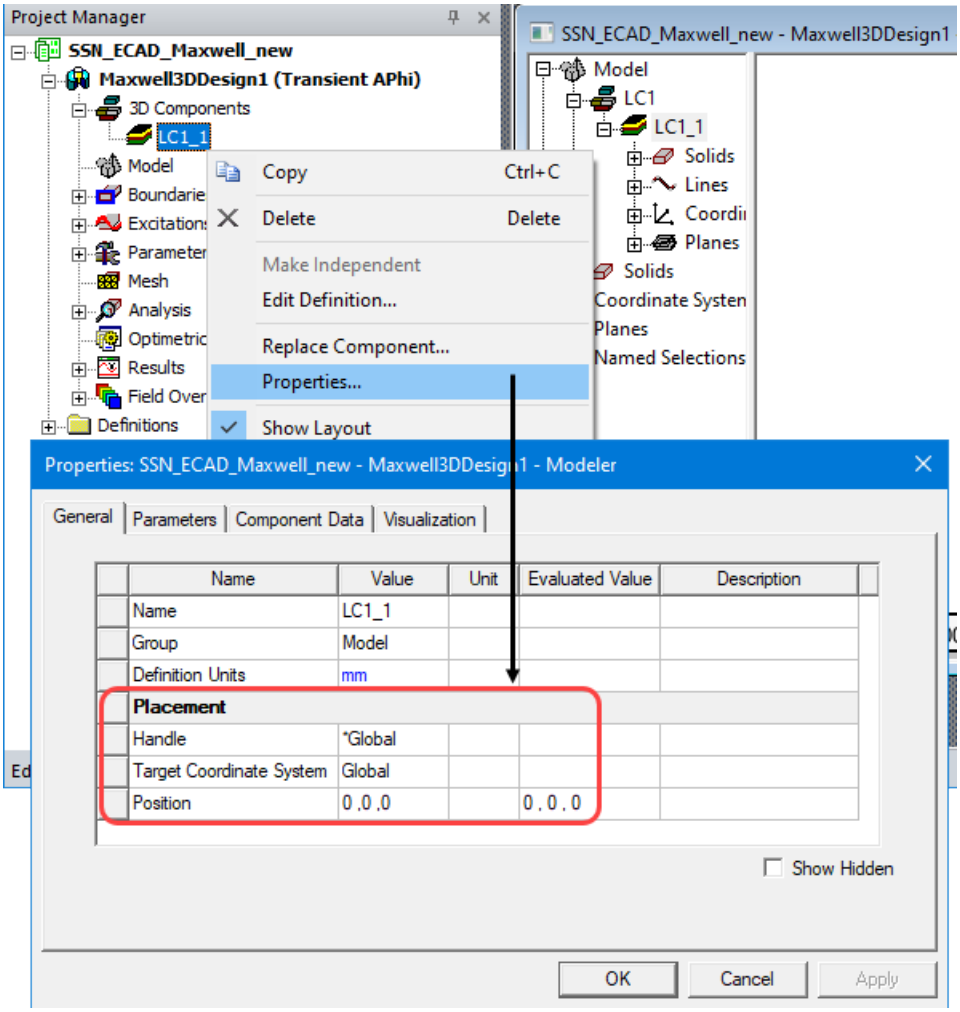
Note: After import, you can edit the placement of a layout component; see Editing Placement of a Layout Component for details. The reference CS is referred to as **Handle** in the layout component's Properties window.

Editing the Placement of a Layout Component

Once you have imported a Layout Component, you can move it by using its handle (referred to as Reference in the Import Layout Component window's **Coordinate System** tab) and target coordinate system or by using standard modeler operations.

To move a layout component, you can edit the component handle and target CS in the **General** tab of layout component instance properties.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



All CS that are included in the layout component are available to be used as the handle. All coordinate systems in the assembly design that are eligible for component placement are available as target CS. Note that the default handle is shown with an asterisk.

You can also use the **Position** parameter to offset the layout component in the X, Y, or Z direction along the axis of the target coordinate system.

> **Note:**
>
> For a video demonstration of how to move a component using its handle and target CS, see Editing 3D Component Properties.

Modeler operations like **Move**, **Rotate**, and **Mirror** also apply to Layout Components. During these operations, snapping points can be helpful to set the start and end points of the move. Any

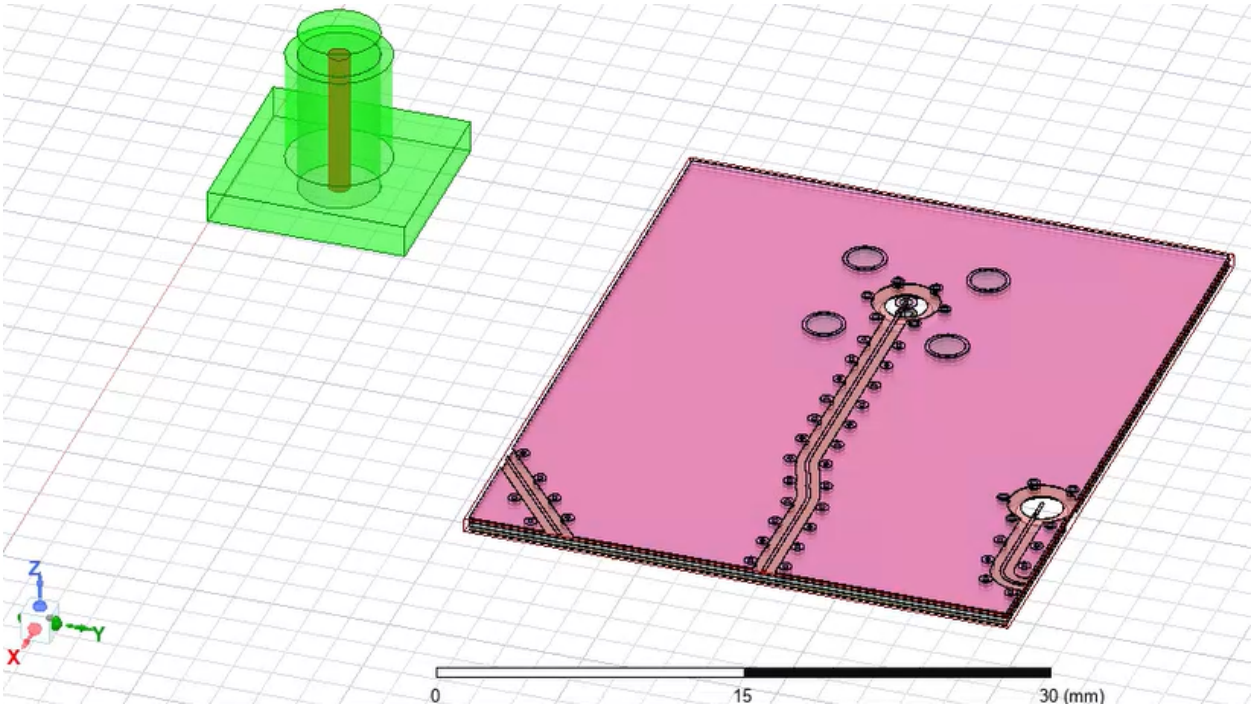
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

visible object (layout component geometry and 3D geometry) will have snapping locations when in snap mode or when creating a CS. Note the following about layout component snapping:

- The snapping location of a layout component behaves the same way as that for a 3D component.
- All snapping types are supported, which include vertex, face center, edge center, edge quadrant, and arc center.
- If an object from a layout component is visible, its snapping locations will be turned on. Once the object becomes invisible, its snapping locations will be removed.
- Snapping locations are restricted to the top and bottom of layers.

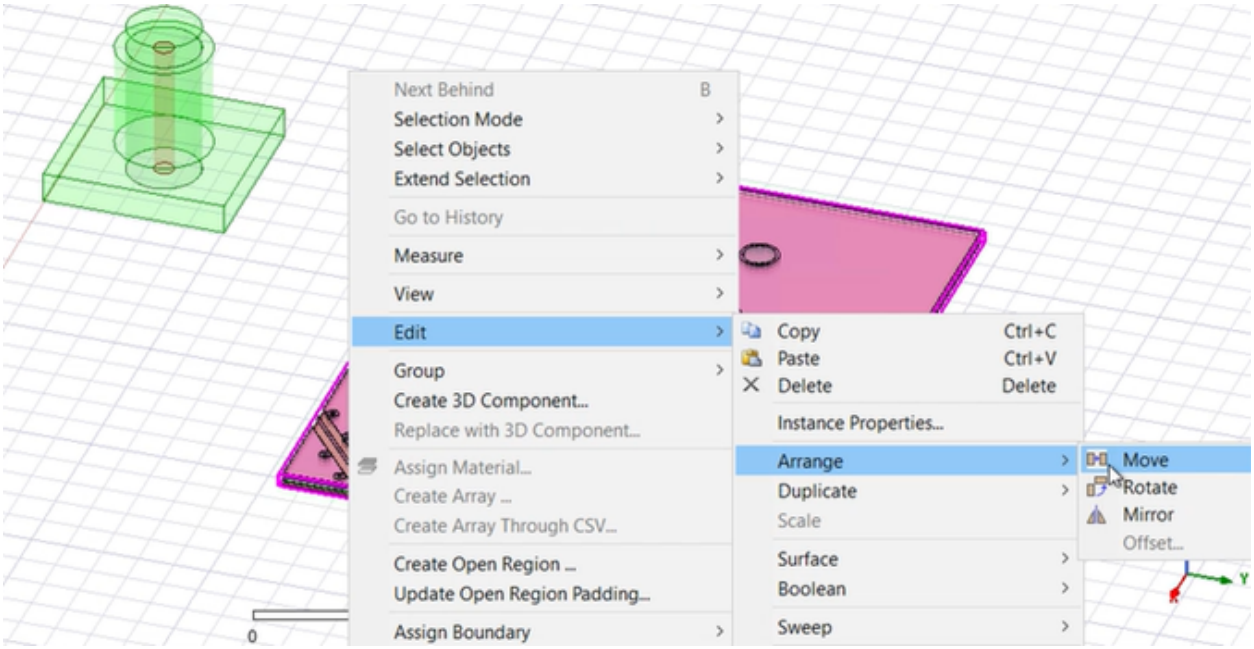
To move a layout component, select one of its vertices snapping point as the start location and another 3D object's vertex as the end location. The layout component will be moved from the start to the end location.

To demonstrate, we will have the example shown below. The green component is a 3D component, and the pink component is an imported layout component. In this example, you want to place the layout component under the 3D component so that its holes (in the middle of the component) are aligned with the hole from the 3D component.

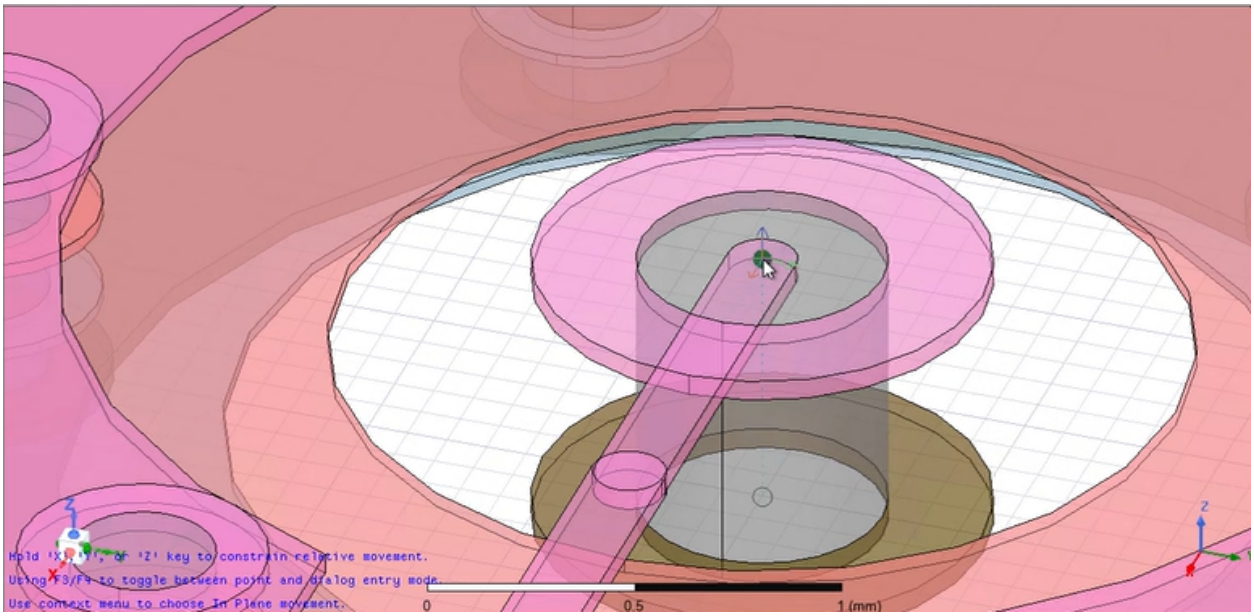


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Right click on the Layout Component, and select **Edit** > **Arrange** > **Move**.

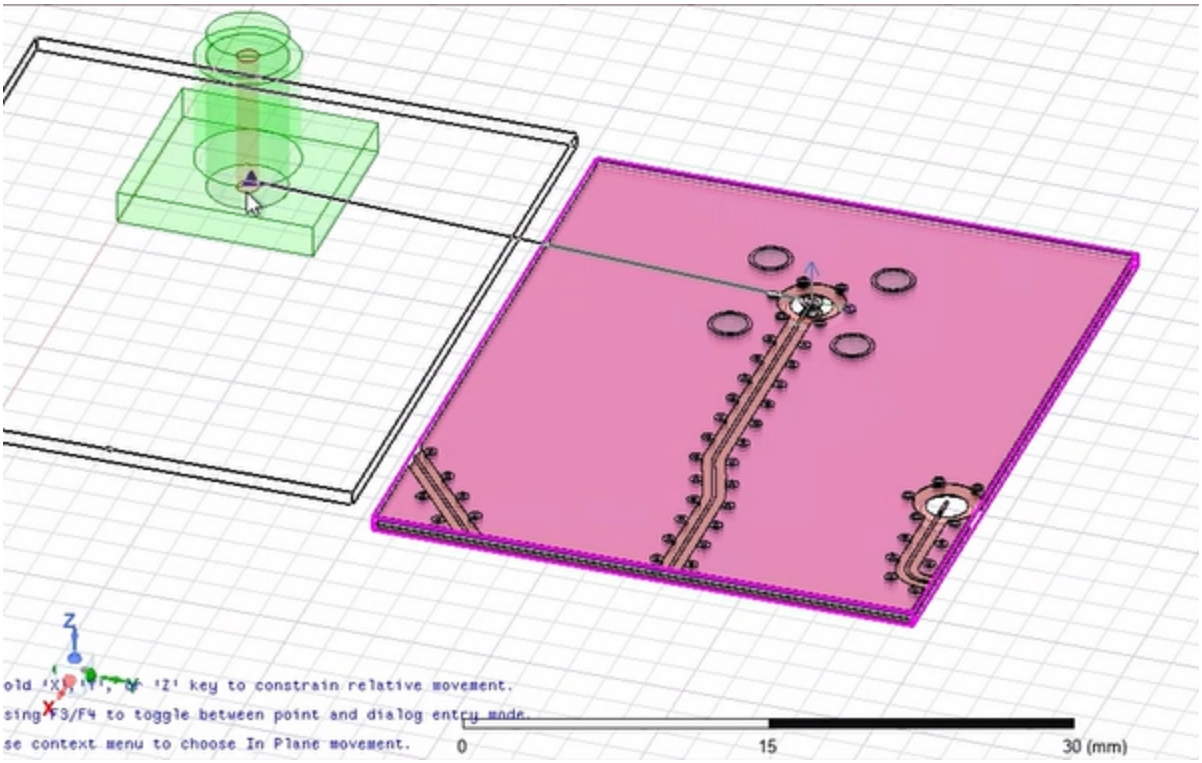


2. Zoom in on the center hole of the Layout Component, and a snapping point will appear. Click to select.

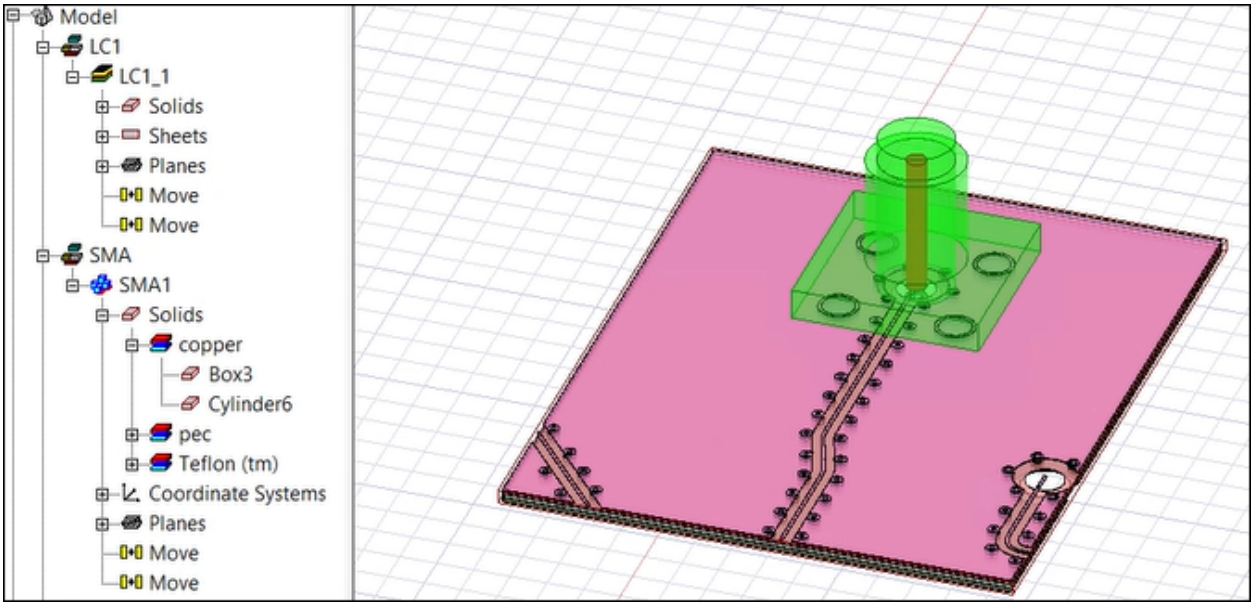


3. Drag your cursor to the snap location where you want to move the layout component, and then click.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The layout component is now located underneath the 3D component.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Inserting Additional Layout Component Instances

Once you have added a layout component to a design, you can insert additional instances of it easily, duplicating the definition of the first one.

To insert two or more instances of the layout component:

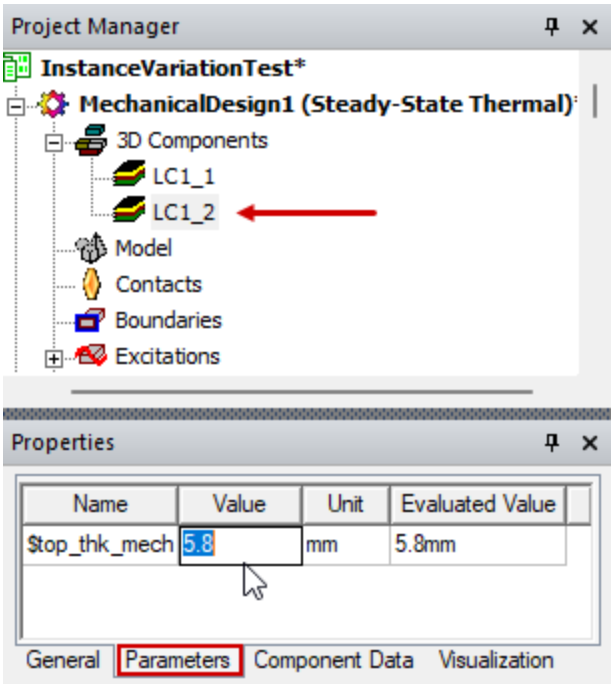
1. Insert additional instances of the layout component using one of the following:
   - Right-click **3D Components** in the Project Manager and choose **Insert Instance** > ***{Layout_Component_Name}***
   - From the menu bar, select **Draw** > **3D Component Library** > **Insert Instance** > ***{Layout_Component_Name}***

   Instances of a layout component will be placed so that the component reference coordinate system is aligned with the current working coordinate system.
2. After inserting an instance, select it in the Project Manager and move it using standard modeling commands; see Editing the Placement of a Layout Component for more details.
3. Under *3D Components* in the Project Manager, select the instance for which you want to change a mapped parameter (such as *"LC1_2"*).

   > **Note:** For variations to occur between instances of the same component, at least one project or design variable must be defined and used in the source layout.

4. In the *Parameters* tab of the *Properties* window, change the **Value** of one or more parameters and press **Enter**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The geometry changes should be immediately visible in the Modeler window.

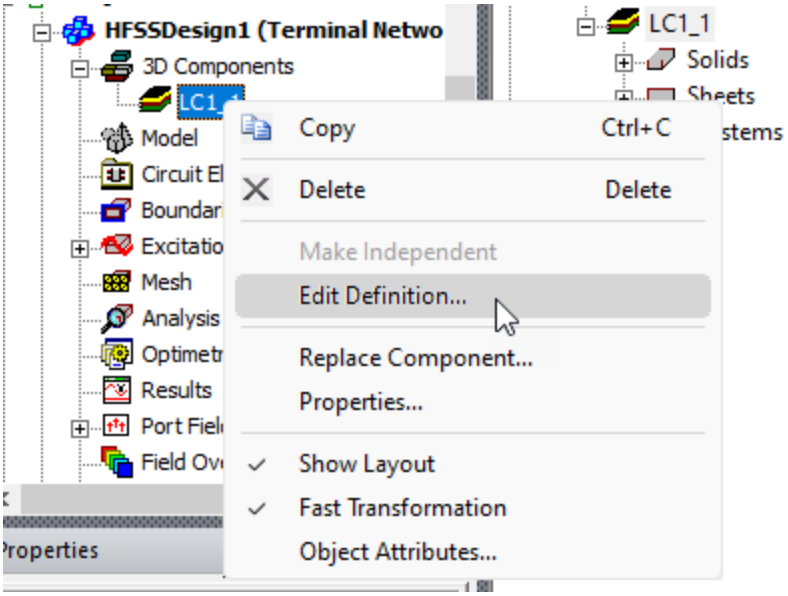
5. Position the viewpoint and zoom in/out as required to clearly compare the geometry for each instance.

If you later decide to change any property in the layout component definition, use the Edit Definition function. However, be aware that changing the definition in this manner affects all instances of the component.

Editing a Layout Component Definition

All layout components in Maxwell designs have a command in the right-click menu called **Edit Definition**.

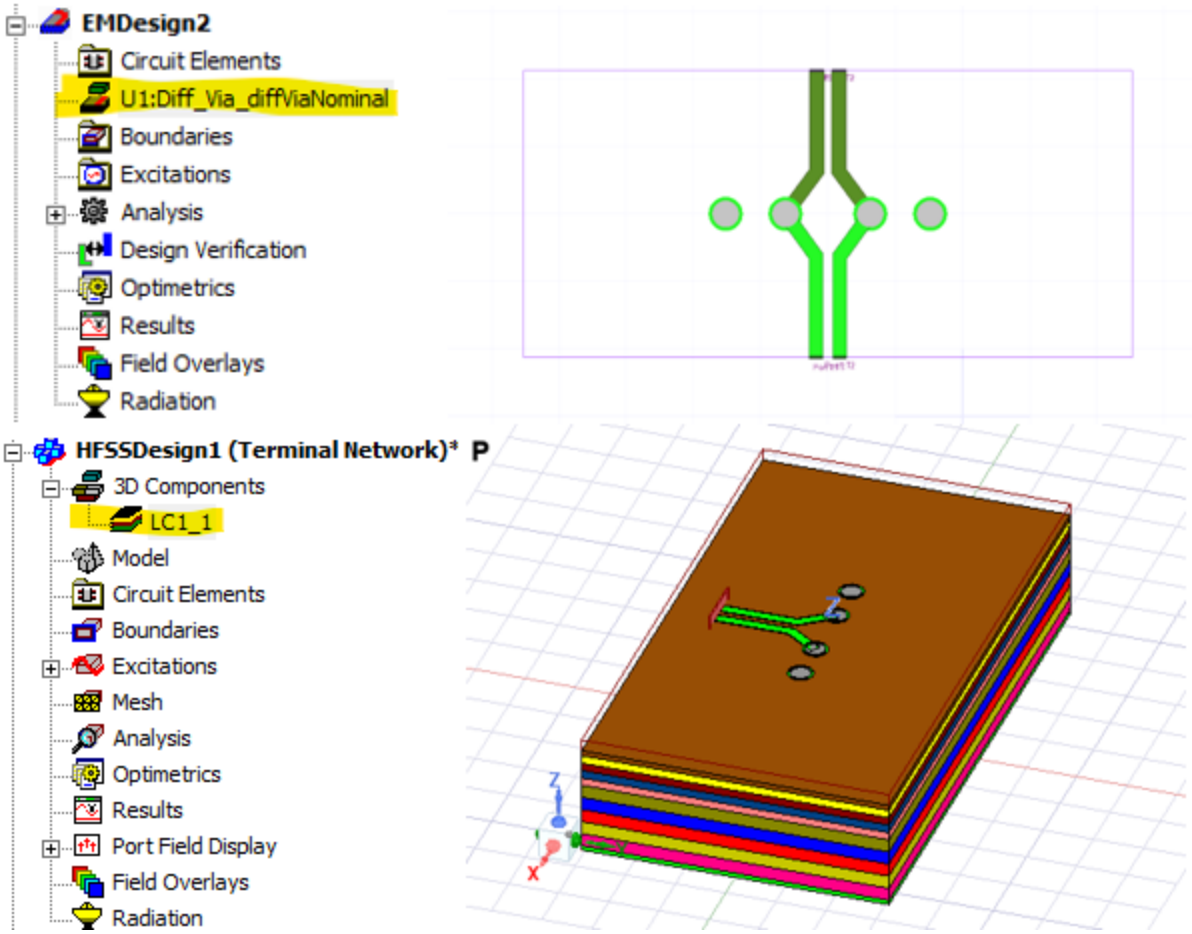
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



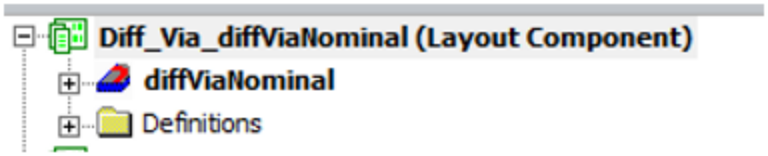
Once clicked, it opens the design containing the EDB definition of the layout component in a separate project to allow for any editing. Once a layout component is edited, either through a parent 3D layout design or through this command, all designs of different types that use this component update it accordingly.

For example, in an Ansys Electronics Desktop project, multiple designs of different types can have layout components of the same definition. Below is an example project where both an HFSS 3D Layout design and an HFSS design have a layout component with the definition “Diff_Via_diffViaNominal”.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

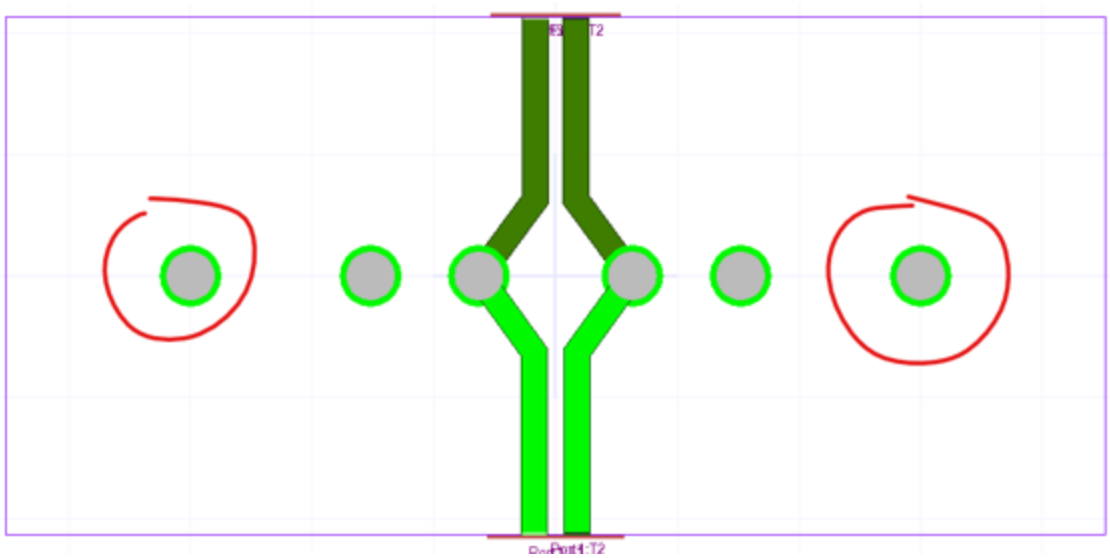


In addition to editing the source layout design using the **Edit Definition** command, you can edit the layout component's local settings, or replace it with a different component, using the **Replace Component** command.



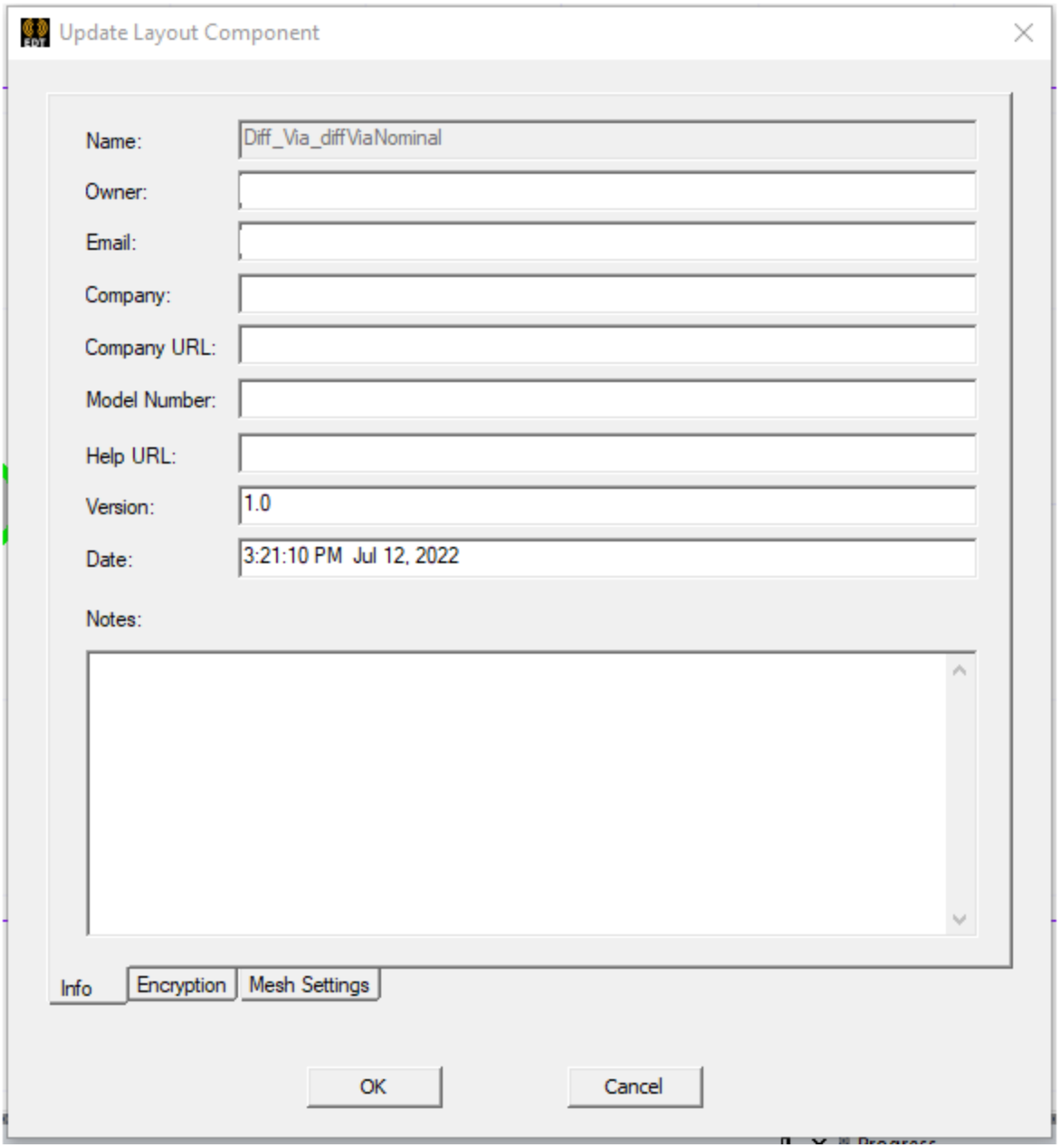
In the source design, you can edit the layout component however you want. This example adds two more vias on the left and right side as shown in the figure below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



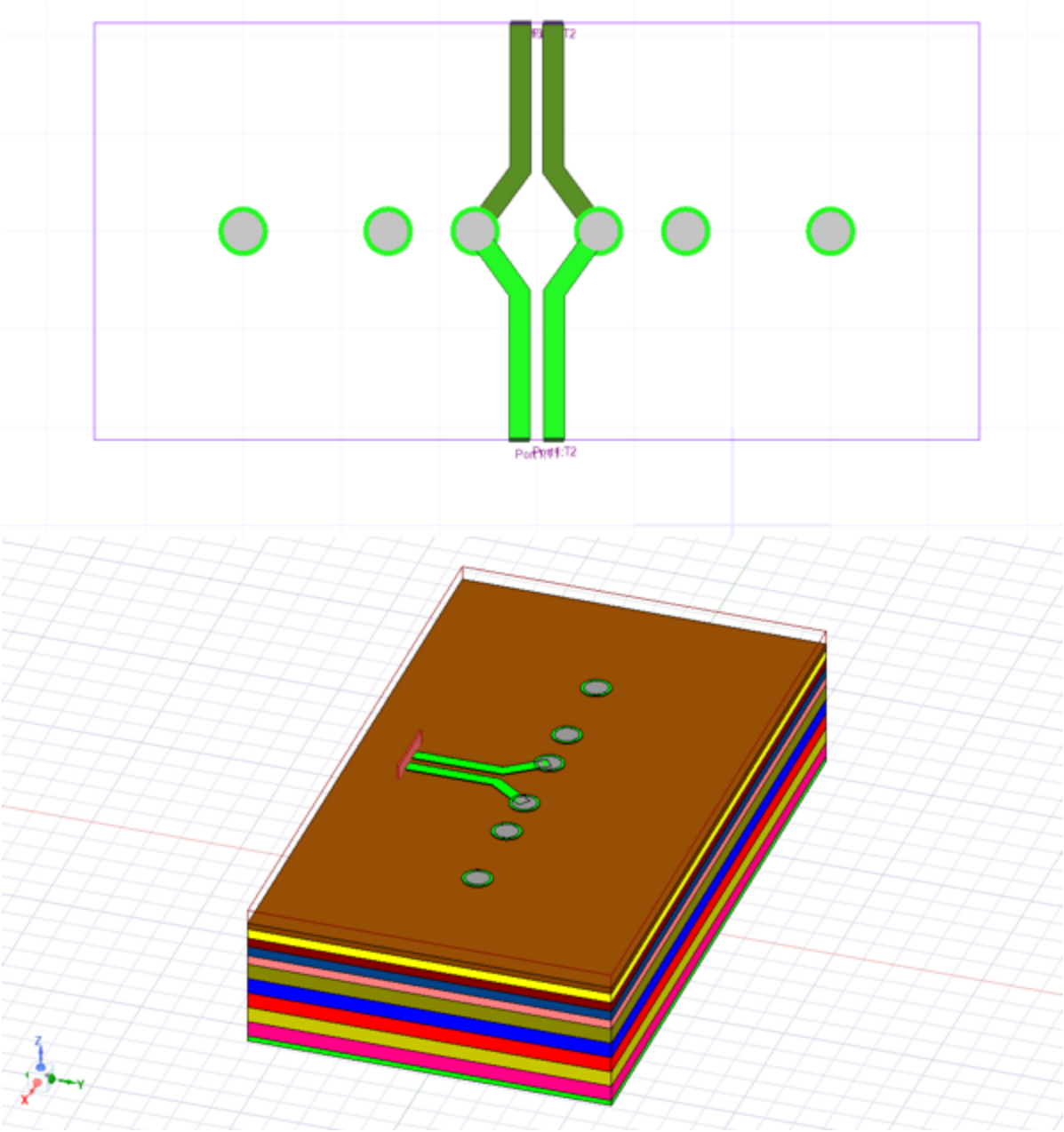
After editing, you can save the source design, and a dialog box opens for you to confirm the changes.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Once the changes are confirmed, both the HFSS 3D Layout design and the Maxwell design containing this layout component update to reflect the changes.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

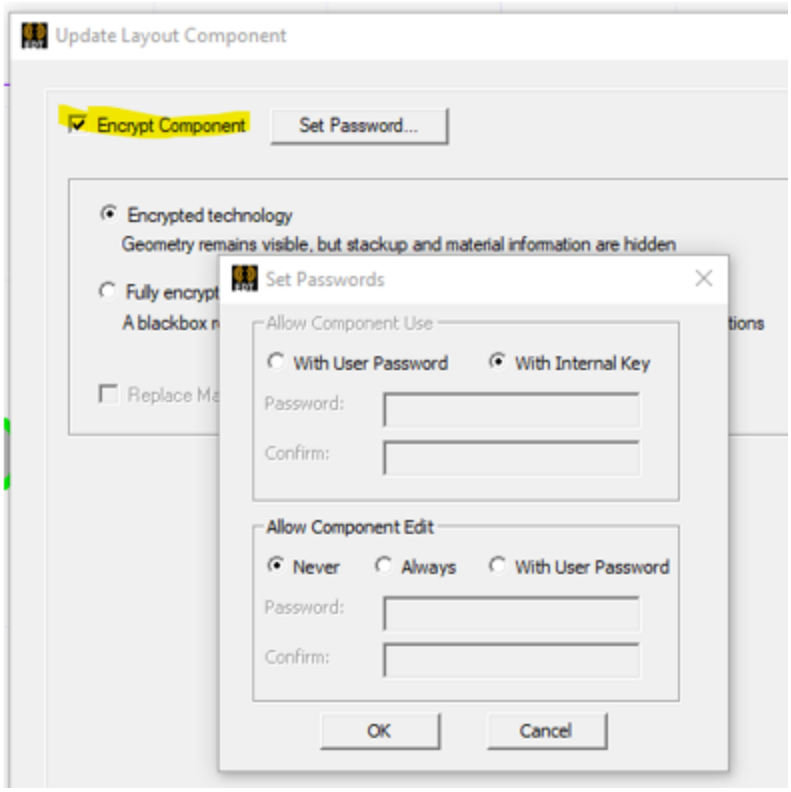


Note:

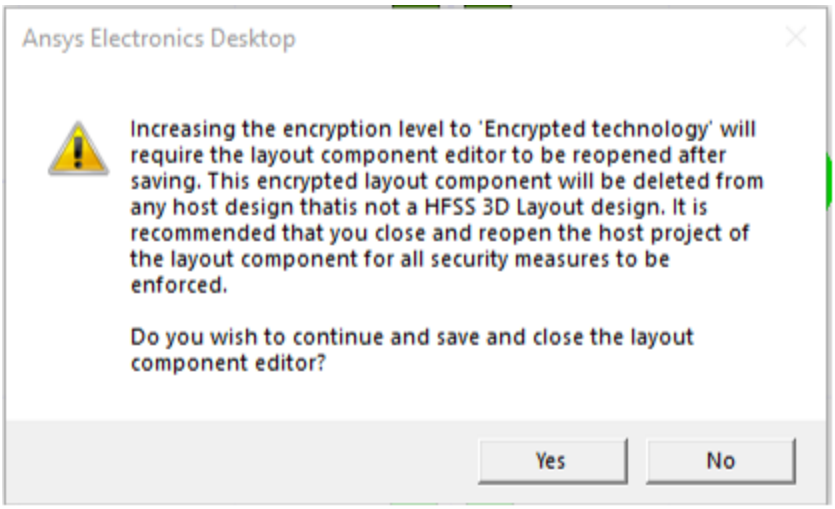
The editing is not undoable. After saving the source model, the target project is automatically saved, and the undo stack is cleared.

While editing, you may choose to encrypt the layout component.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

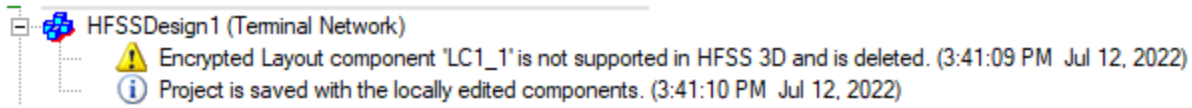


If the layout component is encrypted, a dialog box with warning messages alerts you about the potential impact to the designs containing this layout component.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After editing, all encrypted layout components are deleted from any target designs as encrypted layout components are not currently supported. A warning message appears for each deleted layout component.



Related Topics

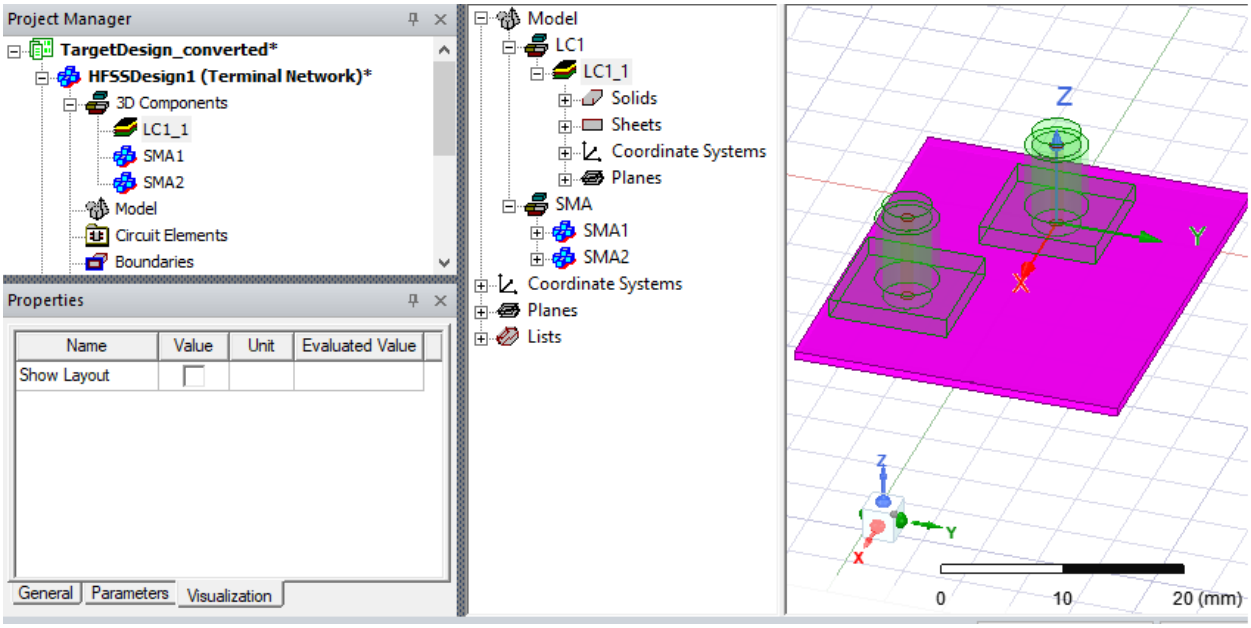
Replacing or Editing Layout Components

Field Plots on Layout Component's Layer or Net

Plotting the Mesh for Layout Components

Visualization for Layout Components

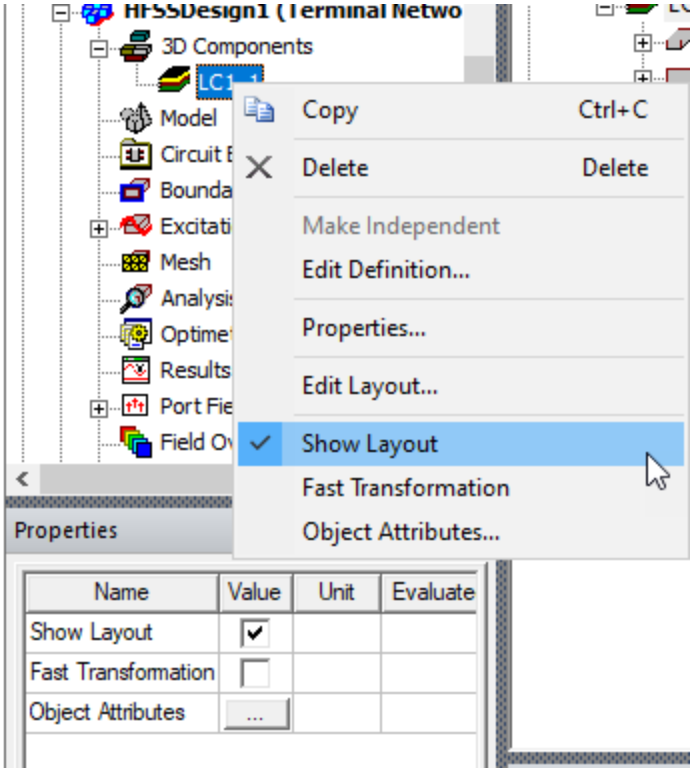
You can show each layout component in the 3D editor and specify the visualization attributes of each layer, net, or object in the layout component. This can be done in each layout component's property window through the **Visualization** tab or by checking **Show Layout** in the right-click shortcut menu on the component. If **Show Layout** is not checked, the layout component is displayed as a bounding box. You can also visualize RLC boundaries assigned on an edge or across layers, and component groups that are categorized as circuit elements.

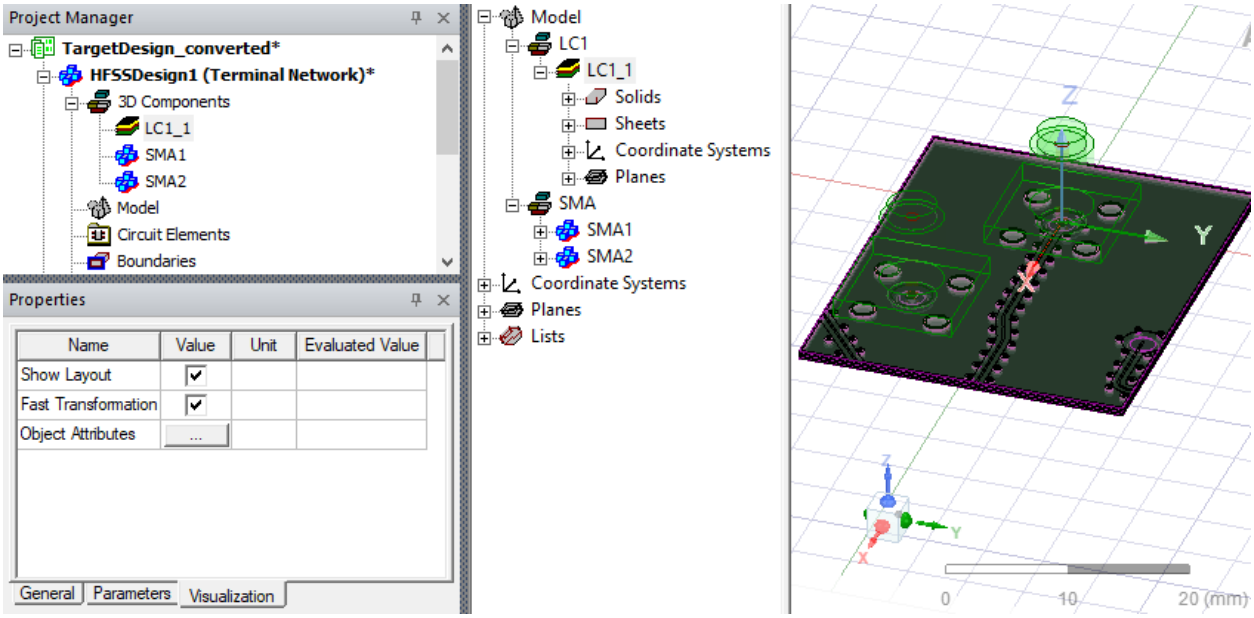


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using Show Layout for Layout Components

Checking **Show Layout** on the shortcut menu or the **Visualization** tab displays the layout.

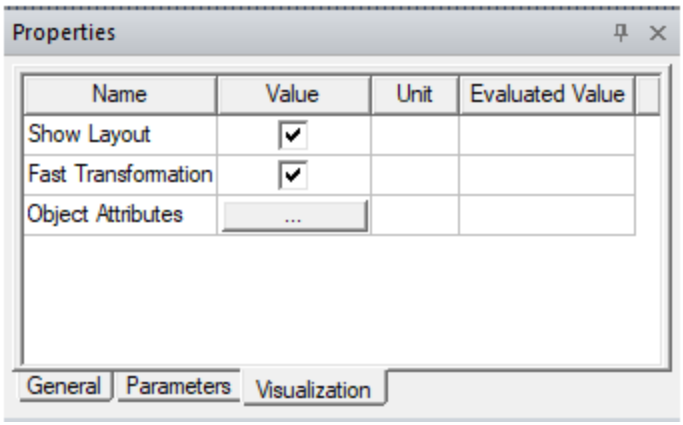




Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

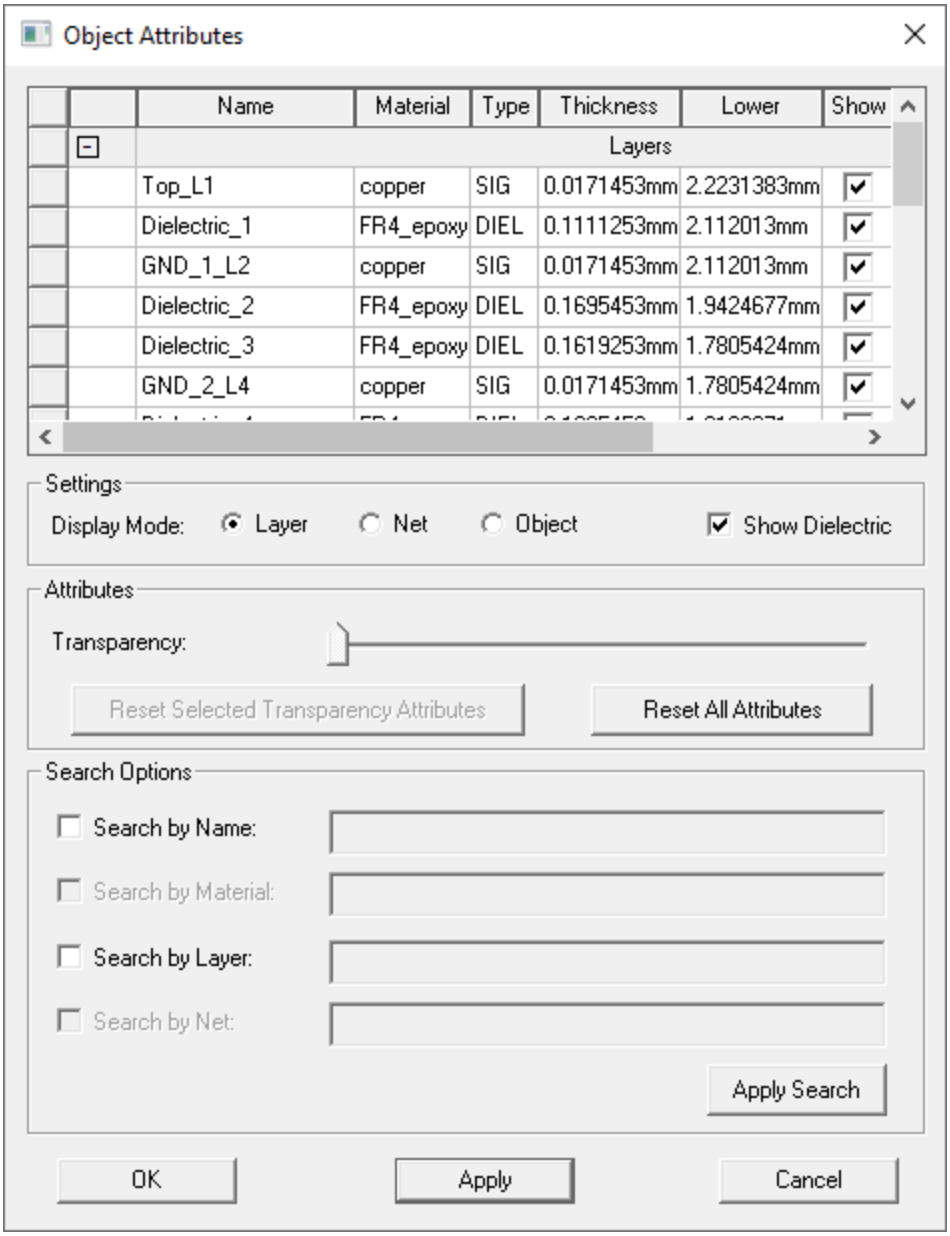
Object Attributes Control of Visualization for Layout Components

If **Show Layout** is checked, the **Object Attributes** [**...**] button provides access to the visualization attributes of the component in a separate dialog box. The **Fast Transformation** option toggles whether layout components show outlines of objects across layers during view changes. This improves the visualization performance.



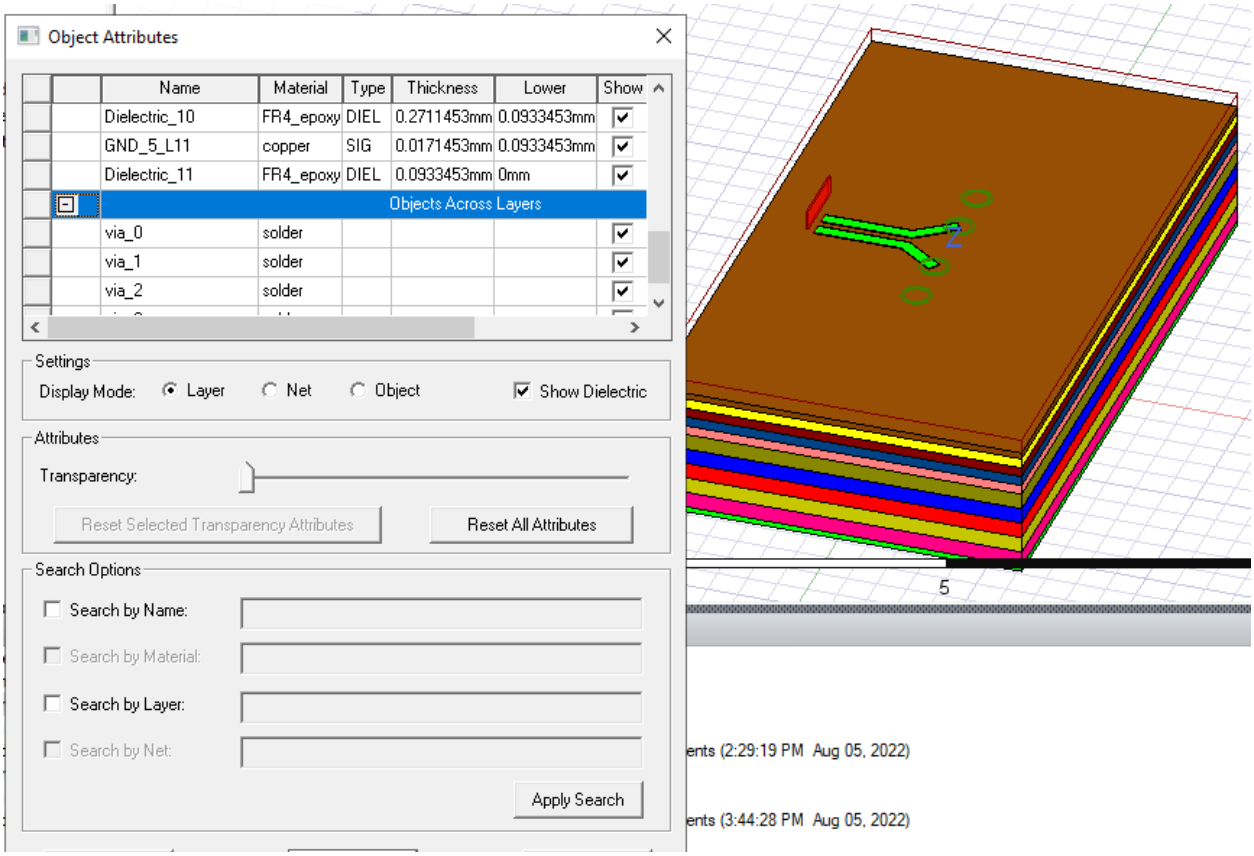
This is the **Object Attributes** dialog box. It provides fine control over how the Layout 3D Component is displayed.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

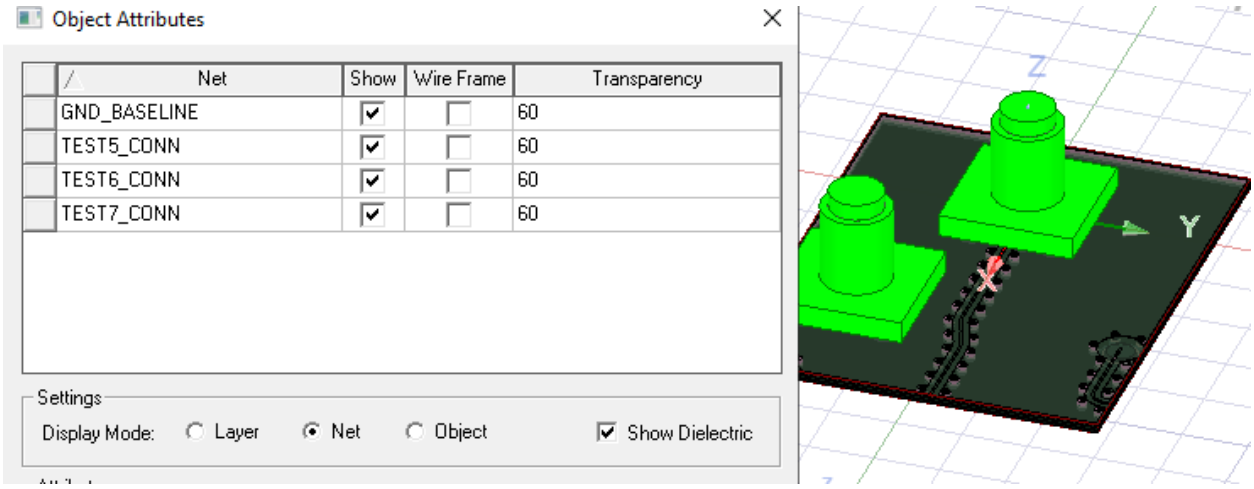


In the Settings group box, you can set the display mode as **Layer**, **Net**, or **Object**, and whether you want to **Show Dielectric** objects in the component. In the layer mode, this dialog displays information about each layer like the material, type, and thickness. Each layer is sorted by height instead of alphabetically. A button named **Reset All Attributes** is added to the dialog box to quickly reset the attributes of all items in this mode. The **Layer** mode displays every layer and every other object across layers separated by a separator. For Layer mode only, the table includes grouping buttons.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



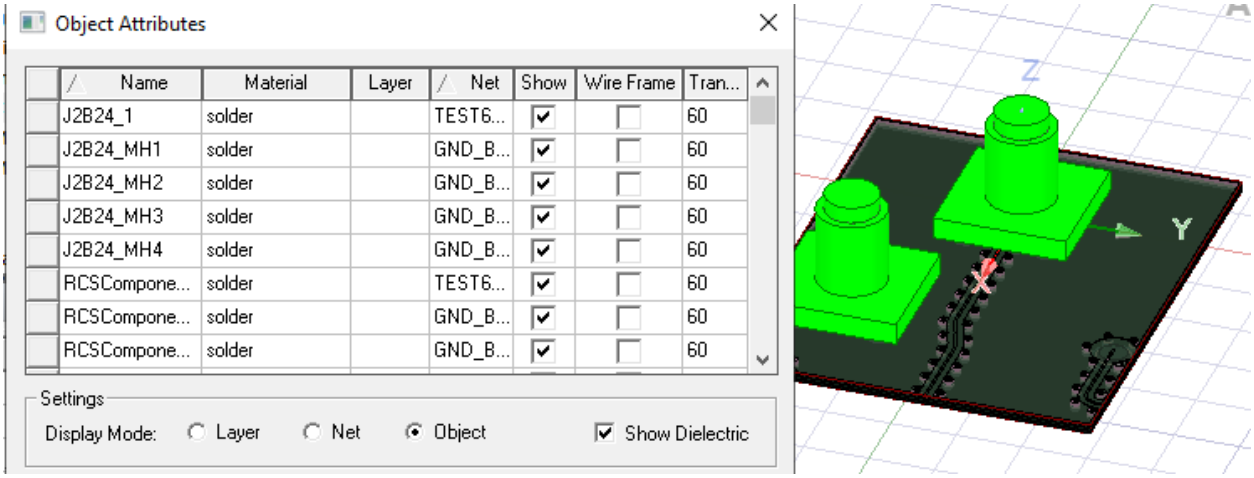
The **Net** mode displays component nets and ignore others objects not belonging to a net.



The **Object** mode displays every object from the component.

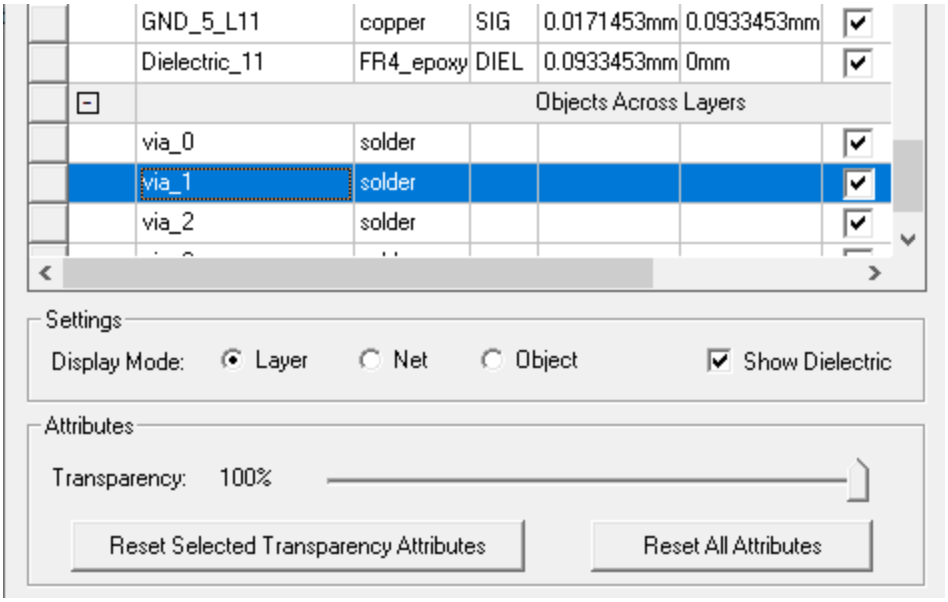
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: For large layout components with many objects, the performance in object mode could be slow.



For each item listed in the table, you can choose how you want to show it by selecting options in the **Show**, **Wire Frame**, and **Transparency** columns. You can check/uncheck the entire **Show** or **Wire Frame** column by clicking the respective column header. Click the **Transparency** column header to select/deselect all rows. Click other column headers to sort the table according to names from that column.

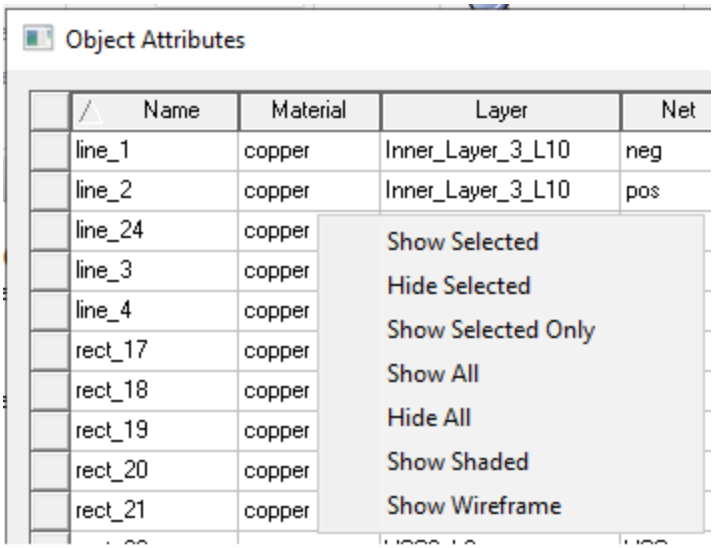
The dialog box supports multi-select functionality. When you select rows, the **Attributes** options become available.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

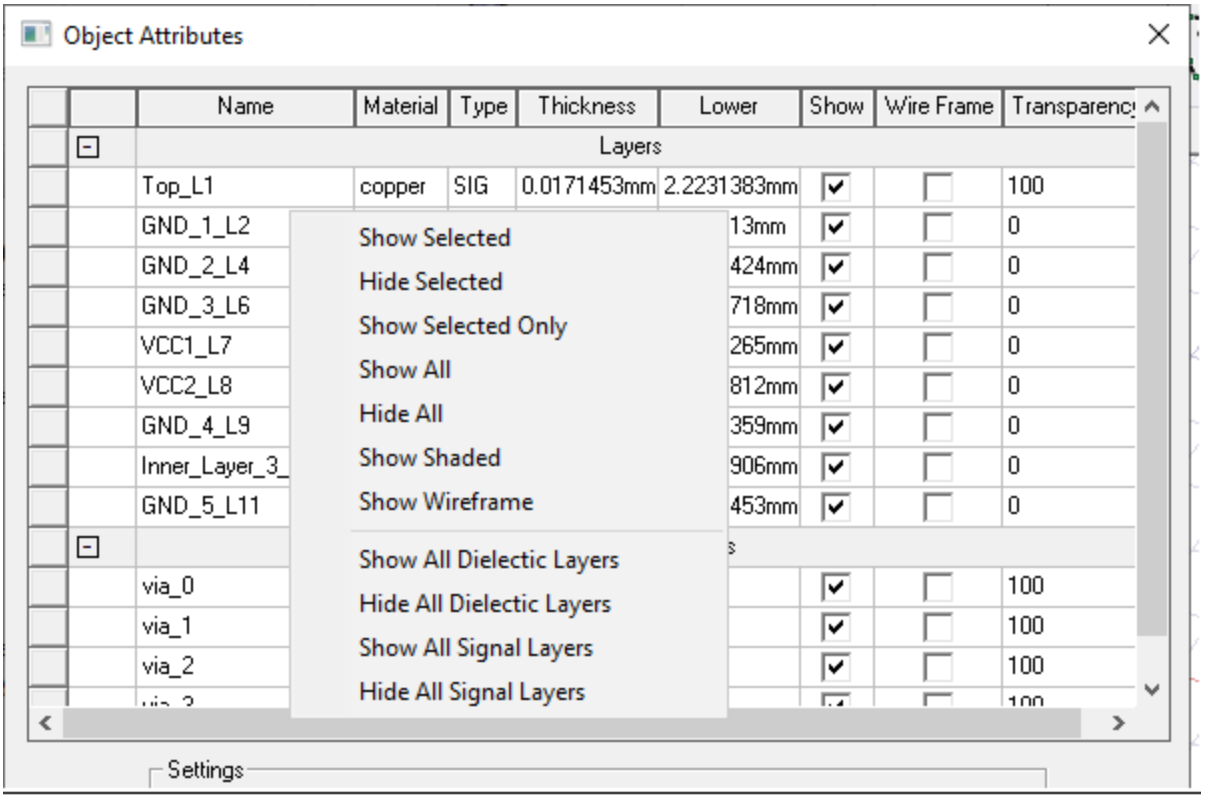
You can drag the **Transparency** slider bar to adjust the transparency value of all selected rows. Finally, the **Reset Selected Transparency Attributes** button sets the transparency value of all selected rows back to the default values when the component was imported. The default transparency of a layer or a net is calculated as the average transparency of all objects belonging to it.

In the **Layout Component Object Attributes** dialog box, if you right-click anywhere inside the table, a pop-up menu will appear to let you quickly turn on/off visualization of objects.

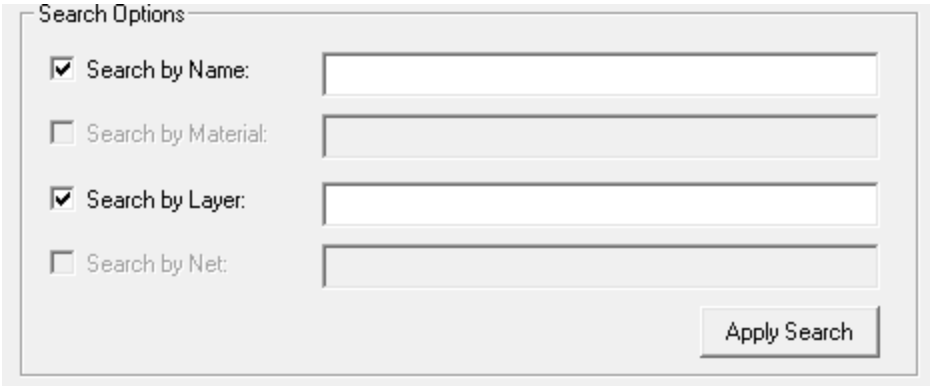


If the dialog box's display mode is layer mode, and you right-click the region containing layers, the pop-up menu contains additional menu items for quickly showing/hiding certain layers.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The dialog box also has search functionality to search for specific items. You can enter their search criteria in the **Search Options** group box.

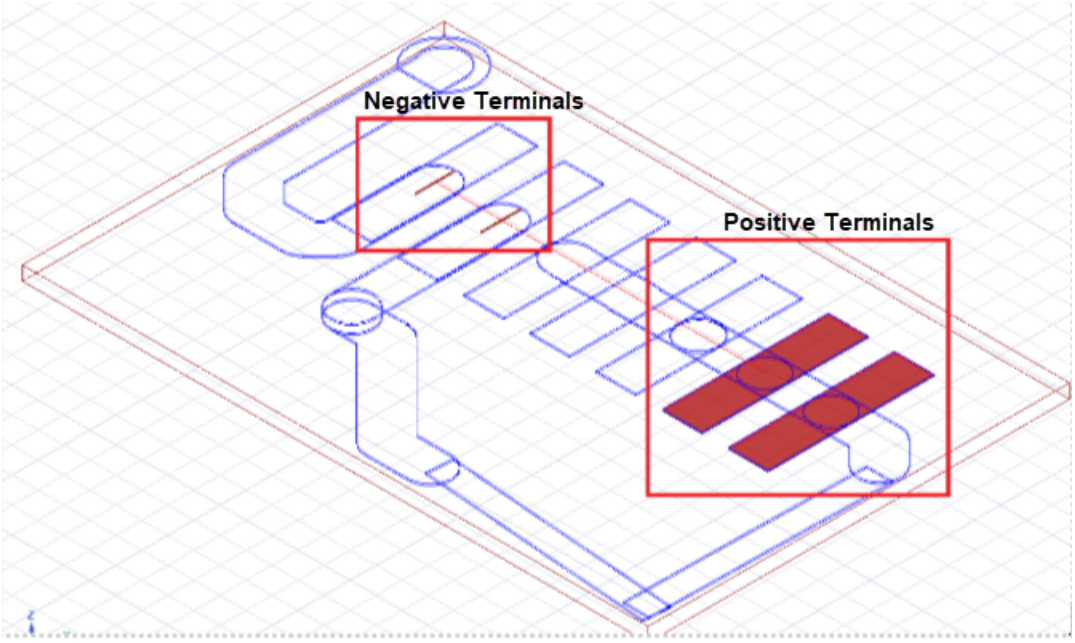


By clicking **Apply Search**, the table refreshes to only show items that match the search criteria.

Visualizing Terminals

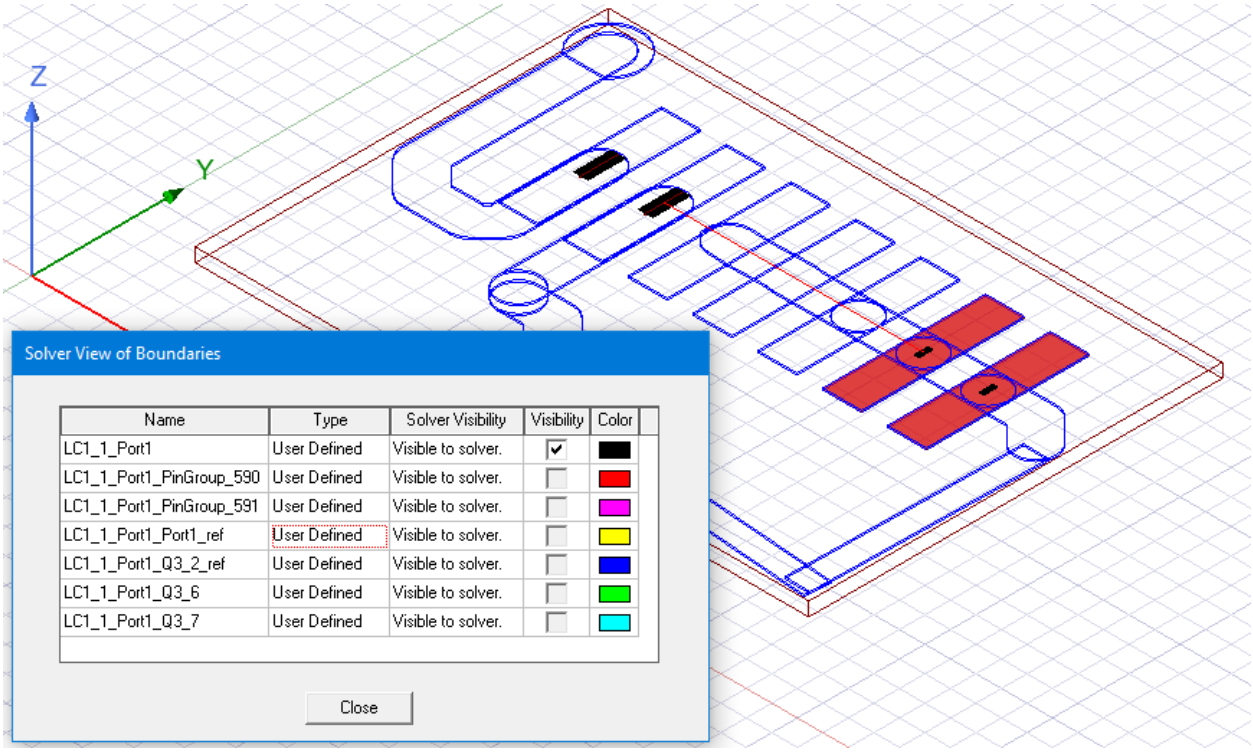
In the Maxwell solver, only the rough representations of the layout component terminals are imported and displayed on the geometry:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



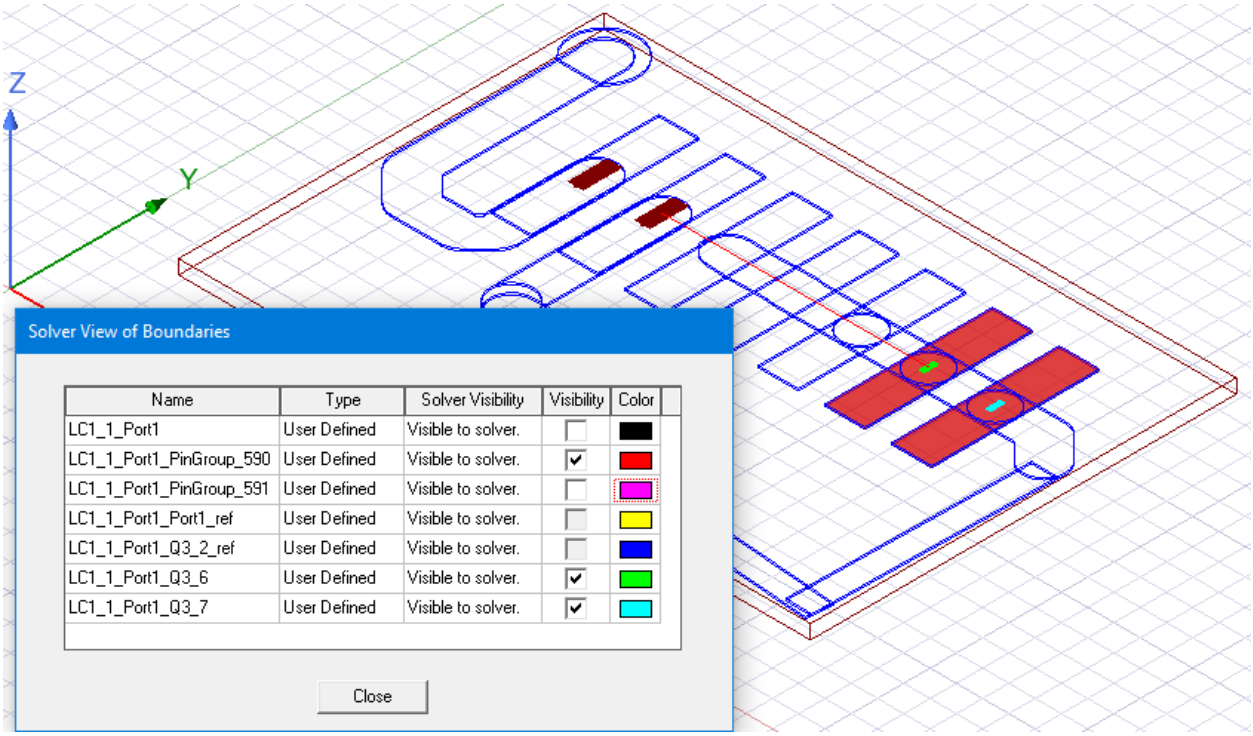
To visualize the actual geometries for the terminal assignment, select the design name in the Project Manager window, and either right click to select **Boundary Display (Solver View)**, or select **Maxwell 3D** > **Boundary Display (Solver View)**. The Solver View of Boundaries dialog box will open for selecting terminals for viewing. In the first example that follows, there is a port assigned on two pin groups. Each pin group consists of two terminals. Turning on the visibility will display all the terminals under this port in the same color. All the child excitations of the port, including the two pin groups and the four terminals are disabled:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In this next example, a pin group and two individual terminals are selected. The two terminals comprising the pin group are the same color. Note that the color of a boundary visualization can be customized by double-clicking its color grid cell and selecting another color with the color chooser.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note that the geometries of the terminals are created when the initial mesh is generated. If you have not generated an initial mesh before opening the Solver View of Boundaries dialog box, the mesher will be launched to create the initial mesh, then each boundary and excitation will be displayed according to its geometry assignment in the initial mesh.

Related Topics

Using a 3D Layout Component

Visualization for RLC Boundaries and Component Groups Designated as Circuit Elements

Replacing or Editing Layout Components

Editing Placement of a Layout Component

Flex PCB Components Support

Field Plots on Layout Component's Layer or Net

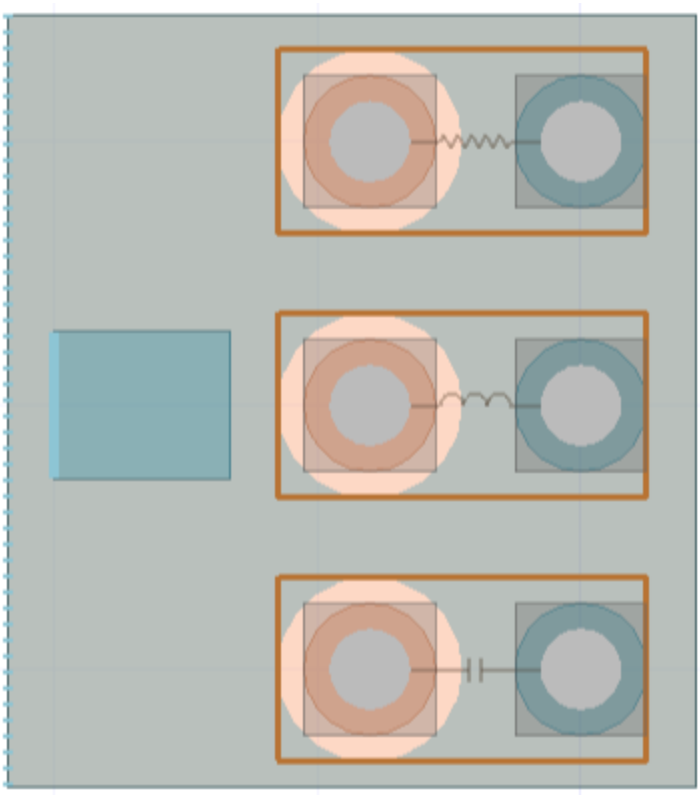
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Visualization for RLC boundaries and Component Groups Categorized as Circuit Elements

You can visualize RLC elements for layout components in Maxwell in the same way as they appear in HFSS 3D Layout Components.

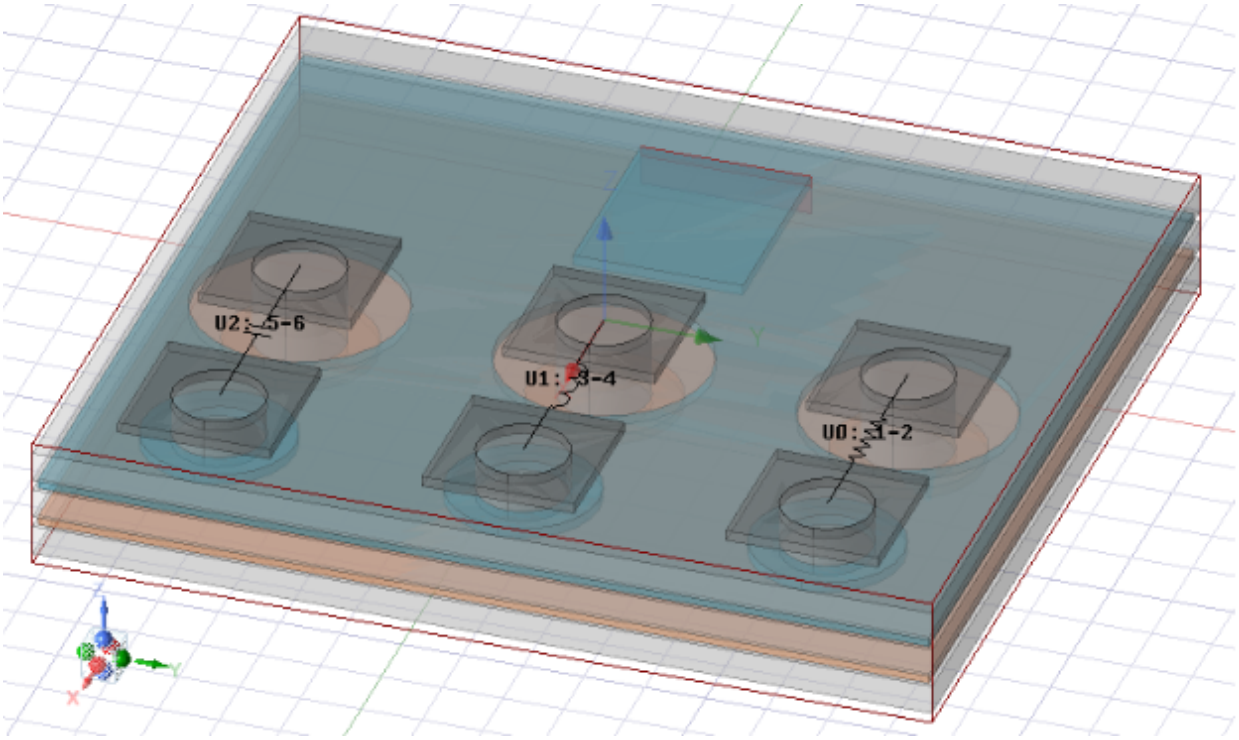
- The types of circuit elements that can be visualized include RLC boundaries assigned on an edge or across layers, and component groups that are categorized as circuit elements. Other forms of circuit elements currently cannot be included in a layout component and thus are not supported.
- Display each circuit element's name on top of the visualization.

You can toggle these RLC visualizations with the **Show Layout** option. The intent is to display each layout component's component data in its property window similar to that of 3D components. For example, given a 3D layout design with circuit elements, the visualization of circuit elements in the HFSS 3D Layout modeler window can appear as shown below.



When imported as a Layout Component into Maxwell, the layout components would appear as shown below.

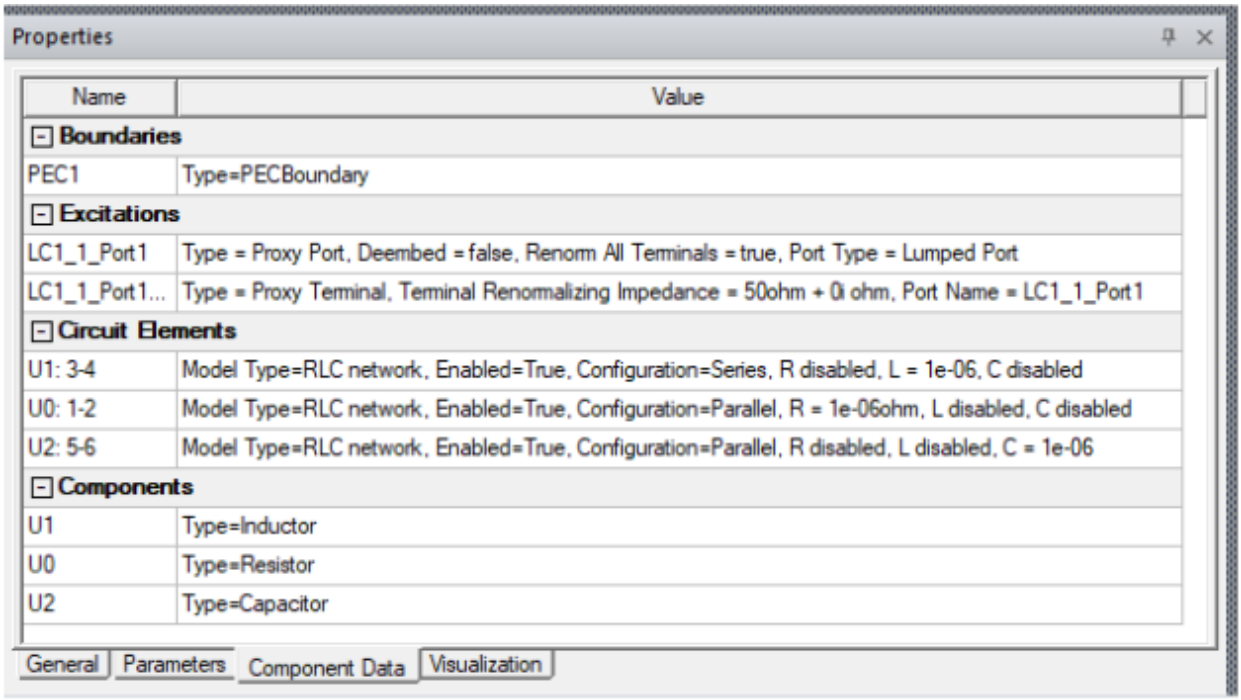
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The names of the circuit elements are also displayed on top of the visualization. When the circuit elements become too small to display due to zooming out, these names disappear to improve the clarity of the window. This visualization of RLC circuits with their names can be turned on/off by toggling the layout component's **Show Layout** option.

This property page will display the information about each boundary, excitation, circuit element, and component group of the layout component.

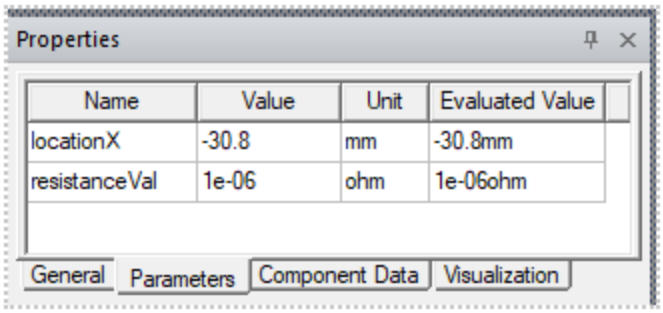
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Parameter Support for Layout Components

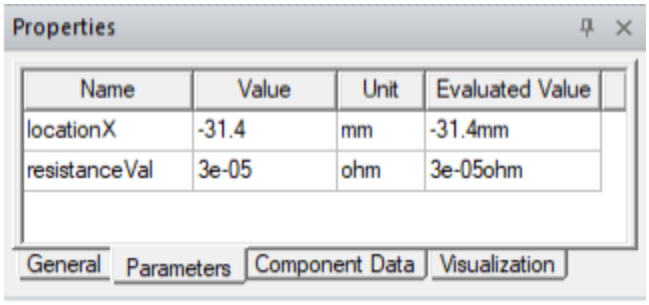
The circuit element visualization and component data property page supports parametrization. If the location of a circuit element or a part of the description in the component data is controlled by a parameter, changing this parameter will cause the visualization or the property page to update and reflect the change.

For example, in the above example, the layout component has two variables as shown below.

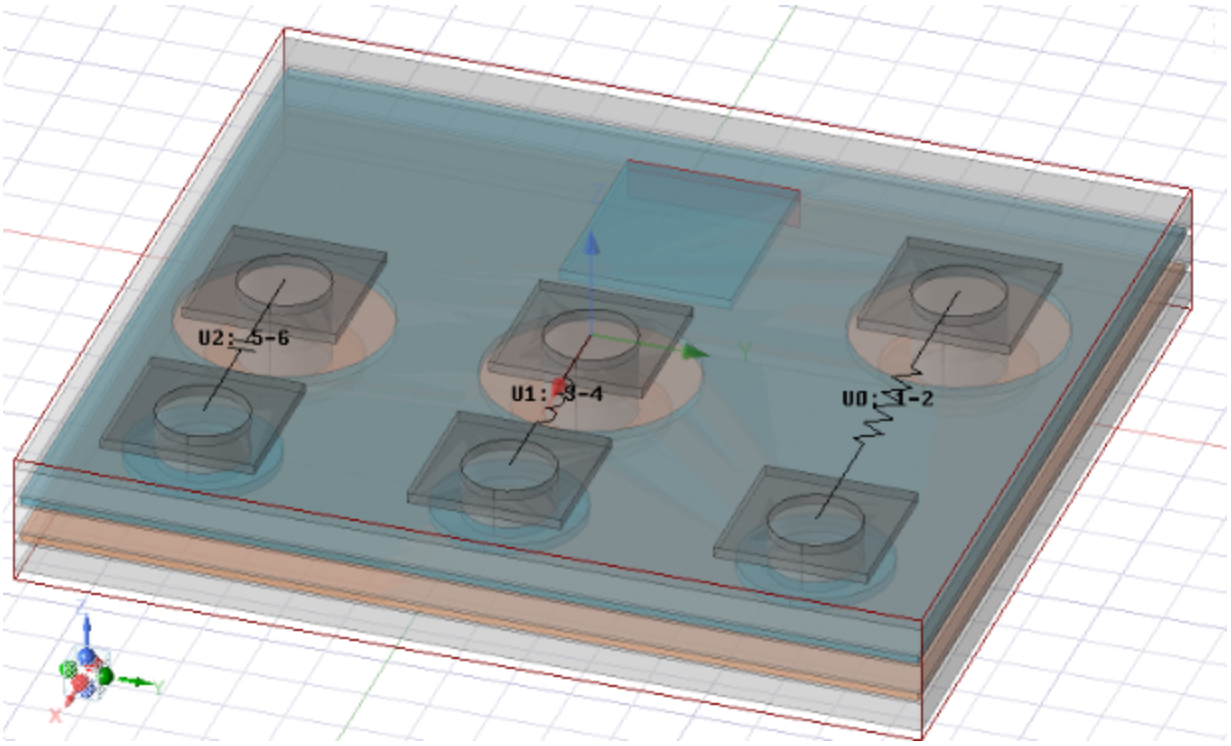


The **locationX** variable controls the x-location of one of the pins in circuit element U0: 1-2. The **resistanceVal** variable controls the electrical resistance in circuit element U0: 1-2. You could change the value of these two variables shown below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

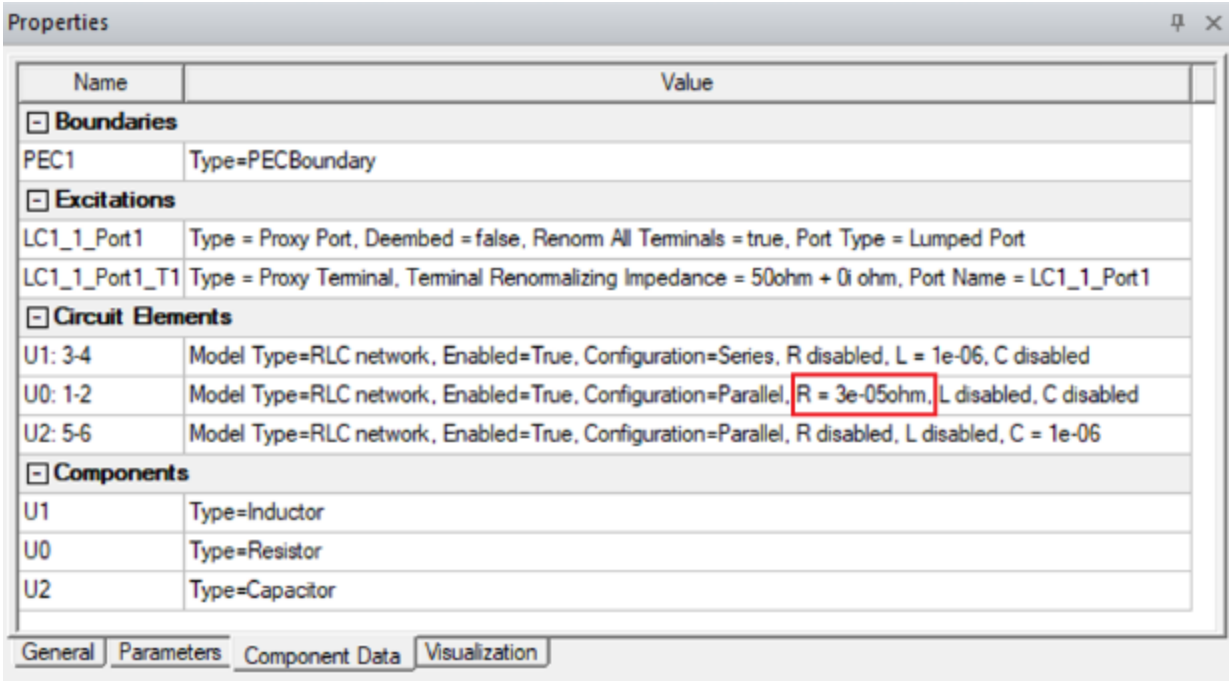


With this variable change, the component's circuit element visualization updates to reflect this change.



The Properties page for affected Circuit Elements also updates.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



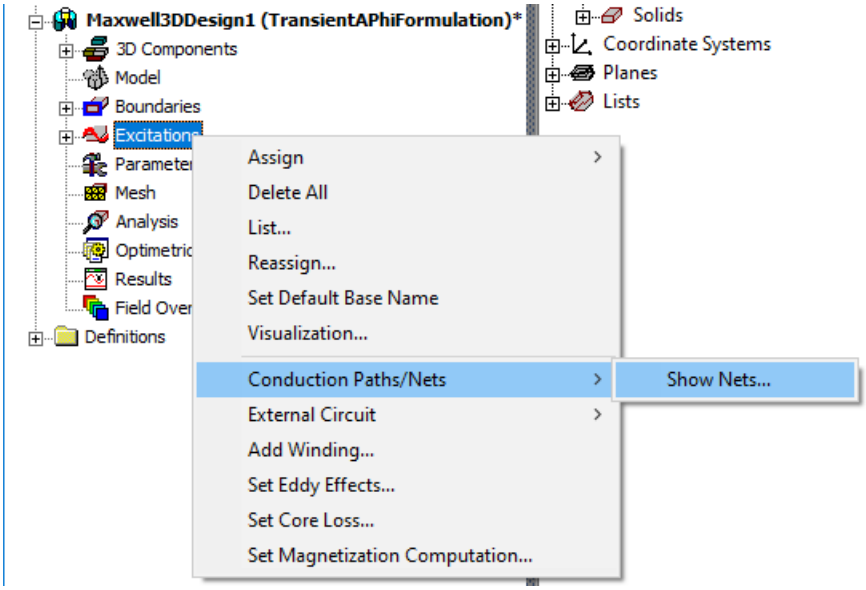
Show Nets for Layout Components

For Maxwell A-Phi designs that include Layout Components, you can use the **Show Nets** command to visualize conducting nets and terminal associations formed by 3D geometries and layout components in a Maxwell design. In the **Net Visualization** dialog box:

- All nets can be correctly identified including 3D geometries and layout components with correct terminal associations.
- All the functionalities are supported for nets containing layout component objects.
- Tools included in the Net Visualization dialog box help users quickly select specific nets containing layout component objects.

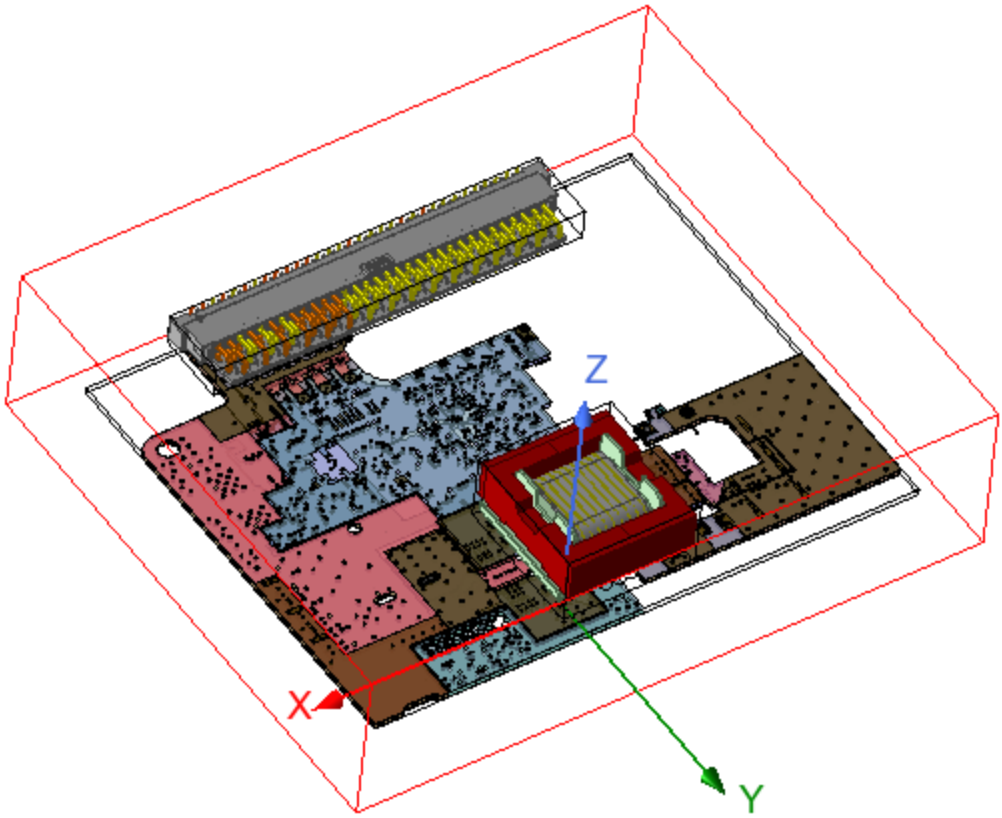
The **Show Nets** command can be accessed from the **Maxwell** > **Excitations** > **Conduction Paths/Nets** > **Show Nets** menu and by right-clicking on **Excitations** in the Project Manager tree.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Selecting **Show Nets...** displays the **Net Visualization** dialog box that lists the nets and associated terminals and includes any included Layout Components. You can use the dialog box to select any net or terminal. You can continue to work in the Modeler window with the dialog box open. Consider the following example design:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

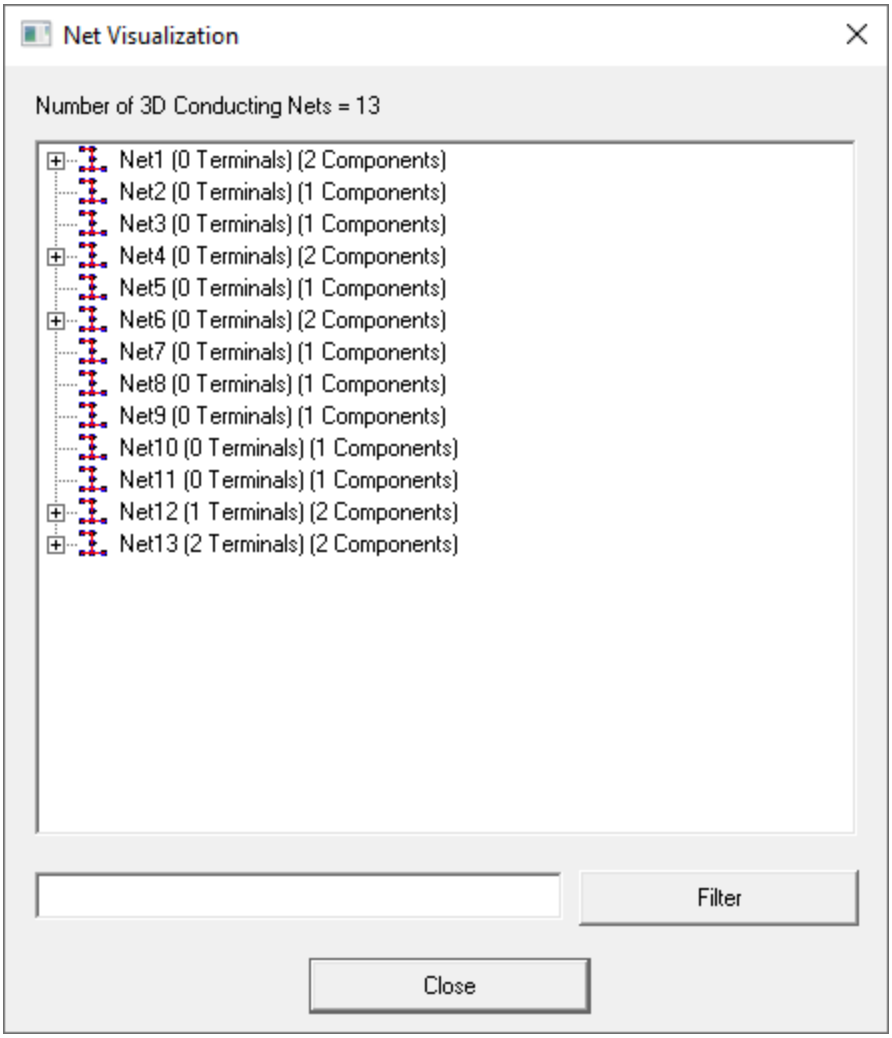
For a net to appear in the **Net Visualization** dialog box, the conductivity of the assigned material must be equal to or greater than 20,000 siemens/m, or the **Resistance** must be less than 3.142 ohms. Conductivity is set by material assignment (in the **3D Modeler** window or a boundary condition), and **Resistance** is set by boundary conditions (e.g., **Impedance** or **LumpedRLC**).

In the **Net Visualization** dialog box, you can sort the nets in ascending or descending order, relative to the number of terminals. This helps you locate GND nets, which have no terminals.

Text at the top of list box provides the total number 3D conducting nets. Naming of nets is automatic and follows the convention: "Net1 (0 Terminals)", "Net2 (1 Terminals)"…"Net<N> (<T> Terminals)". Net Names are not editable.

The **Net Visualization** dialog box is shown below.

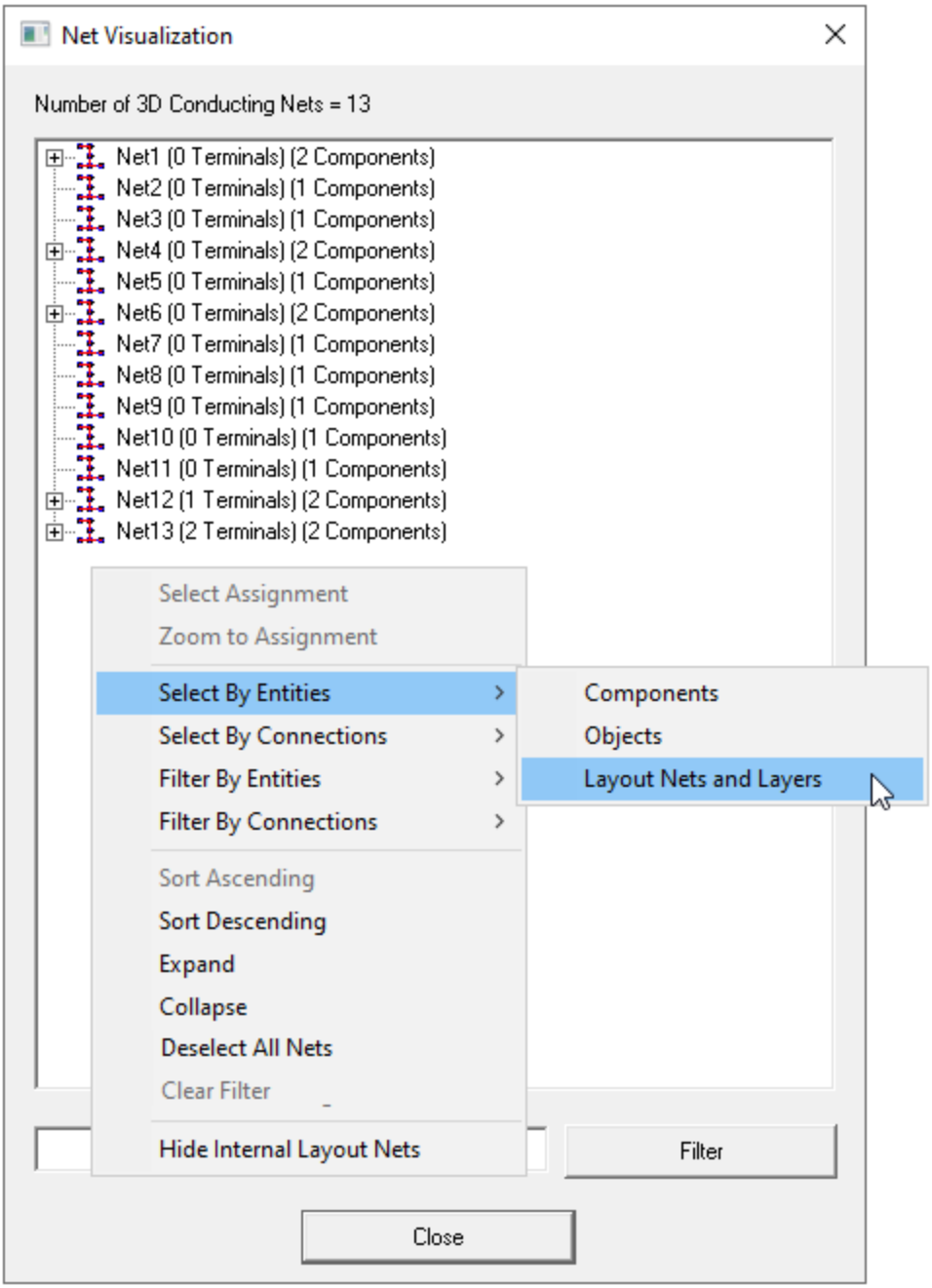
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For designs with many nets, you can enter text and wild cards in the field, and click the **Filter** button.

Right-click in the **Net Visualization** dialog box to view the shortcut menu. The menu is context sensitive, depending on whether you click on a particular net object, or in the white area:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Select Assignment**: If you right-click over selection in the tree, selects entities present in the selection. If the right-click menu is invoked on the surrounding blank space, this menu item is disabled as there is no selection.
- **Zoom to Assignment**: If you right-click over a selection in the tree, zooms to the entities present in the selection. If you right-click on the surrounding blank space, this menu item is disabled as there is no selection.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Select by Entities**: Invokes a dialog box containing object, component, or net and layer names present in the selection if the right-click menu is invoked over some selection in the tree. If the right-click menu is invoked on the surrounding blank space, all object or component names present in the project are displayed. Selecting one or multiple of these object or component names selects all the nets that contain any of the selected entities.
- **Select by Connections**: If you right-click over some selection in the tree, this produces a dialog box containing object or component names present in the selection. If you right-click in the surrounding blank space, a dialog box with all object or component names present in the project displays. Selecting a pair of these objects or component names selects all the nets that contain this pair of entities, signifying a direct or indirect connection between these entities.
- **Filter by Entities**: A new dialog box is invoked containing object, component, or net and layer names present in the selection if the right-click menu is invoked over some selection in the tree. If the right-click menu is invoked on the surrounding blank space, all object or component names present in the project are displayed. Selecting one or multiple of these objects or component names filters all the nets that contain any of the selected entities.
- **Filter by Connections**: A new dialog box is invoked containing object or component names present in the selection if the right-click menu is invoked over some selection in the tree. If the right-click menu is invoked on the surrounding blank space, all object or component names present in the project are displayed. Selecting a pair of these objects or component names filters all the nets that contain this pair of entities, signifying a direct or indirect connection between these entities.
- **Sort Ascending**: Sorts the nets in ascending order of their names.
- **Sort Descending**: Sorts the nets in descending order of their names.
- **Expand**: Expands the selection in the tree if the right-click menu is invoked over some selection in the tree. If the right-click menu is invoked on the surrounding blank space, it expands all the nets.
- **Collapse**: Collapses the selection in the tree if the right-click menu is invoked over some selection in the tree. If the right-click menu is invoked on the surrounding blank space, it collapses all the nets.
- **Deselect All Nets**: Deselects all the entities present in the project. Clears the selection of all the previously selected entities.
- **Clear Filter**: Clears all the filtering applied on nets and shows all the nets.
- **Hide Internal Layout Nets**: Hides all the nets that are internal to layout component

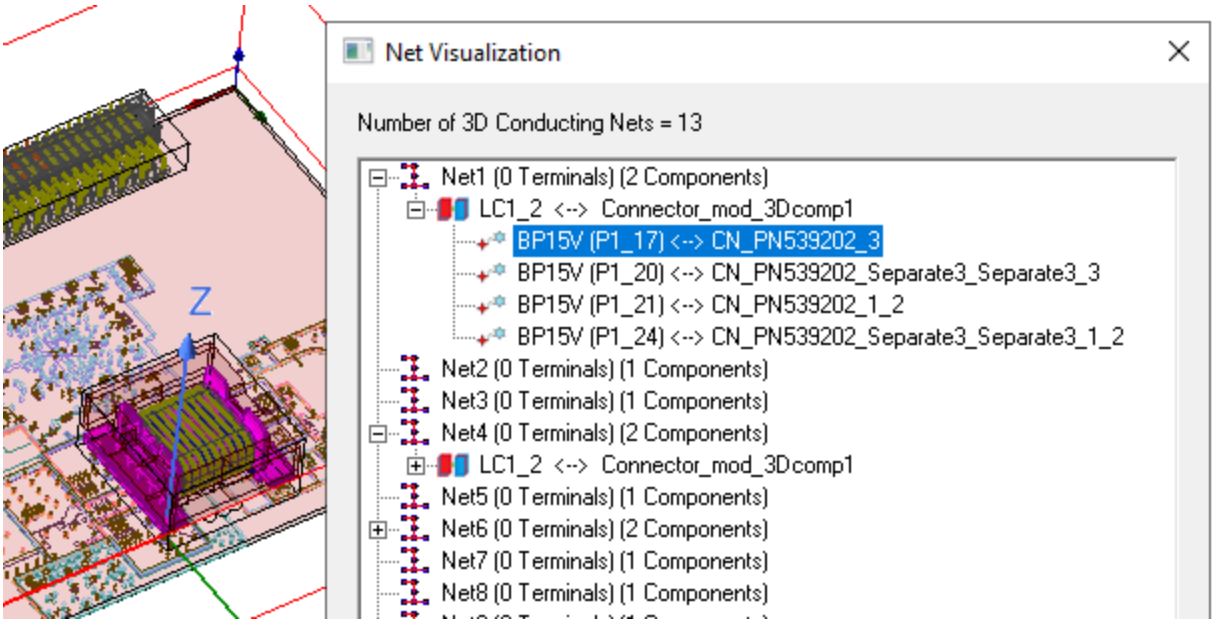
Along with the **Filter by Entities** and the **Filter by Connections** menu items, you can also filter nets using text input. The text box for this functionality is shown below:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Features of this text box are

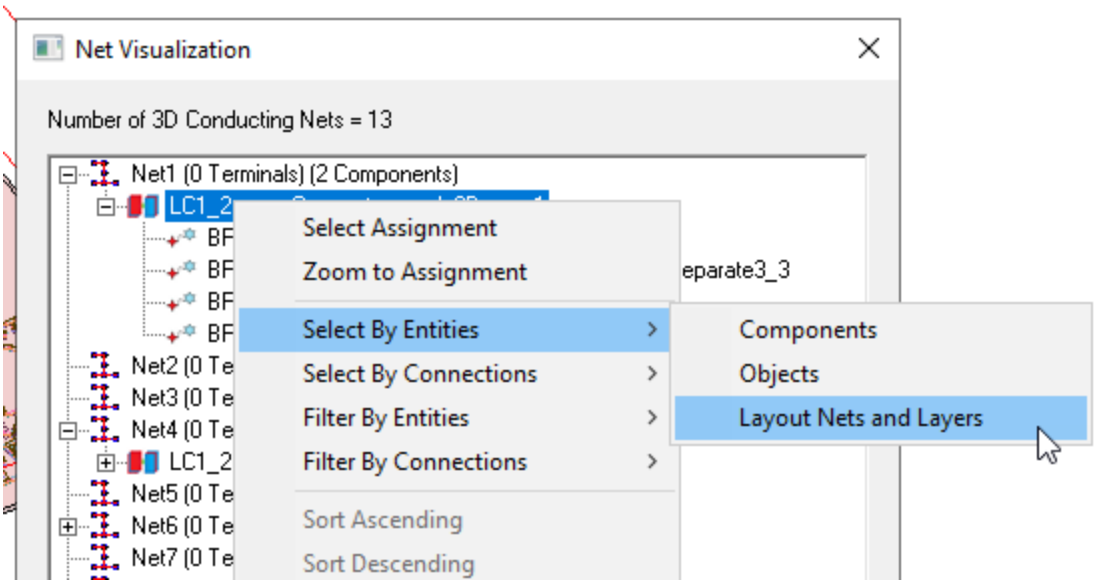
- It accepts one or more object or component names.
  - Multiple entries can be entered by separating them with comma operator (“,”).
  - Entity connections can be entered by separating entity names with dash operator (“-”).
  - Wild cards “*” and “&” can be applied. For example, LayoutComp1 and HfssComp2 can both be represented simultaneously as “*Comp*”. Similarly, LayoutComp1 and LayoutComp2 can be represented simultaneously as “LayoutComp&”. For example, (Box1, Box2, Comp1 – Comp2): This string applies filter based on if a net contains Box1, or Box2, or connection between Comp1 and Comp2.

Selecting any component or net in the **Net Visualization** list highlights it. For example, the following figure shows a Layout Component selected in the expanded tree list and highlighted in the **Modeler** window.



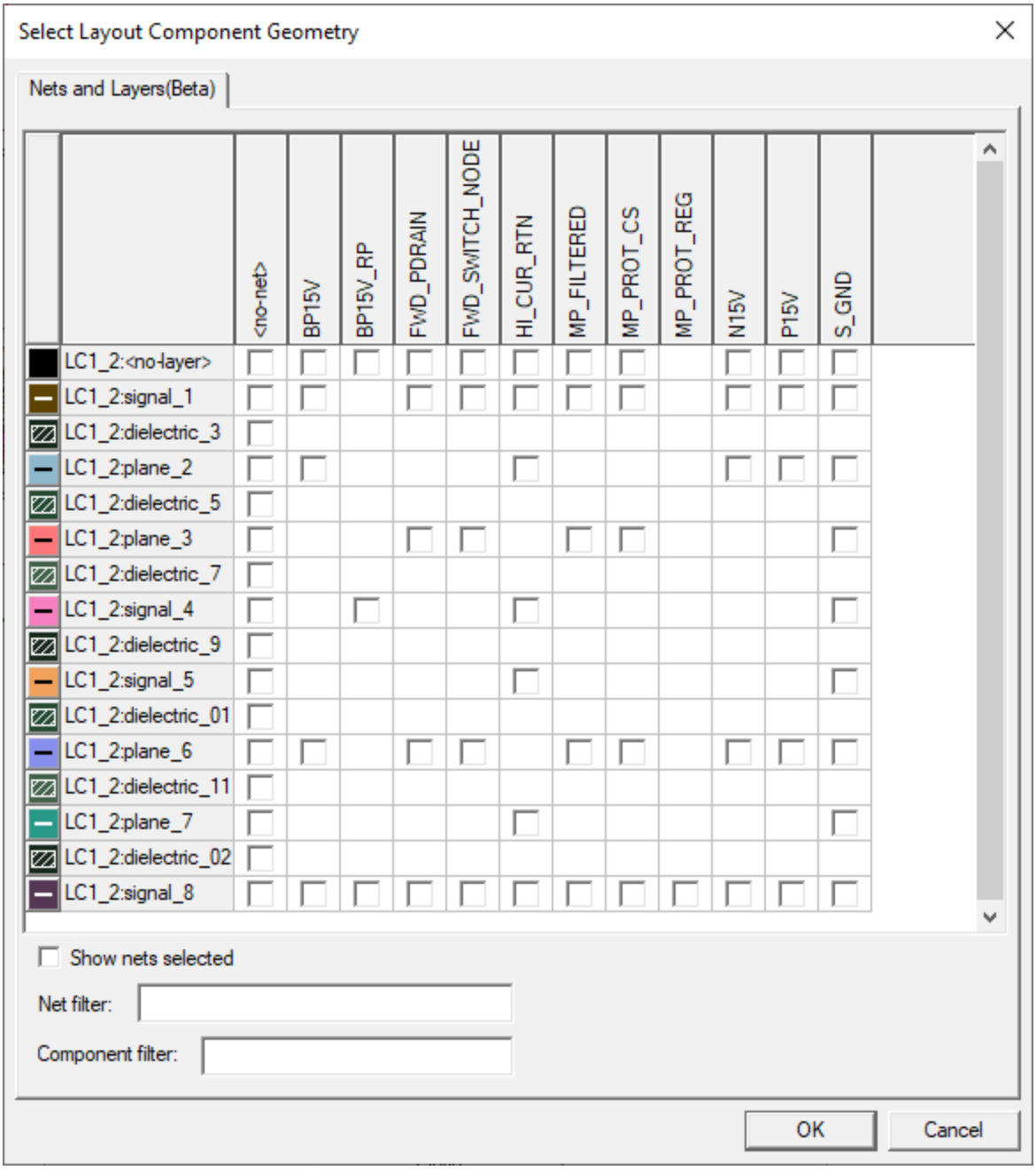
For example, right-click on the Layout Component name in the **Net Visualization** dialog box to display the menu and select **Select by Entities** > **Layout Nets and Layers**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This displays the **Select Layout Component Geometry** dialog box:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



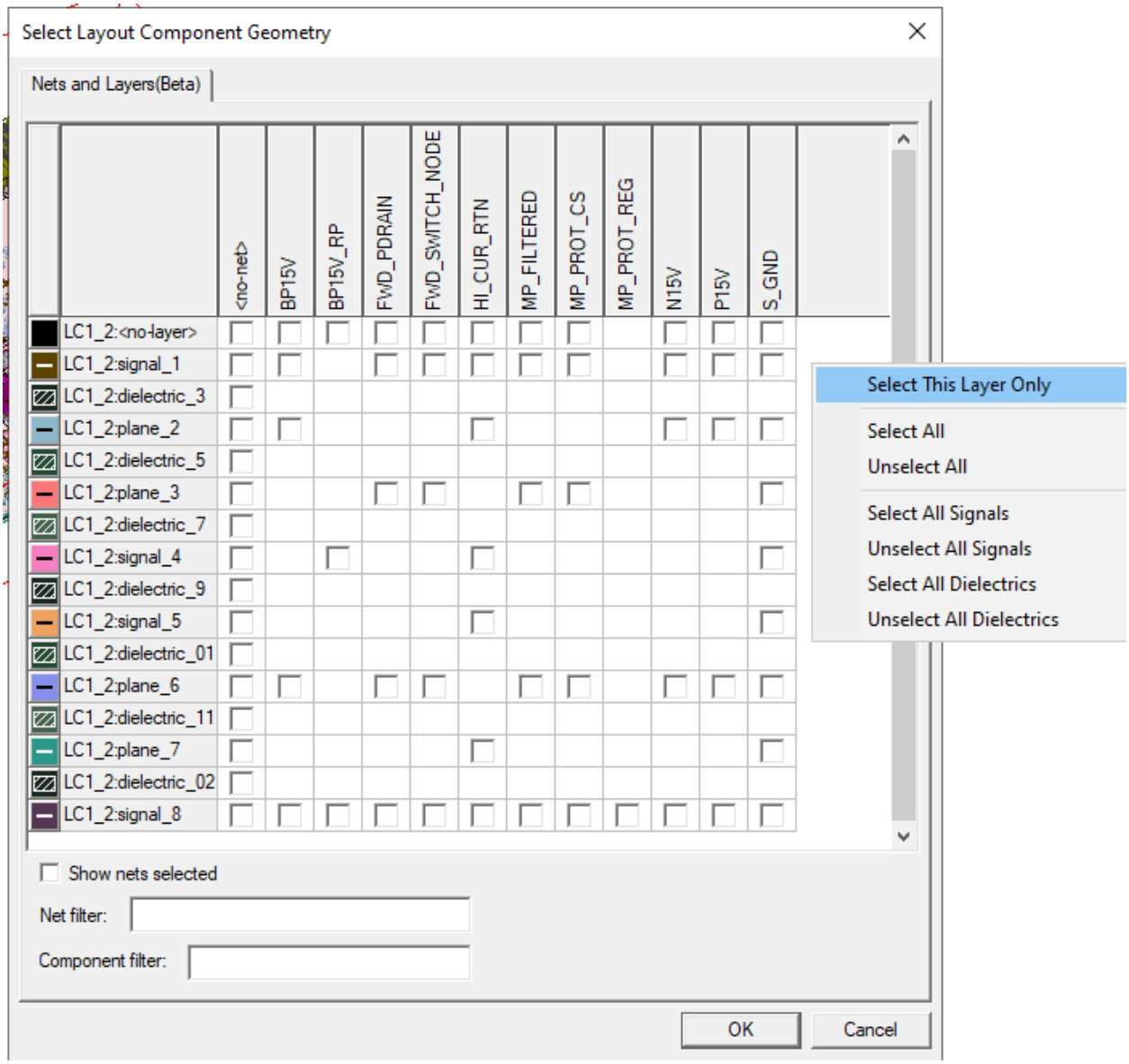
With the **Select Layout Component Geometry** dialog box, you can select objects from layout components' local nets and layers and search for nets in the overall design that contain these objects. This dialog box displays all the layers and local nets of every layout component in the design. If there are objects in the layout component that belong to a specific layer and net, a checkbox will become available for you to select. Once you make selections and click **Apply**, you can go back to the parent **Net Visualization** dialog box and click **Select** button, which selects all the nets in the **Net Visualization** dialog box that contain these objects.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Select Layout Component Geometry** dialog box is modeless, that means you can work in the 3D Modeler while keeping this dialog box open. This way, you can use the modeler window to better find the layout component objects you want to search for.

By clicking the column headers, you can select or deselect check boxes in the entire column. The **Show nets selected** check box can be toggled to move all the selected nets to the left side of the table. Use the Net filter and Component filter to filter fields to designate specific nets and components. These filters support '*' and '?' wildcards and can delimit multiple filters with comma. The table is refreshed each time users update the filter text.

Additionally, the **Select Layout Component Geometry** dialog box has a right-click menu as shown below.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can use the dialog box to select any net or terminal. You can continue to work in the Modeler window with the dialog box open.

The **Net Visualization** dialog box identifies all nets and will include both 3D geometries and layout components with correct terminal associations. As shown in a previous figure, any net you select in the Net Visualization list highlights the objects in the net in the modeler window. By default, 3D geometries are highlighted in blue, and objects from layout components will be highlighted in red. The rest of objects in the layout component not belonging to the selected net will be shown as wireframes.

All the functionalities in the Net Visualization dialog that works for regular nets are also supported for nets containing layout component objects, such as **Zoom To**. Because layout components do not have any actual geometries in the modeler window, the **Select Assignment** will only select 3D geometries in each net.

Replacing or Editing Layout Components

Use the **Replace Component** command to edit the properties that were specified when a layout component was inserted into your design or to replace the component with a different one. This command is located in the shortcut menu that appears when you right-click a layout component entry in the Project Manager tree's **3D Components** folder. A multi-tab window appears, *Edit Layout Component*, which is very similar to the *Insert Layout Component* window discussed in Adding a Layout Component except for the following differences:

- None of the tabs have the **Target Coordinate System** parameter. If you would like to change the placement of a layout component, see Edit Placement of a Layout Component.
- The **General** tab of the *Edit Layout Component* window includes a **Definition** drop-down menu for selecting an AEDB folder, *.aedbcomp file, or existing component definition in the current design. Use this drop-down menu to replace the current component with a different one.
- In the **Variable Mapping** tab of the *Edit Layout Component* window, you cannot revert previously mapped instance parameter variables to the original numeric values as you can in the *Insert Layout Component* window. Once the mapping is committed by completing the insertion process, the variables remain as instance parameters.
- The **Parameters** tab of the *Edit Layout Component* window does not include an option to make local design or project variables from component instance parameters. You can only create local design or project variables when originally adding the layout component to your design. Even if you have chosen to map additional source layout variables to instance parameter variables when editing a component, you cannot automatically create associated local design or project variables from the *Edit Layout Component* window.
- The tab locations differ between the two windows. For *Insert Layout Component*, the tabs are located near the bottom of the window. For *Edit Layout Component*, the tabs are

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

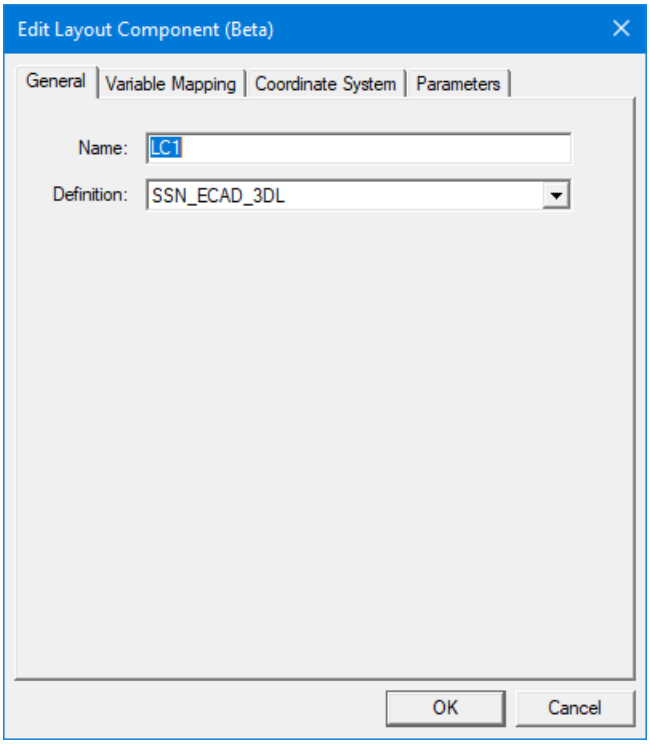
located near the top of the window. However, the same tabs are included in both windows, and they appear in the same order.

Edit/Replace Procedure

To edit an existing layout component in your Maxwell design, or to replace it with a different component, do the following:

1. Under *3D Components* in the Project Manager, right click the ***<Layout_Component_Name>*** and choose **Replace Component**.

   The *Edit Layout Component* window appears with the **General** tab initially selected:



2. Optionally, to select a different component to replace the current one, choose *one* of the following three options from the **Definition** drop-down menu:
   - **Import .aedbcomp File:** Choose this option if you have exported an HFSS 3D Layout as a single-file Layout Component (*.aedbcomp)

     A *Select layout component file to import* dialog box appears in which you can browse to the file location and select and open it.
   - **Browse to Project .aedb Folder:** Choose this option to specify the location of an Ansys electronics database folder (*.aedb) to use in the creation of the layout component. This folder is where HFSS 3D Layout design filesets are stored.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The layout may contain multiple designs. In the *"Select layout component folder to import"* dialog box that appears. Browse to the desired *.aedb folder as follows:

- Double-click the folder to open it and continue drilling down through **Layout Components\<*Design_Name*>\<*Design_Name*>.aedb** until a single **Stride** folder is the only thing contained in the selected folder. Then click **Open**.

- Layout component definitions that have previously been added to the current project will also be listed in the *Definition* drop-down menu. You can **choose an alternative component**, if any exist, to replace the one being edited.

3. Progress through the **Variable Mapping**, **Coordinate System**, and **Parameters** tabs of the *Edit Layout Component* window in the same manner as detailed in the Adding a Layout Component topic. Take into account the limitations described above.
4. Click **OK** when you are done editing the layout component.

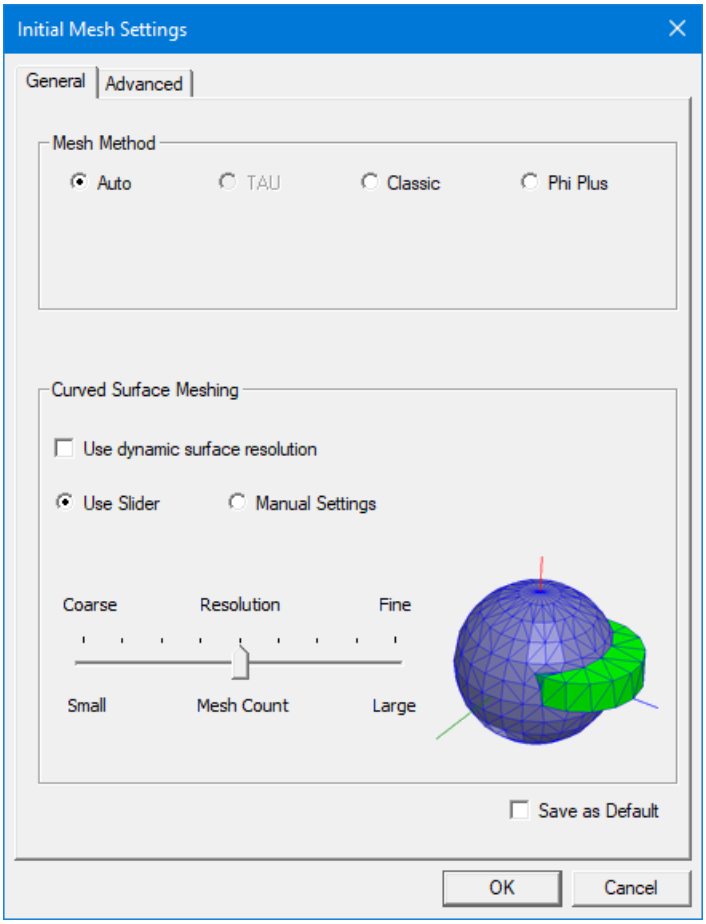
If you selected a new source file or folder for the layout component, a new layout component is added under *Definitions* > *Components* in the Project Manager. Additionally, the layout component in the Modeler window is regenerated.

> **Note:**
>
> If the component's visualization options were customized in the design, they will be reset to their default configuration whenever the layout component is replaced or regenerated after editing. When this occurs, repeat step 8 in the Adding a Layout Component procedure to restore your preferred visualization options.

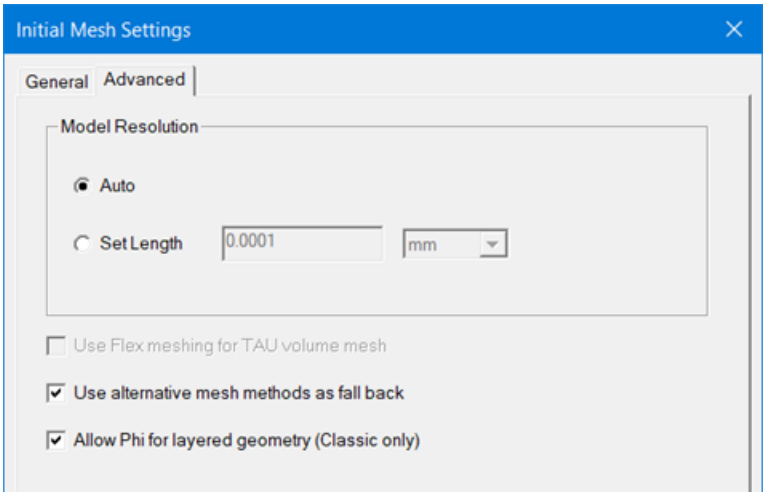
Meshing with a Layout Component

If the Maxwell design has layout components, its Initial Mesh Setting window is modified. The window will disable the TAU mesh method and add an additional mesh method called Phi Plus:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



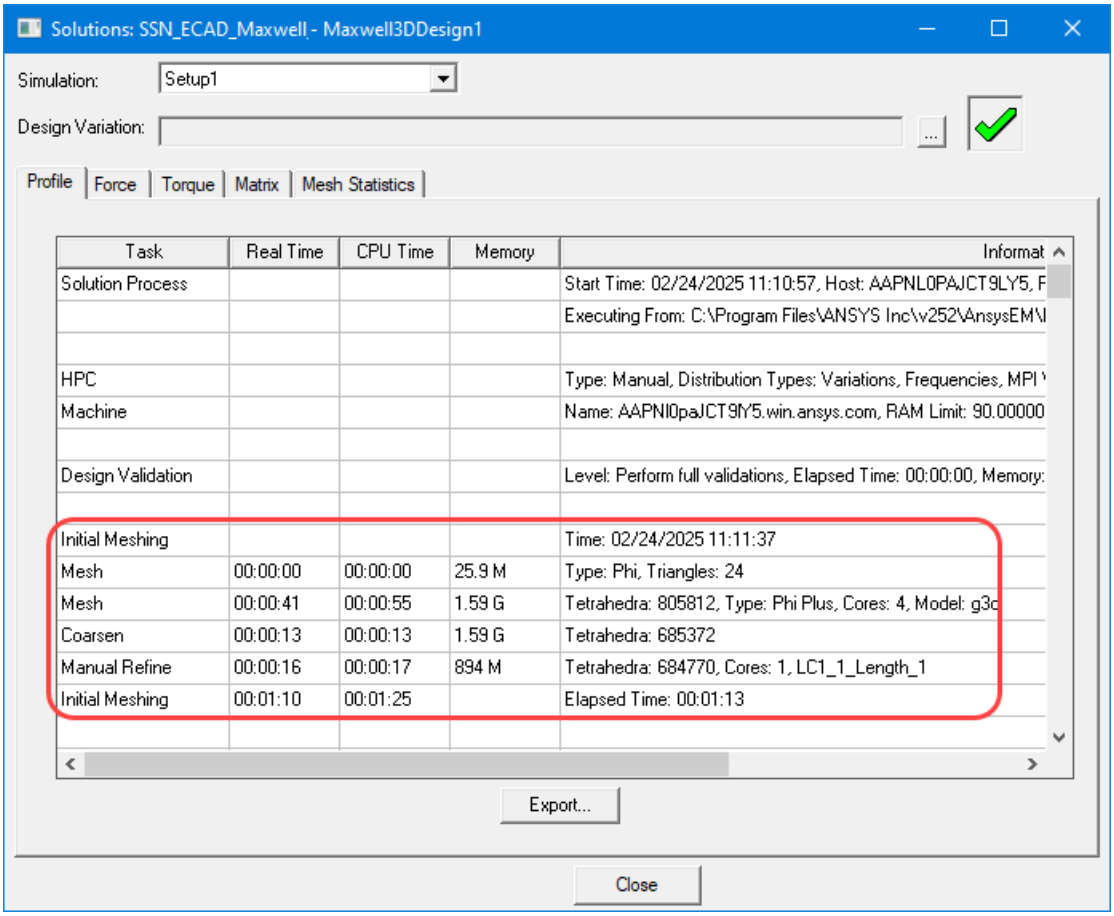
The options related to TAU in the **Advanced** tab are disabled.



Mesh Method Recommendations:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For imported layouts with layered geometry that is drawn in the XY plane and swept in the Z direction, the Phi mesher is recommended (activated by selecting **Classic** on the **General** tab and **Allow Phi for layered geometry** on the **Advanced** tab). This mesher is not suitable for geometry with faces that are not oriented in a vertical or horizontal direction.
- For complex assemblies that is not suitable for the Phi mesher, the Phi Plus mesher is recommended.
- If an assembly fails to mesh with the Phi or Phi Plus mesher, the Classic mesher can be used.
- If the Initial Mesh Settings **General** tab is left to **Auto**, the software will determine the best mesh method. The mesh method used for a solution is reported in the Solution Data**Profile** tab, which is accessed by right-clicking on **Results** in the Project Manager.



Mesh Linking

To save meshing time, you can set up a meshing link between an AC Magnetic ECAD model and an equivalent transient model. To link between the AC Magnetic and the transient model:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The source and target design must be located in the same project.
- The list of layout component instance names should be the same.
- The layout component instance with the same name should use the same layout component definition.

Specifying Initial Mesh Settings

Simulating a 3D Layout Component with an A-Phi Solver

With the Transient or AC Magnetic A-Phi solver, Maxwell 3D can run an analysis that includes a 3D layout component. This feature makes it easier to simulate electronic applications such as multilayer PCBs to calculate the Lorentz force due to trace currents and external magnetic fields.

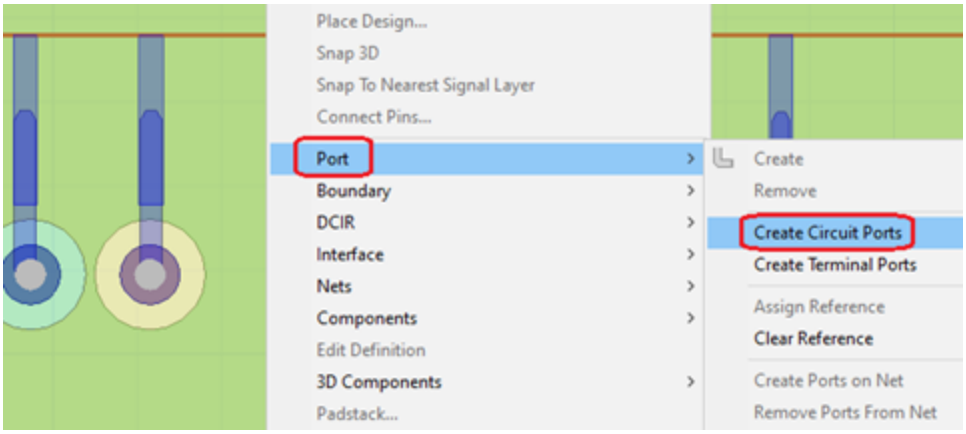
Running an analysis on a design with a 3D layout component requires the use of both HFSS 3D Layout and Maxwell 3D products. You first create the layout component with excitations, mesh operations and RLC components defined in HFSS 3D Layout. Then, you must import the component in a Maxwell 3D design. Maxwell will automatically import the layout excitations and circuit components (RLC). You need to select the type of the excitations (voltage or current), and enter certain information related to each excitation type. The rest of this section will explain the details of the entire design flow of a Maxwell design with the 3D layout component.

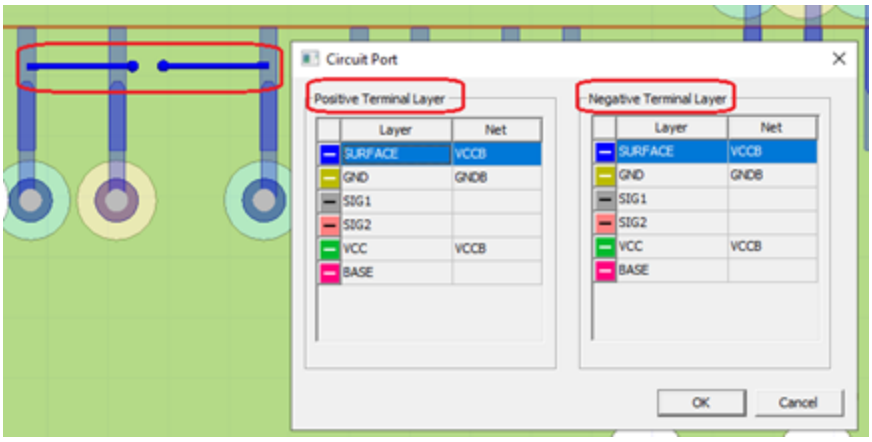
Creating a Layout Component in HFSS 3D Layout

You can either create a component from scratch or can import third-party layout design data (Refer to *Importing Layout Design Data* in the HFSS 3D Layout help). You can use all capabilities of the Layout Editor such as geometry check, shorts/opens in a design.

You need to define excitations in the Layout design by creating circuit ports. Existing port definitions are still valid (refer to *Circuit Ports and Elements* in the HFSS 3D Layout Help). In the Layout Editor, one way to create a circuit port is to right-click the geometry view, select **Port** > **Create Circuit Ports**, select two points, and then select a **Positive Terminal Layer** and **Negative Terminal Layer** in the pop-up dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

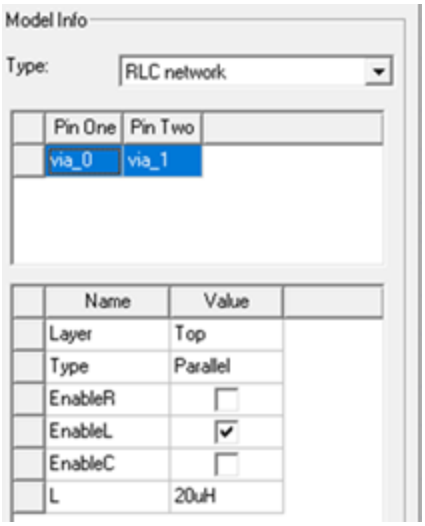




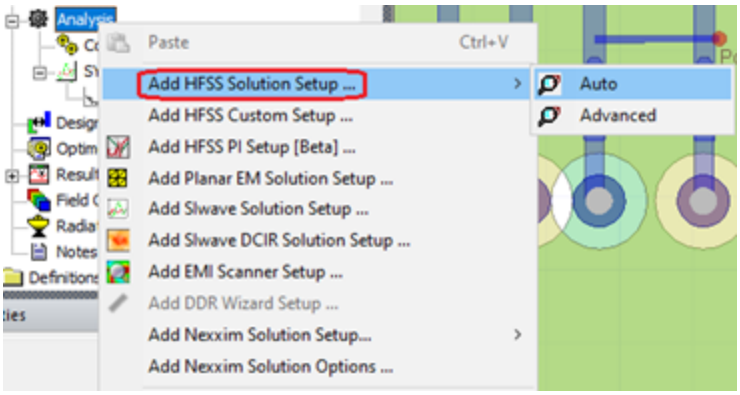
> **Note:** Maxwell does not use any information defined in the Port definition dialog box. You will specify the excitation information in the Maxwell interface.

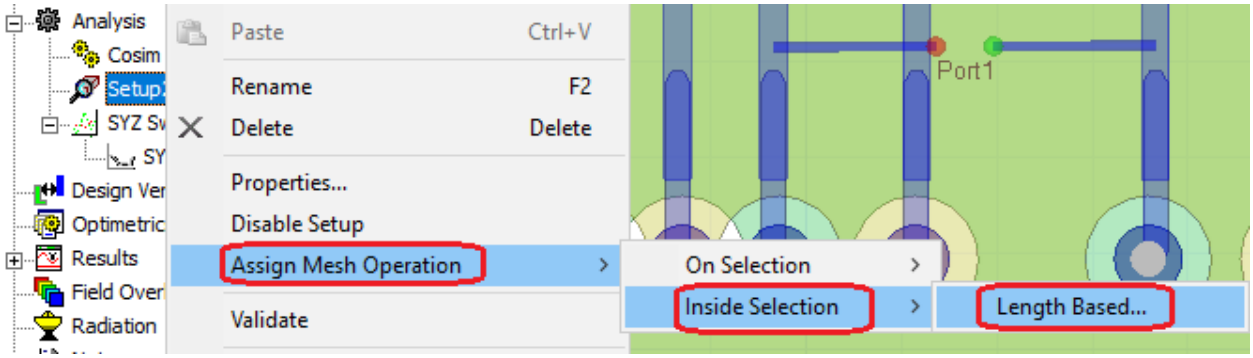
You can add lumped circuit elements into the design. For RLC support, you define a component for each lumped element (R, L, or C). The component has two pins and associated padstacks. Each component must have only one lumped element type (enable only one, as shown). Note that serial or parallel connection of RLC is not supported, so Maxwell will ignore the Type selection.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



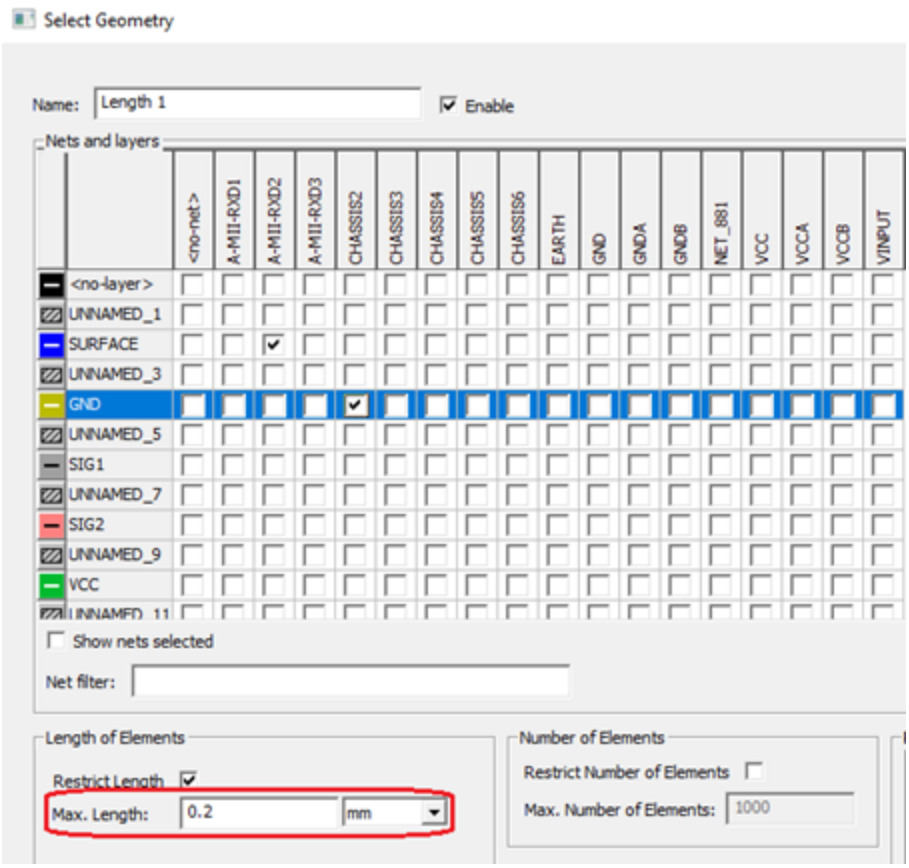
Mesh operations for the layout design should be defined in the layout. To assign mesh operations, first add an HFSS solution setup. Then right-click the solution setup and select **Assign Mesh Operation**:





Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the Select Geometry dialog box, you can specify the Max Length to refine the mesh. Note that the solution setup is not used for the solution of the design.



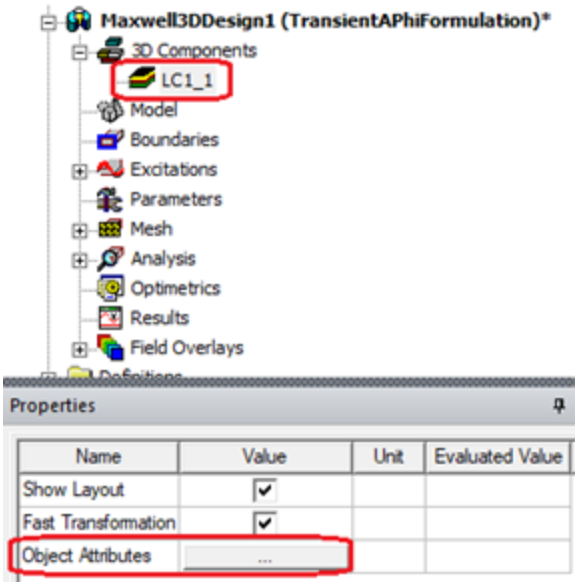
Once the design is complete, you can save it as a component or just save the layout design. When the component is imported in Maxwell, the design data including layout model (with virtual geometry view), excitation assignments, mesh operations, RLC data, and materials will be automatically imported into Maxwell. Pin groups are imported as excitations, but the pin group information is still available for simulation.

Importing a Layout Component to Maxwell 3D

To use a 3D layout in the Maxwell solver, you must import it. For information about importing a layout component into Maxwell, see Adding a 3D Layout.

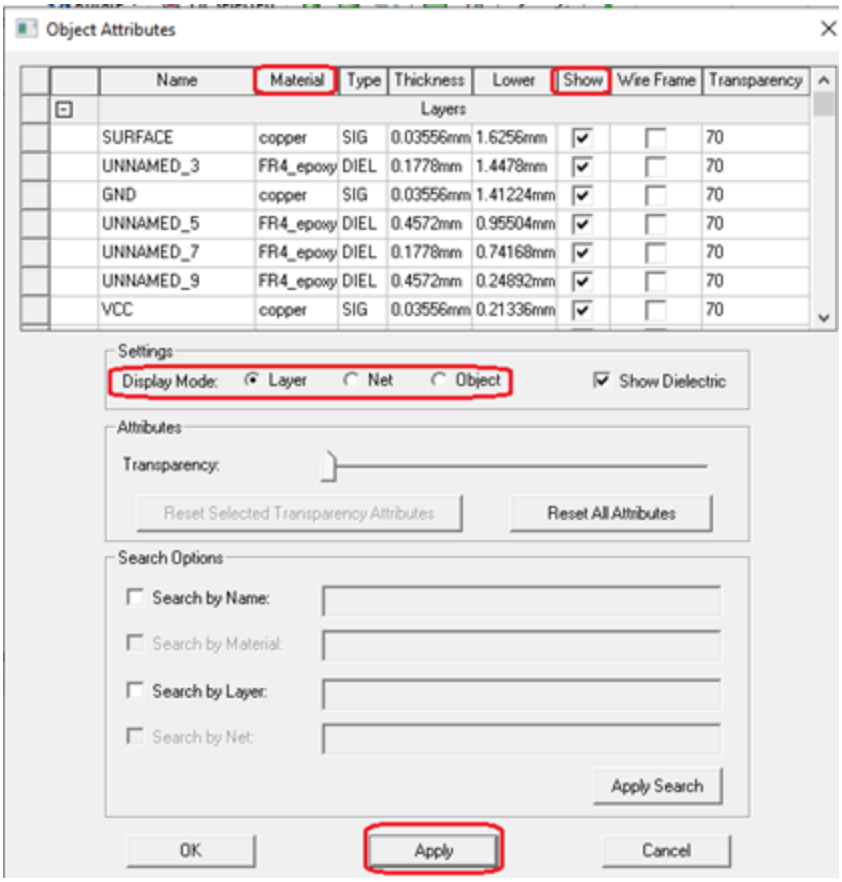
Once you have imported a 3D layout, you can view object attributes: click **Object Attributes** in the component property window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

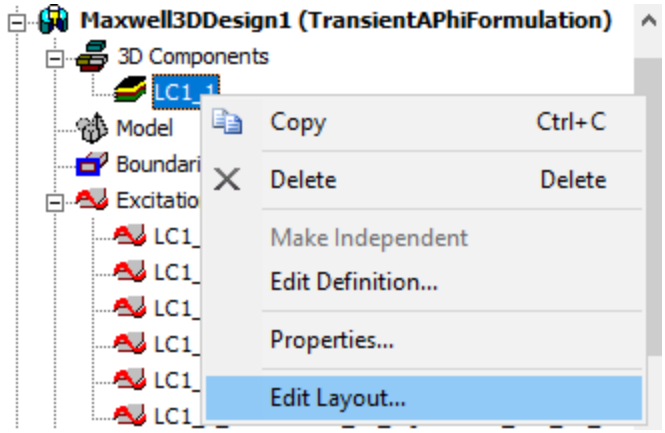


In the **Object Attributes** dialog box, you can view object attributes such as: **Name**, **Material**, **Type**, and so on. There are different display modes: **Layer**, **Net**, and **Object**. In the **Show** column, after changing the object visibility, you can see the checked objects in the geometry view by clicking **Apply** button. See Visualization for Layout Components for more details.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



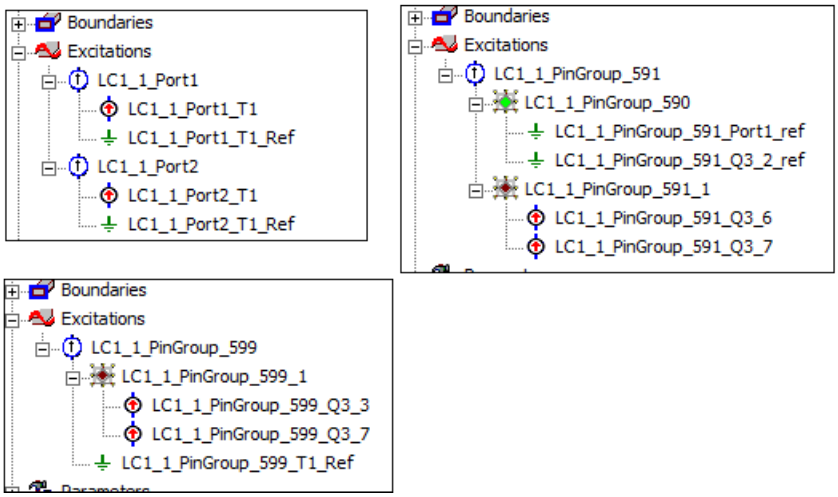
You can modify the Layout component. To edit the layout, right-click on the component, and select **Edit Layout...** This will open the layout design of the component in a new project. You can then modify the design in the new project. Once the modified design is saved, Maxwell will automatically update the layout component.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note: Edit Definition...** will launch the **Create Layout Component** dialog box in which you can choose another layout design.

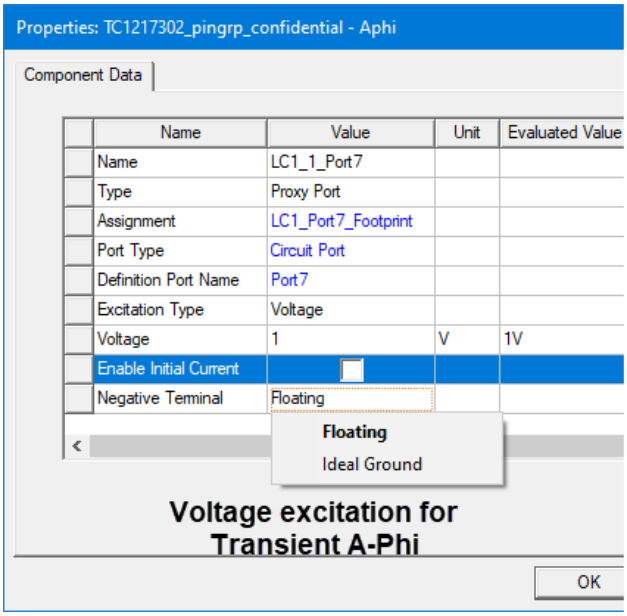
After a layout design is imported into Maxwell, the layout circuit ports will be converted into a *Proxy Port* in Maxwell. Each proxy port has two nodes. One node is positive (current flows into the conductor), and the other one is negative (current flows out of the conductor). A node can be a terminal or a pin group (group of terminals). All terminals of a pin group are parallel connected, so they have the same potential value. Maxwell treats a terminal as an equipotential surface. Examples of port definitions are shown below:

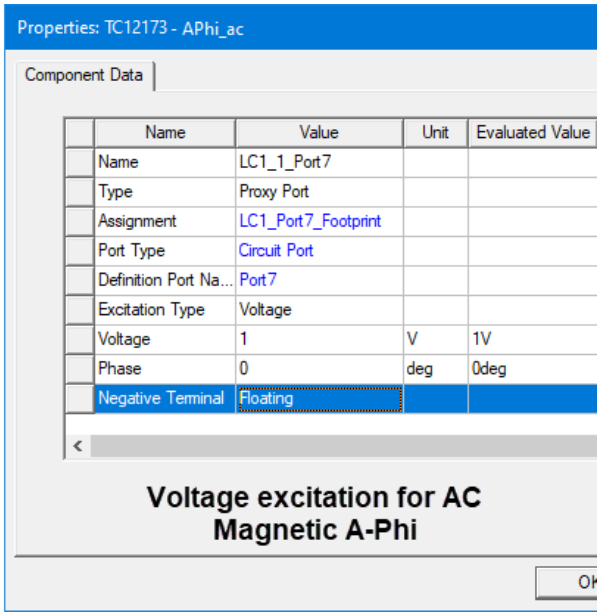


Port excitation data can be modified for different excitation types. The **Excitation Type** can be **Voltage** or **Current**.

- For voltage excitation, enter a voltage value (and a phase value for AC Magnetic A-Phi), and select the behavior of the negative terminal (**Ideal Ground** or **Floating**). The default is **Floating**. The solver applies the specified voltage value (and the specified phase value for AC Magnetic A-Phi) as a potential drop between the terminals. If **Ideal Ground** is selected, the negative terminal potential is set to 0 V. For **Floating**, the negative terminal potential is unknown. For the transient solver, you can also specify the initial current value by selecting **Enable Initial Current**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

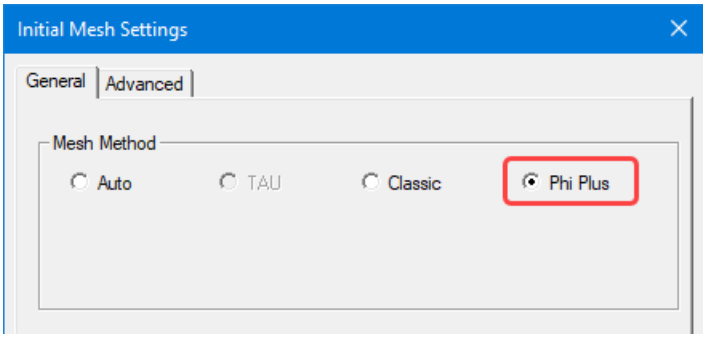




- For current excitation, enter a current value (and a phase value for AC Magnetic A-Phi), and select the behavior of the negative terminal.

> **Note:** The circuit port model in Maxwell is different than the model in HFSS. Maxwell does not introduce a conduction path between the terminals for the current flow.

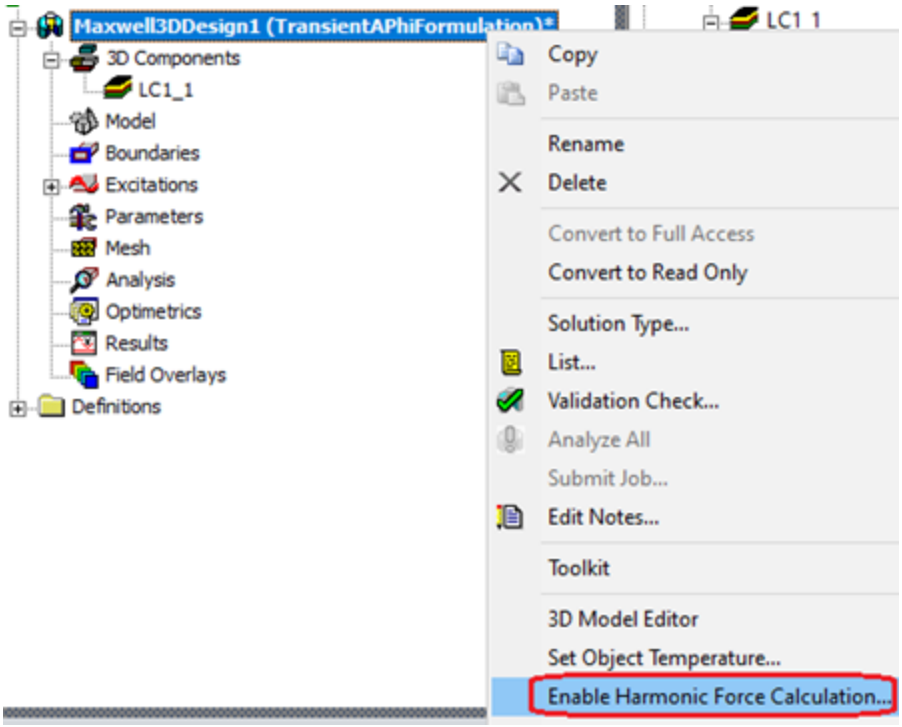
For a Maxwell design with a 3D layout component, the Phi Plus mesher setting is available in the **Maxwell 3D** > **Mesh** > **Initial Mesh Settings** dialog box; it replaces the TAU mesher, which will be grayed out.



You can continue to use existing mesh operations for non-component objects. The suggested mesh method is Phi for a layout component. You can specify this in the Maxwell **Initial Mesh Settings**, see Meshing Recommendations for Layout Component for more details.

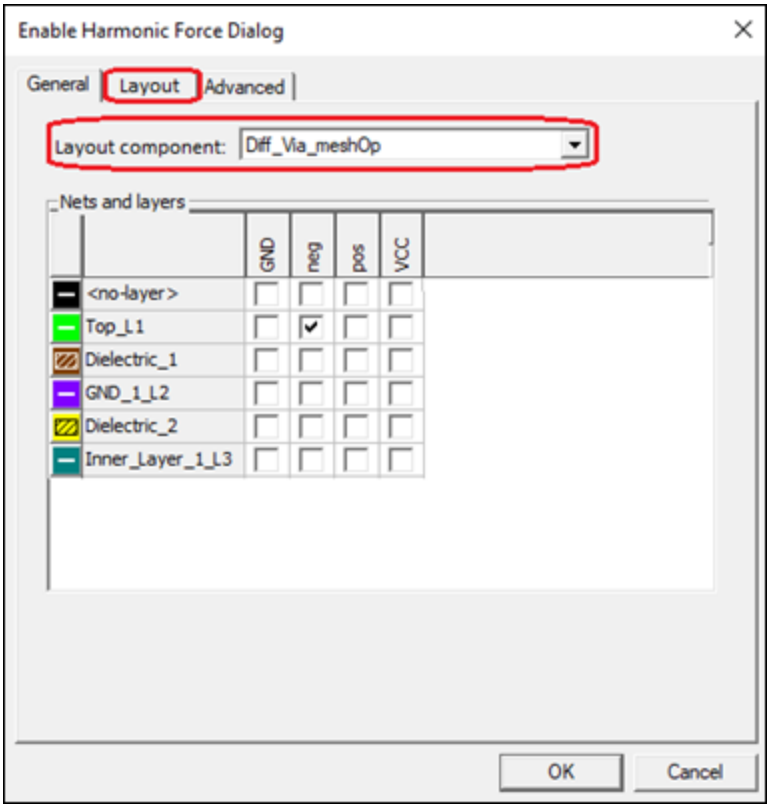
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To calculate harmonic force on objects, right-click the design, then select **Enable Harmonic Force Calculation...**

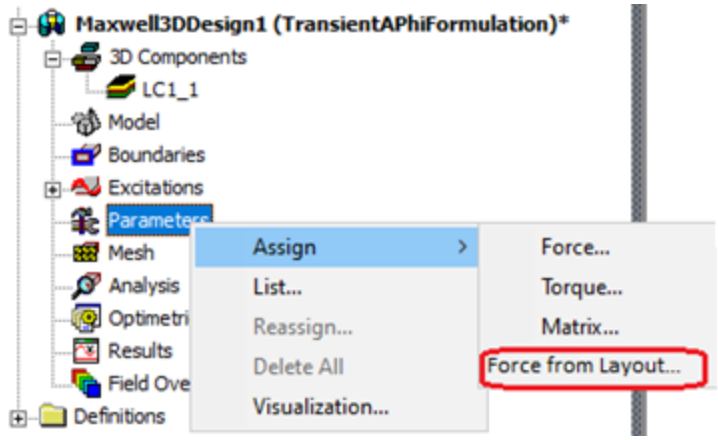


In the **Enable Harmonic Force Dialog** window, on the **Layout** tab, you can select one layout component from the **Layout component** drop-down menu, and select <layer, net> pairs in the following grid, each <layer, net> pair represents the object(s) in the intersection of corresponding layer and net.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

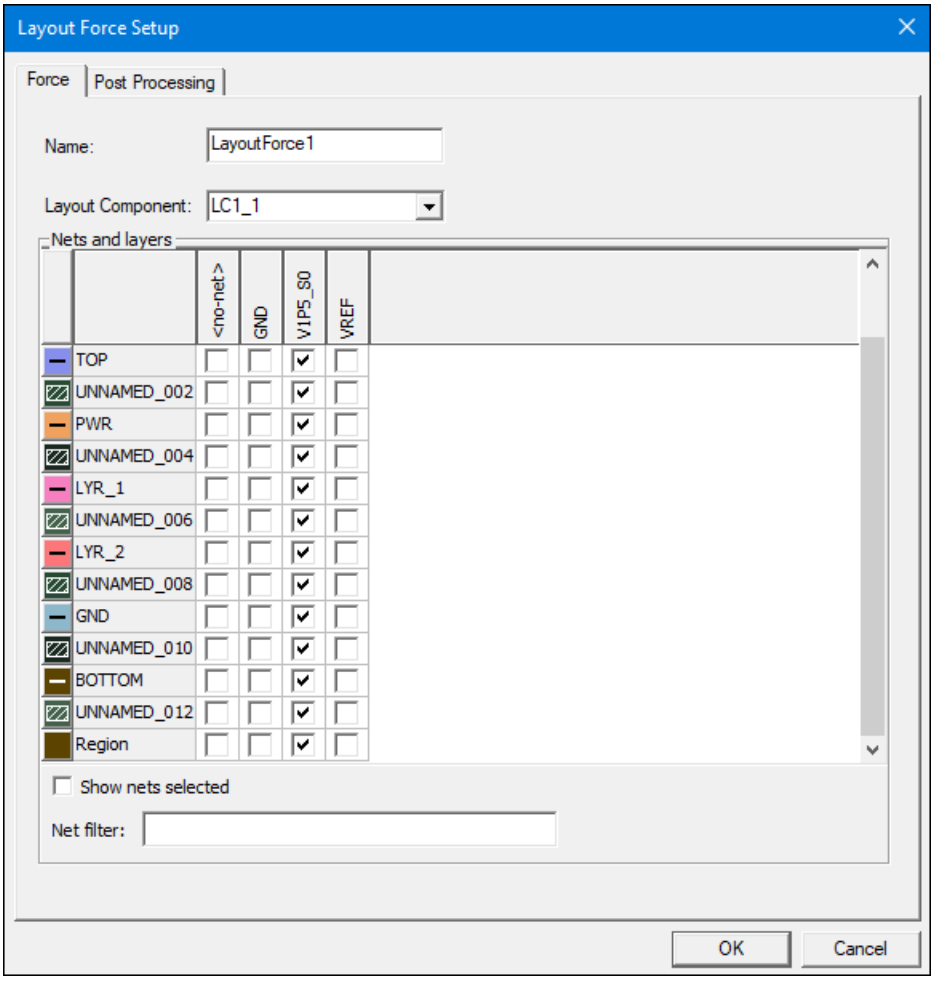


To calculate Lorentz force on objects, right-click **Parameters**, and select **Assign** > **Force from Layout...** Note that this menu item will only be available if there is a layout component.

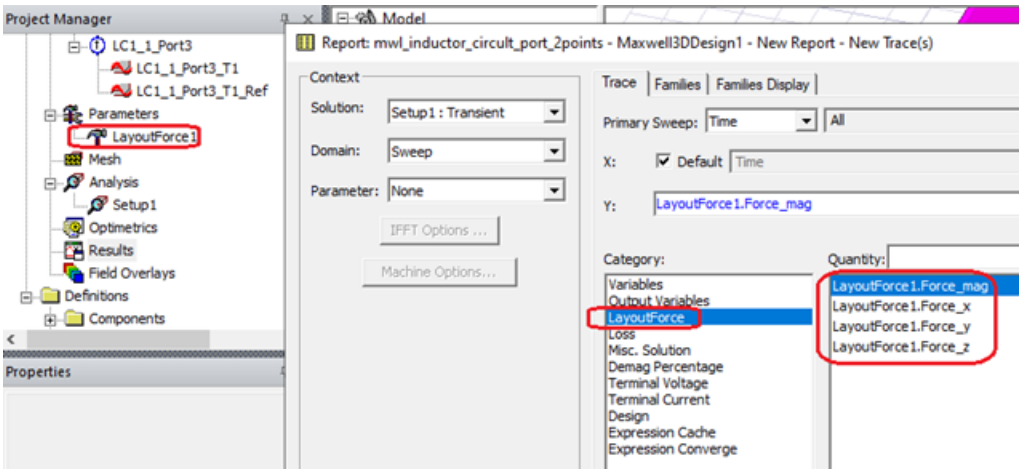


In the **Layout Force Setup** window, on the **Force** tab, select one layout component from the **Layout Component** drop-down menu, and select <layer, net> pairs in the following grid. Each <layer, net> pair represents the object(s) in the intersection of corresponding layer and net. Then select the postprocessing coordinate system on the **Post Processing** tab.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

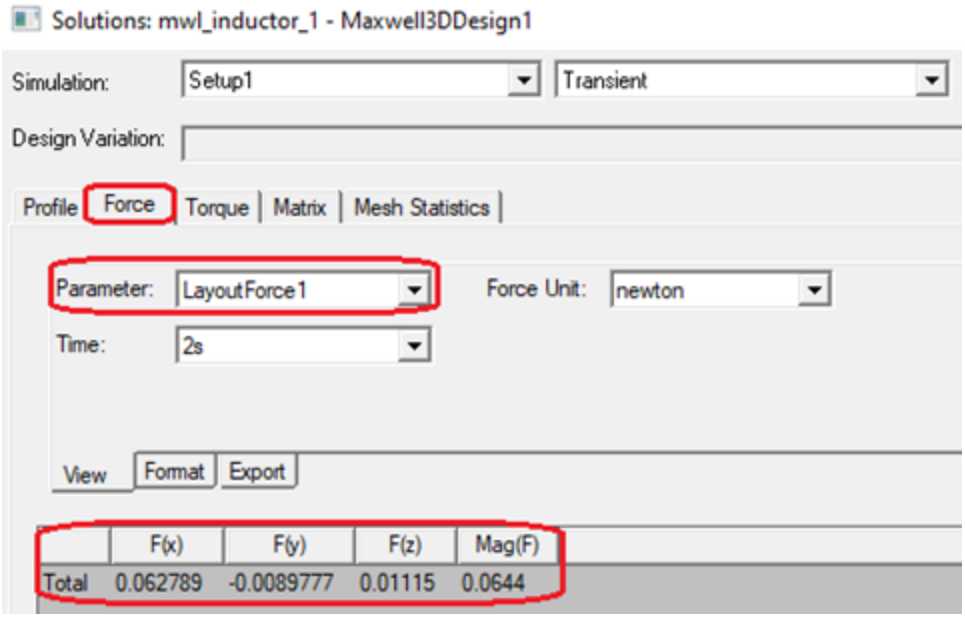


The **LayoutForce** can be plotted from the reporter as shown below:

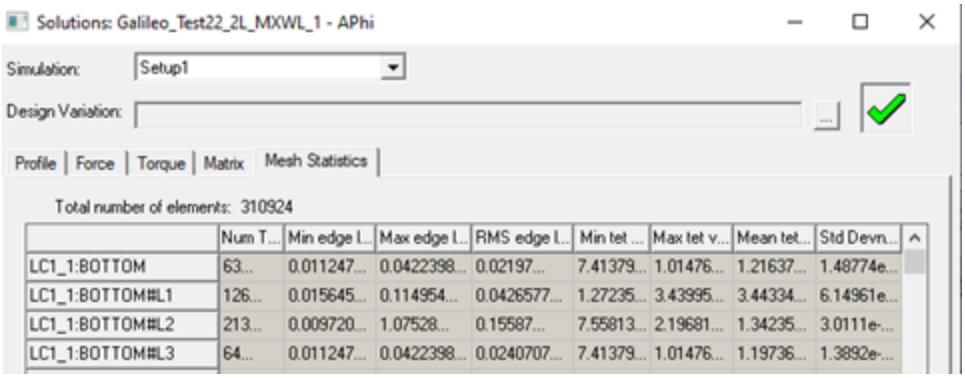


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Layout force can also be viewed in the Solution Data window:

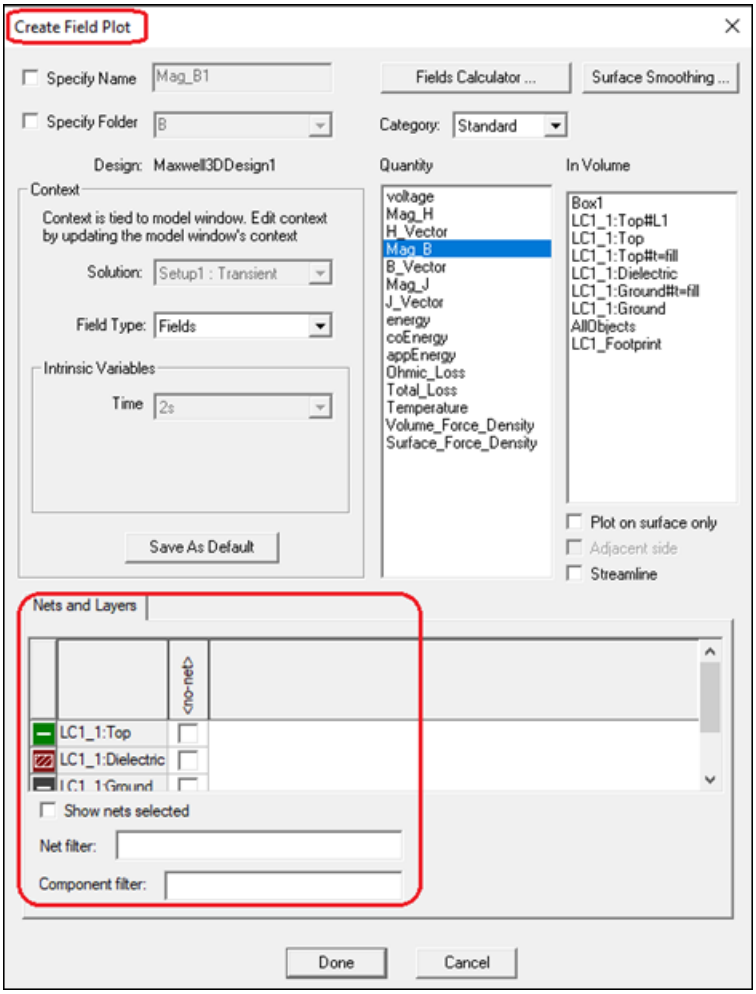


Mesh Statistics for layout objects can also be viewed in the Solutions window:



To review results from the Field reporter, you can select the layout objects in the **Create Field Plot** window:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note: For Maxwell 3D Transient A-Phi designs that include a layout component, you can export field data related to the layout component. Export options include

- Export Grid Max Field Data
- Export Via Average Current Density
- Export Via Current
- Export Layer Induced Eddy Current

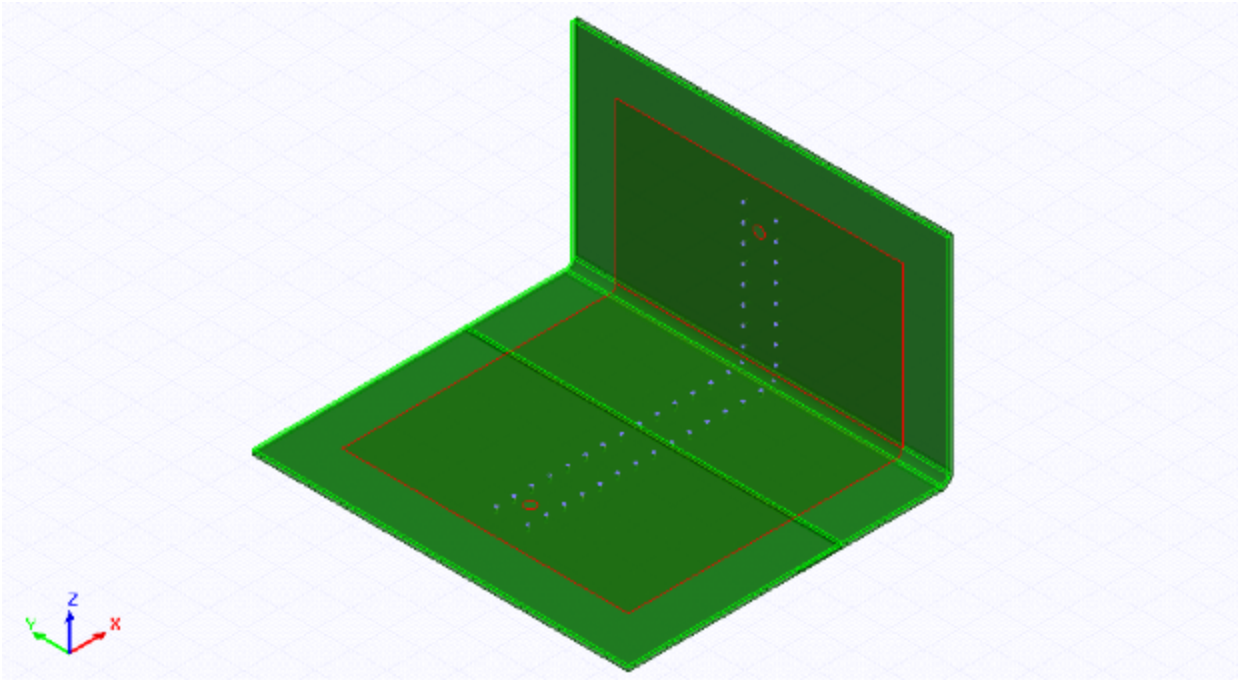
Limitations for the 3D Layout Component Feature in the Maxwell Solver

- Field calculator for expression cache/output variables can only be setup after solving.
- Conduction path functionality in Maxwell is not applicable to layout component.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

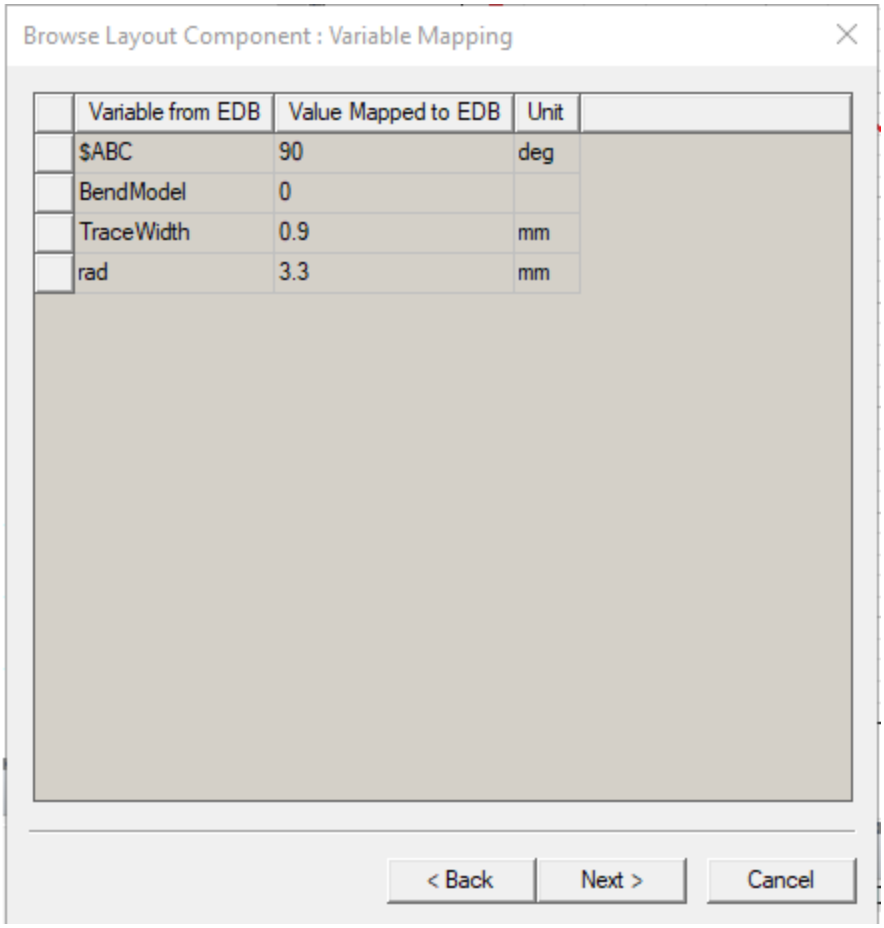
Flex PCB Component Support in Maxwell

Users can now export Flex PCBs designs (i.e., components with rigid-flex bends) from HFSS 3D Layout as components and import them to Maxwell 3D. All regular component functionality is supported for components with rigid-flex bends (e.g., visualization attributes, validation checks, component coordinate systems, circuit elements visualization, net identification, mesh fusion, postprocessing, and so forth).



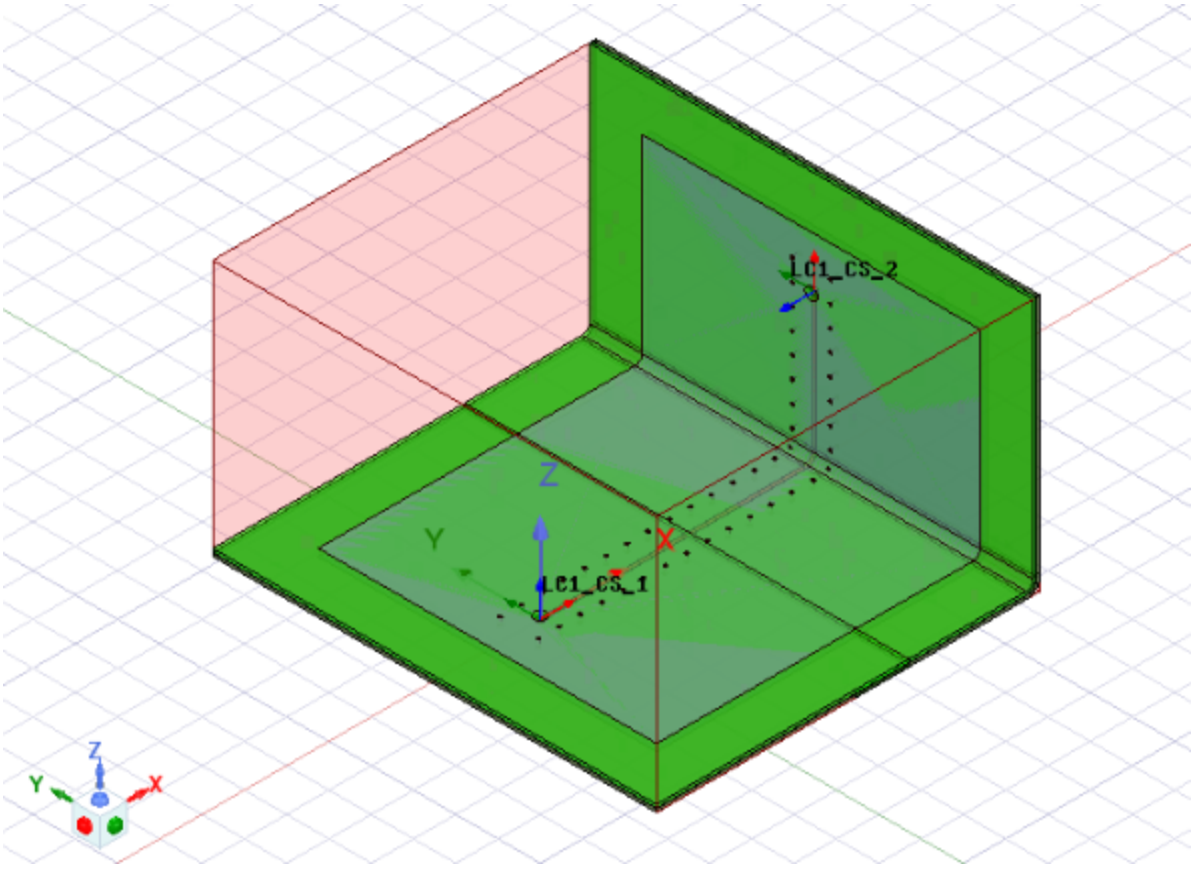
When components with rigid-flex bends are imported, the **BendModel** variable appears in the **Browse Layout Component : Variable Mapping** window, in addition to other variable types shared by other components in the definition design. When the value of **BendModel** is 0, all rigid-flex bends on the components are inactive. Otherwise, the bends on the components are active.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



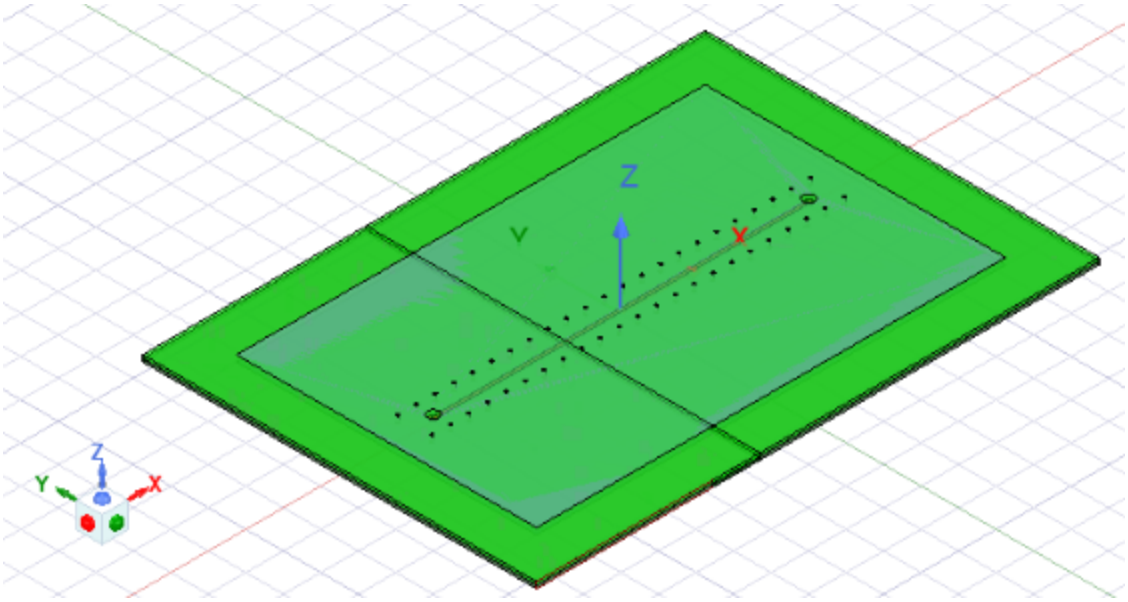
When rigid-flex bends are active, all ports, circuit elements, and component coordinate systems on the bent part of the components are transformed to accommodate the bends.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

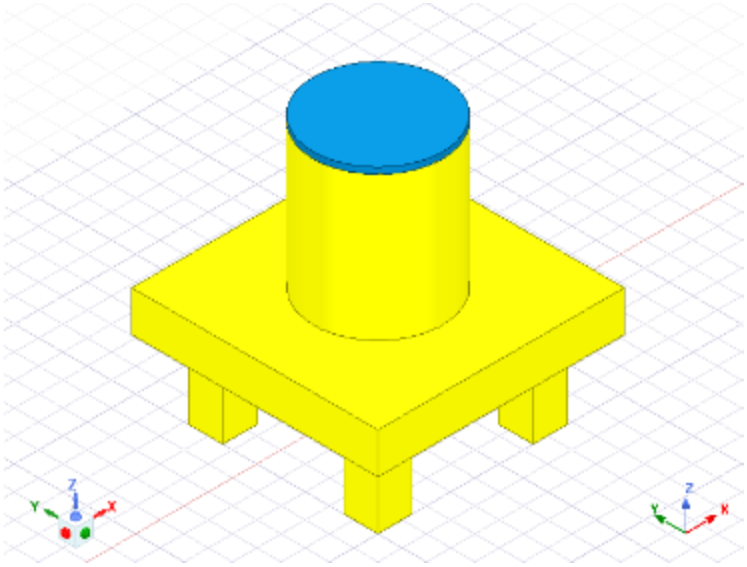


If bends are inactive (i.e., **BendModel** = **0**), all bends in components are inactive, and the components appear in their unbent states.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

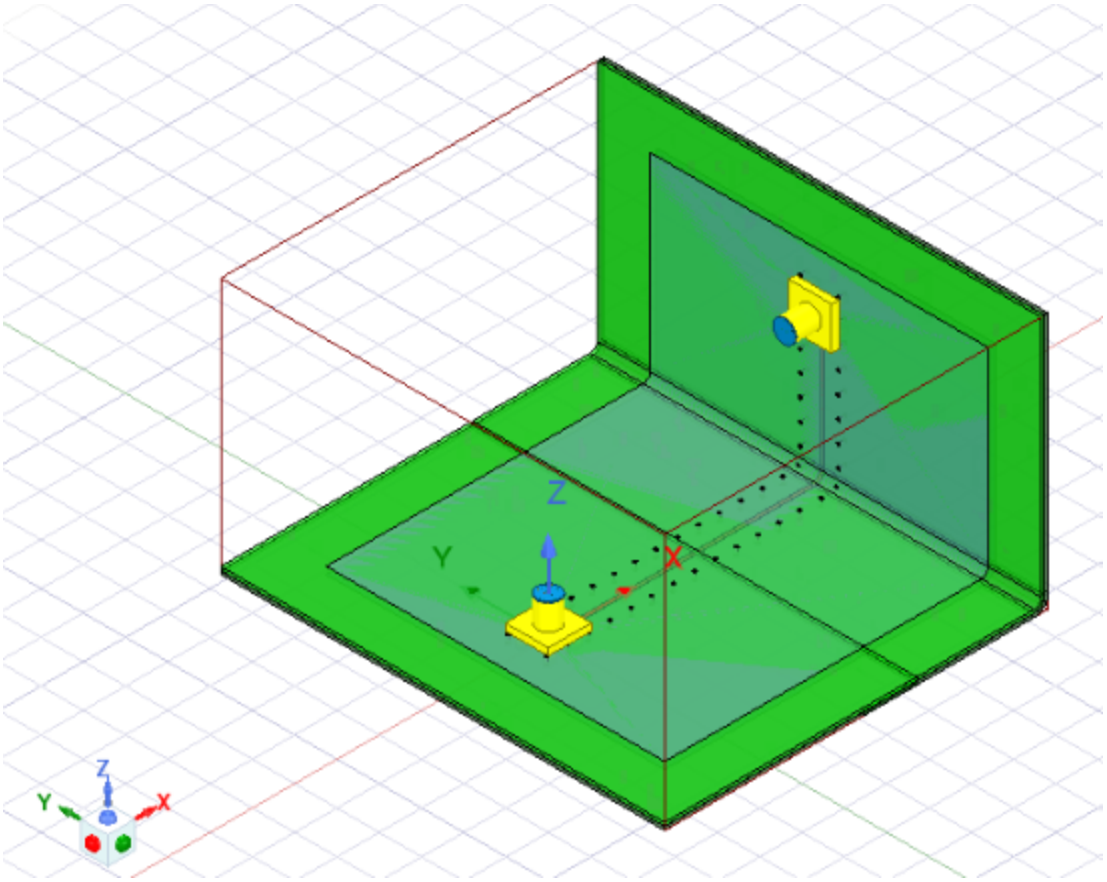


With bends re-enabled, other 3D components can be imported and assembled with bent components.



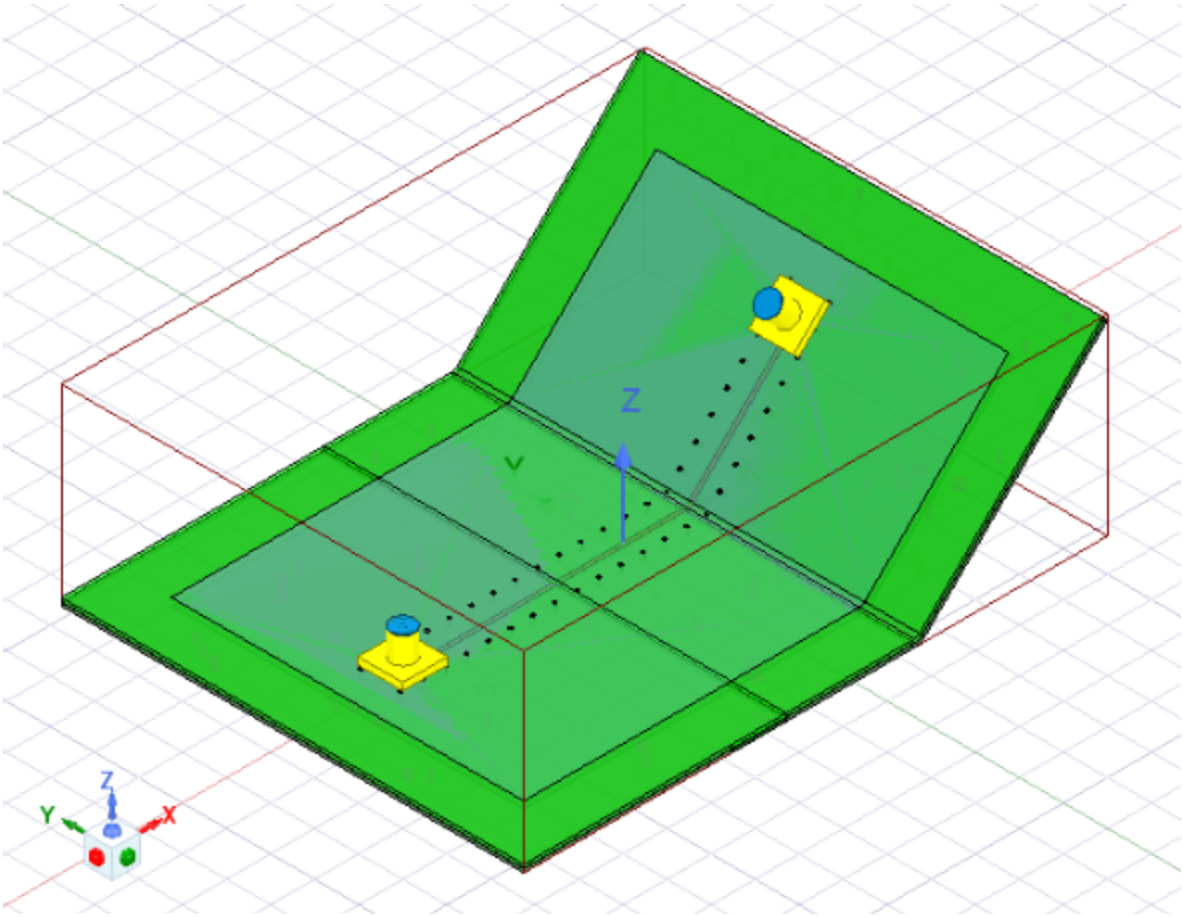
The final assembled system may match the following example (*i.e.*, two 3D components placed on the two coordinate systems of the component).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



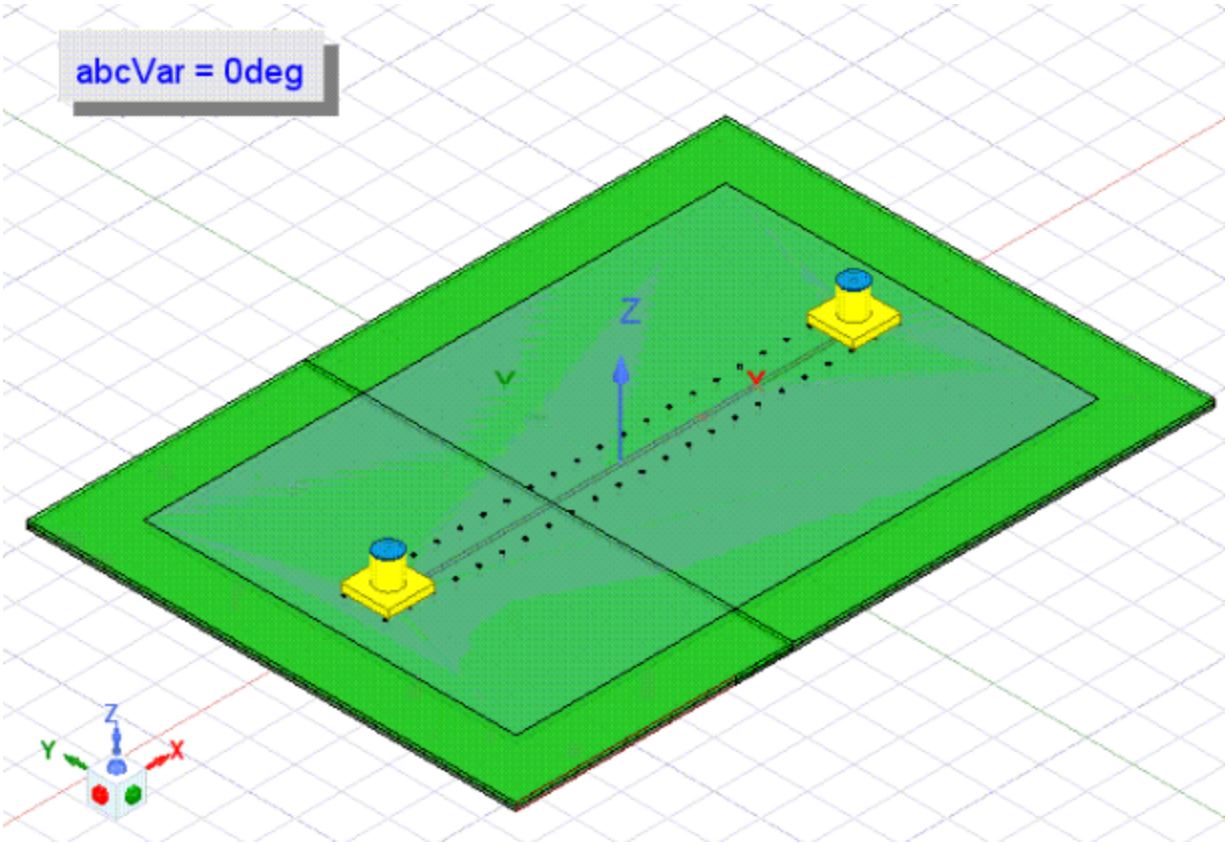
In the following example, the angle of the bend is parametrized with a variable. When components are imported to Maxwell, the parametrized variable can be used to control their bends. If the value is changed, the bend angle of the components update to reflect the change. Other parts of the design that depend on the bend angle are also updated.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



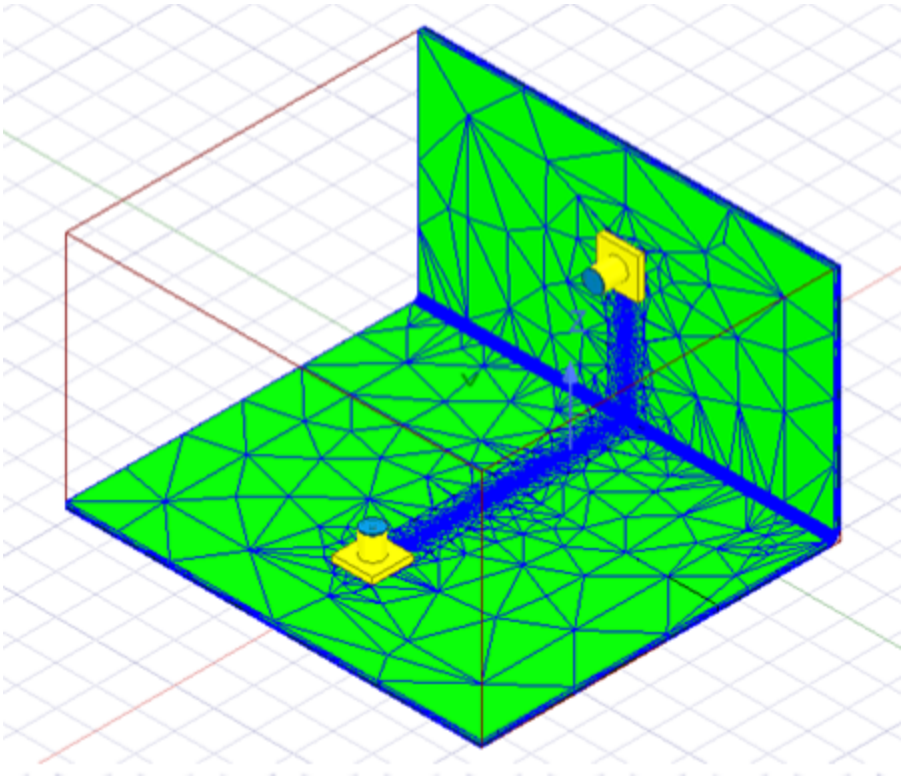
Users can also generate an animation based on the different values of the bend angle.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



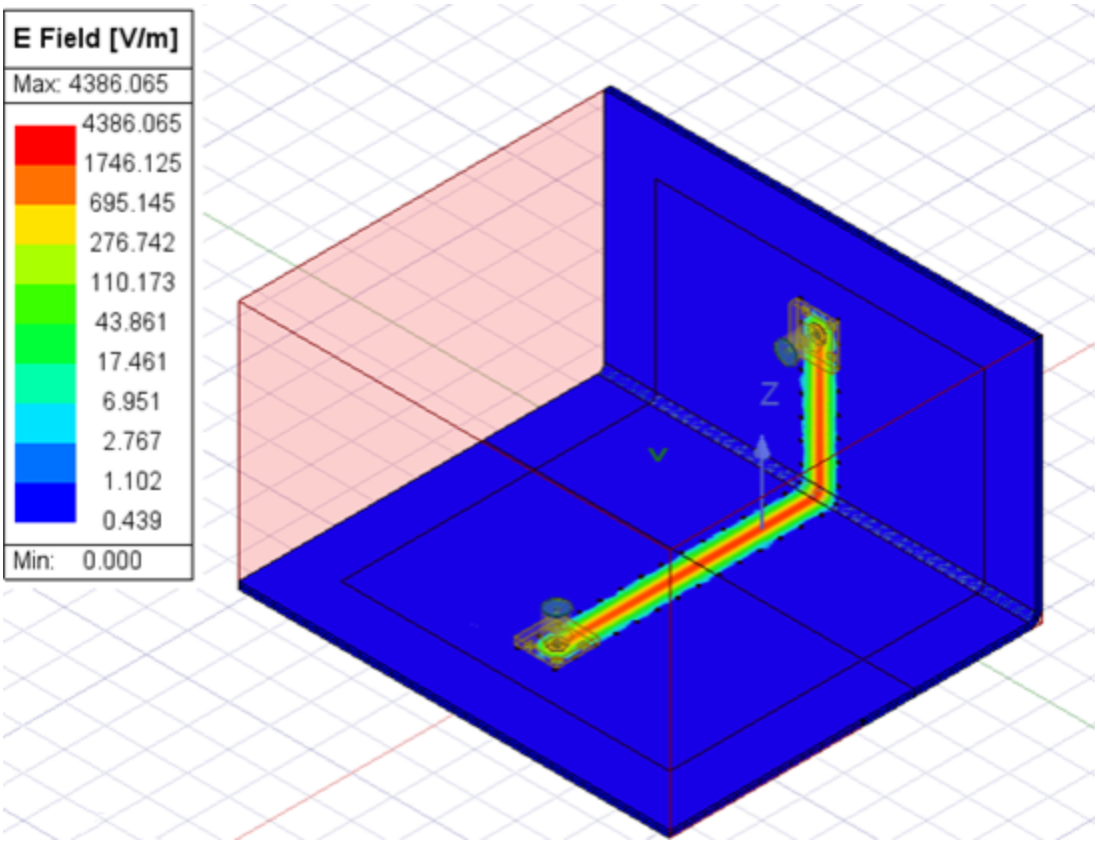
Components with rigid-flex bends can also be meshed and simulated. All the simulation and post-processing functionality for regular components are supported for components with rigid-flex bends and the workflows are identical; plot the mesh-selected nets and layers of the components, as appropriate.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



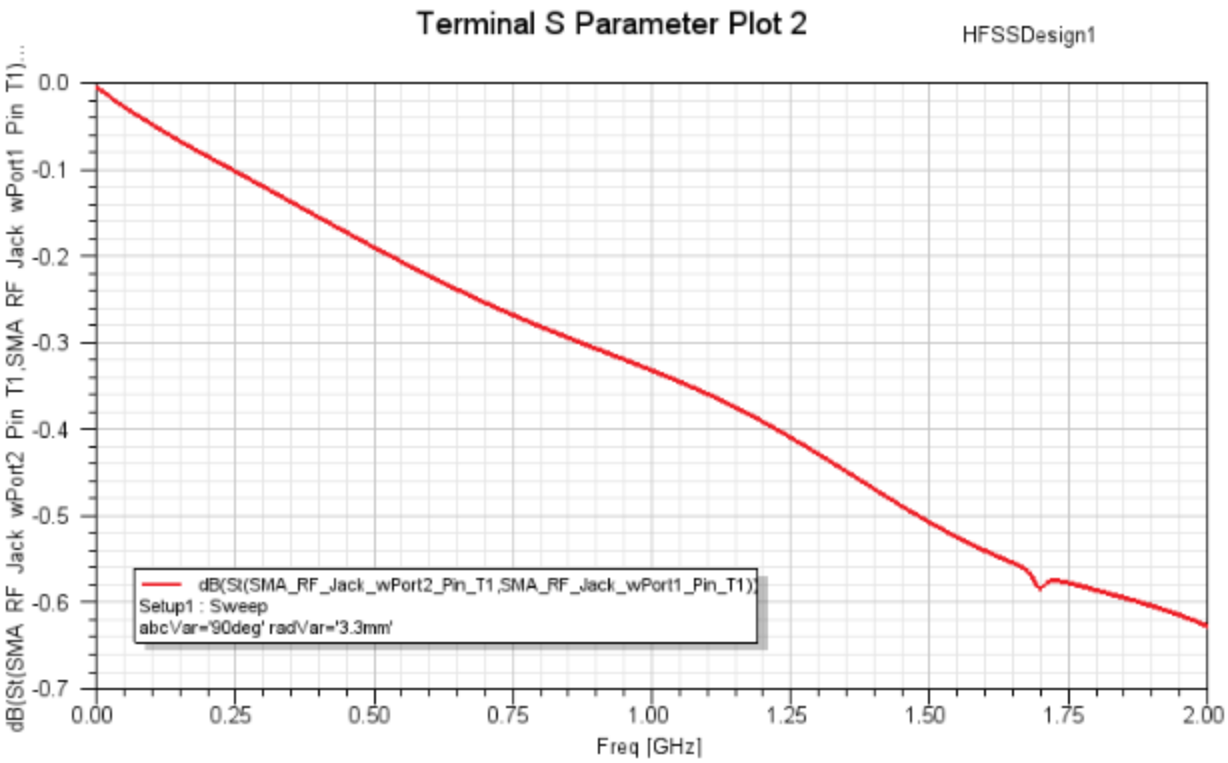
Or plot the fields-selected nets and layers of the of the component, as appropriate.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Finally, create plots (e.g. S Parameter plots).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Field Plots on Layout Components Layer or Net

If a Maxwell design includes Layout Components, you can plot field quantities on layout components' layers and nets. This functionality works seamlessly with the capabilities of plotting fields on 3D geometries. Additionally, the Fields Calculator can calculate field quantities on certain objects inside a layout component that is only drawn in the modeler window without actual geometry.

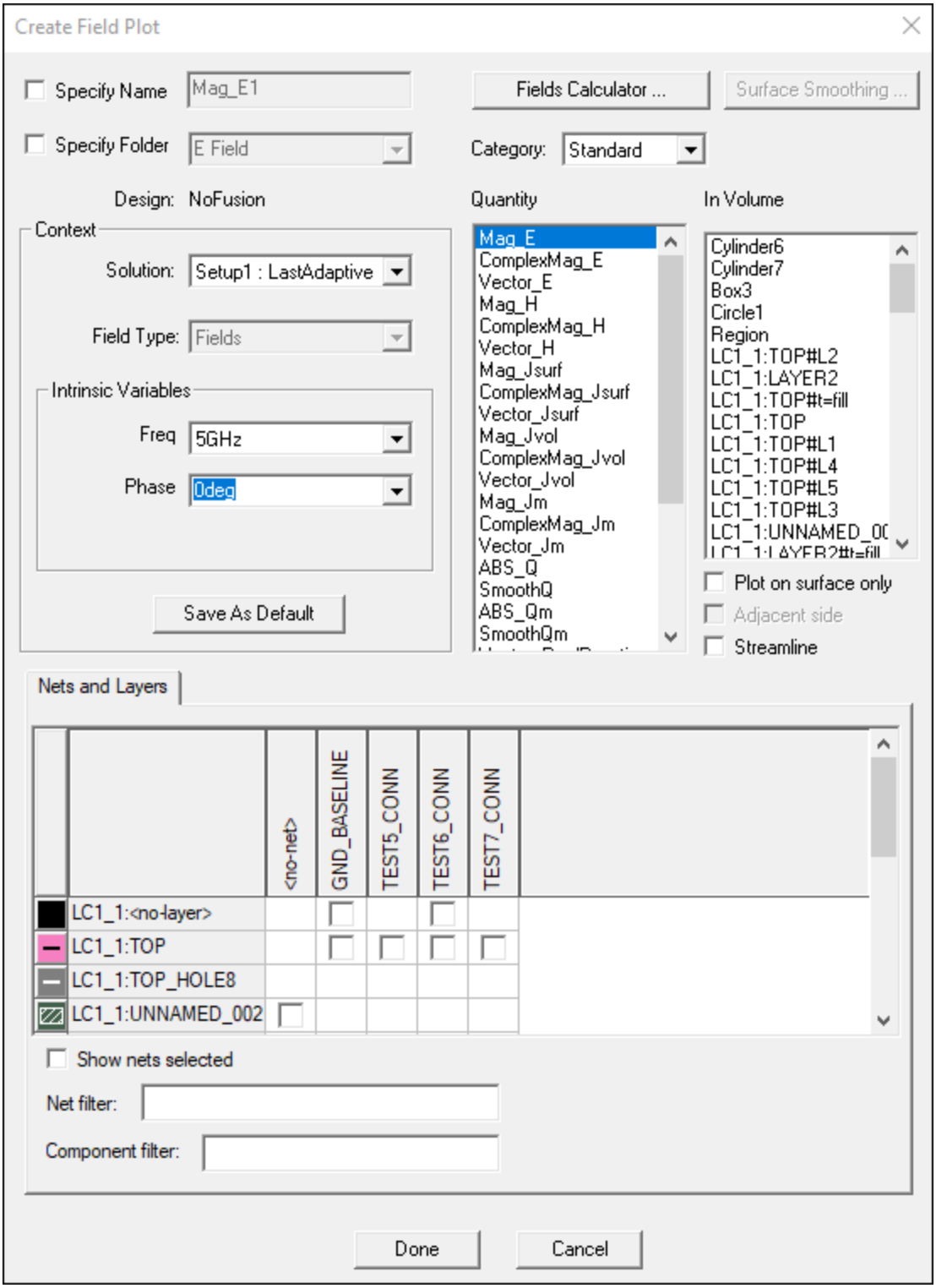
To Plot Fields for a Design Including Layout Components

First, analyze the design. During simulation setup, ensure the Save Fields option is selected. Designs analyzed before release 2023R2 must be analyzed again to extract the necessary information for plotting.

Whereas for designs without layout components, you must select a geometry for the plot before you select the **Plot Fields** command, for designs with layout component, you make selection both before and after selecting **Plots Fields** to open the **Create Field Plot** dialog box.

For designs that include Layout Components, the **Create Field Plot** dialog box includes a Nets and Layers panel that provides additional functionality, showing the layers and nets of every layout component in the design. The **In Volume** field also allows selection of geometries from the layout components.

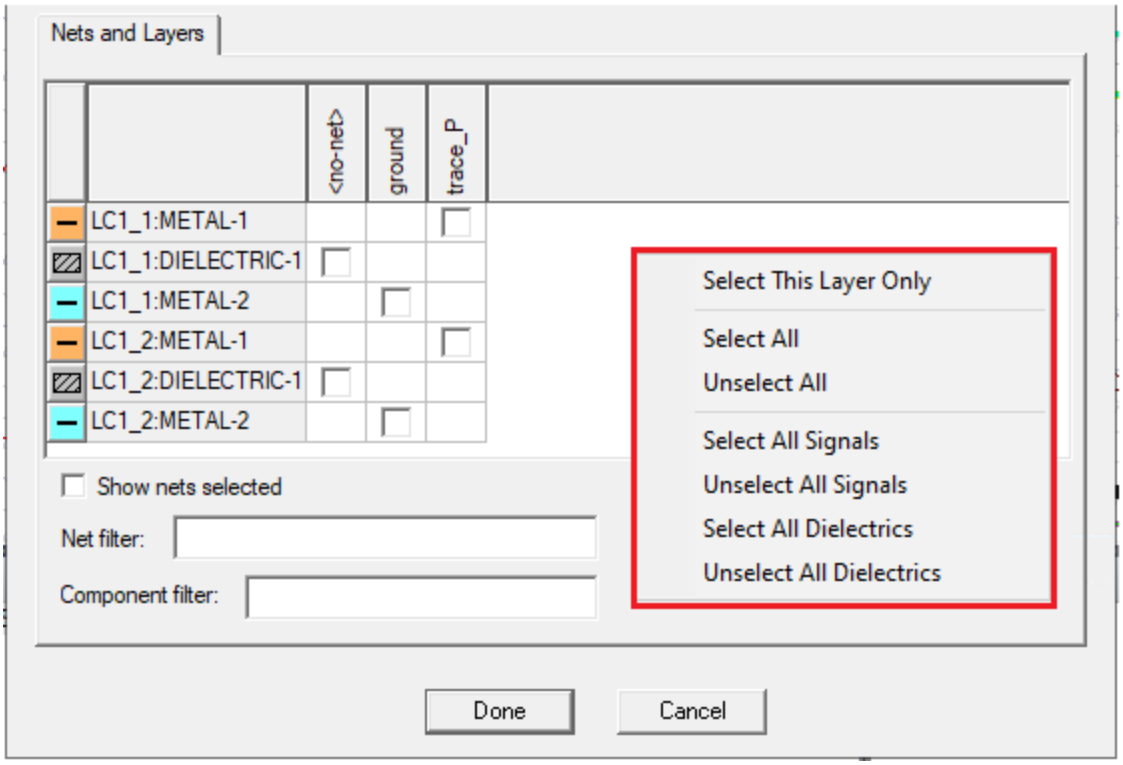
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This **Nets and Layers** section displays all the layers and nets of every layout component in the design. Upon opening, it adjusts its size to try to show all the rows and columns for the layout components. You can resize the parent dialog box. For objects in the component belonging to a specific layer and net, you can select a checkbox. By clicking the column headers, you can select or unselect check boxes for an entire column. You can toggle the “Show nets selected” checkbox to move all the selected nets to the left side of the table. You can use the **Net filter** and **Component filter** fields to filter out specific nets and components. The net and component filters support ‘*’ and ‘?’ wildcards and you can delimit multiple filters with commas. The Nets and Layers table is refreshed each time you update the filter text.

If you right click on the Nets and Layers panel, a shortcut menu appears that lets you **Select This Layer Only**, **Select** or **Unselect All**, **Select** or **Unselect All Signals**, or **Select** and **Unselect All Dielectrics**.

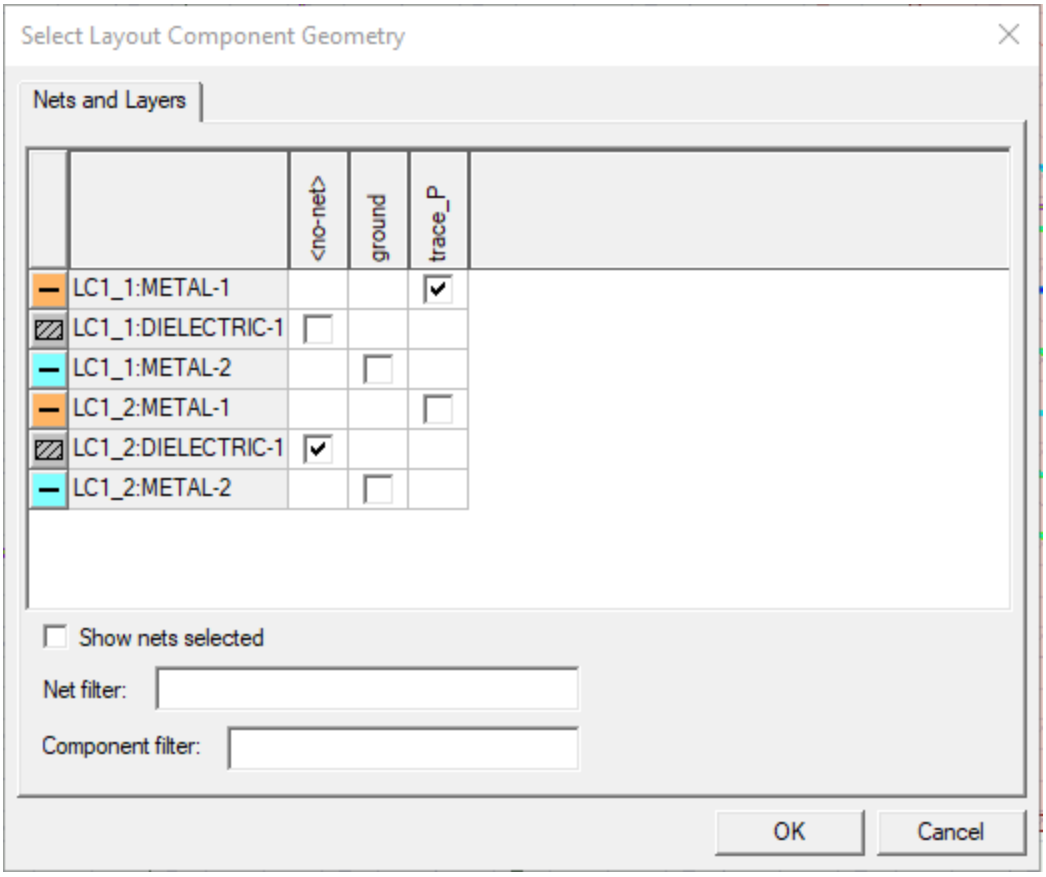


You select Context, Quantity and In Volume information as usual for Plotting Field Overlays.

Once you click **Done** to apply and close the dialog box, Maxwell plots the fields on all objects belonging to the selected layers and nets, in addition to any 3D geometries selected from the modeler window.

If there are layout components in the design, and you want to reassign an existing field plot or mesh plot, a new dialog box will pop up as shown below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This dialog box allows you to reassign the existing plot on layout components' nets and layers. As with the **Nets and Layers** sub dialog for the **Create Field Plot** dialog box, this dialog box shows layers and nets of every layout component in the design, with identical behavior and usage, including the short-cut menu that lets you Select This Layer Only, Select or Unselect All, Select or Unselect All Signals, or Select and Unselect All Dielectrics.

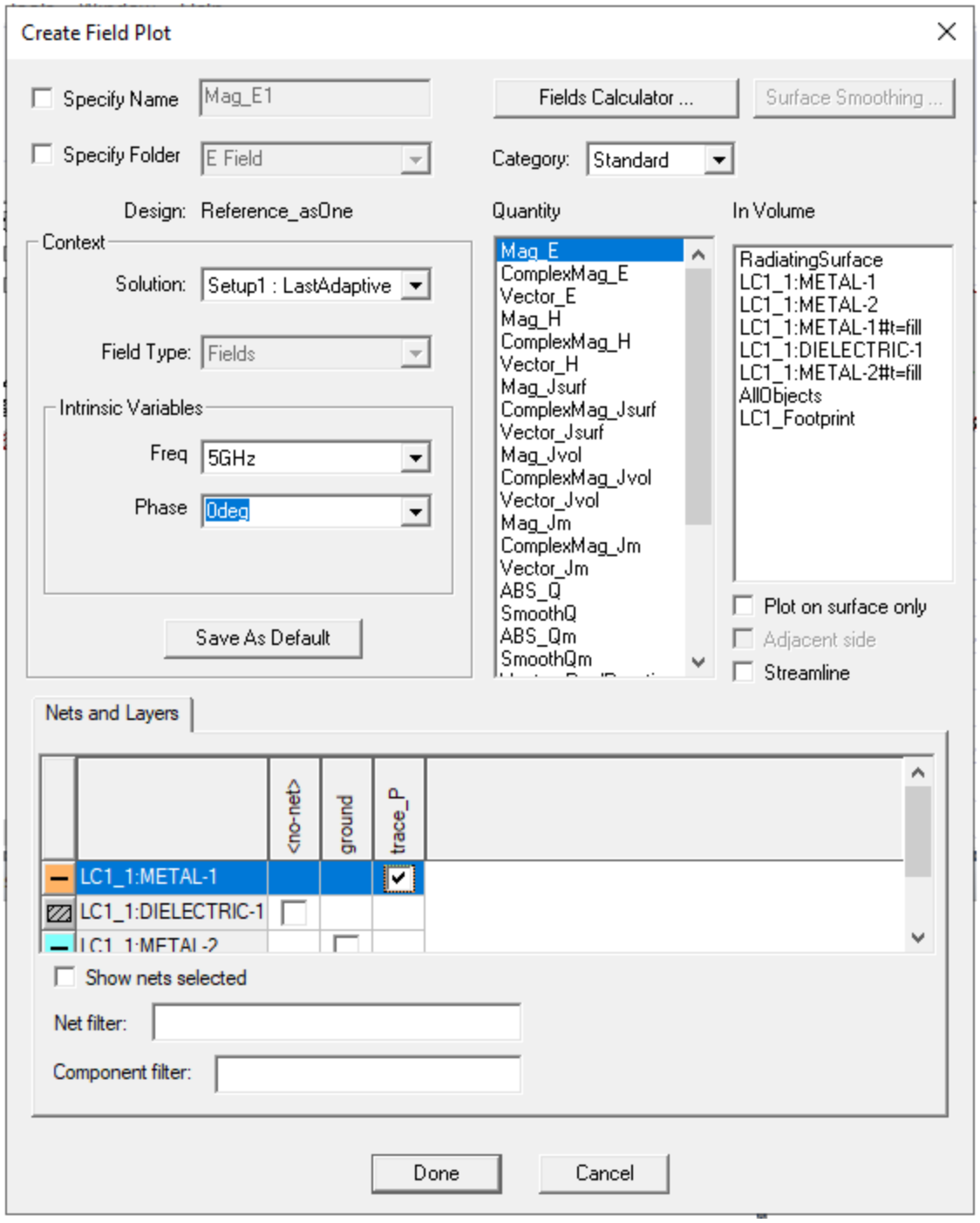
Once you click **OK**, Maxwell reassigns the field/mesh plot on all objects belonging to the selected layers and nets, in addition to any 3D geometries previously selected from the modeler window. If you click **Cancel**, Maxwell will only reassign the field/mesh plot on 3D geometries previously selected. If there are no previously selected 3D geometries, the program issues an error message.

If there are no layout components, this dialog box is not shown, and the workflow for reassigning field/mesh plot is unchanged.

Example with Metal Trace from Layout Component

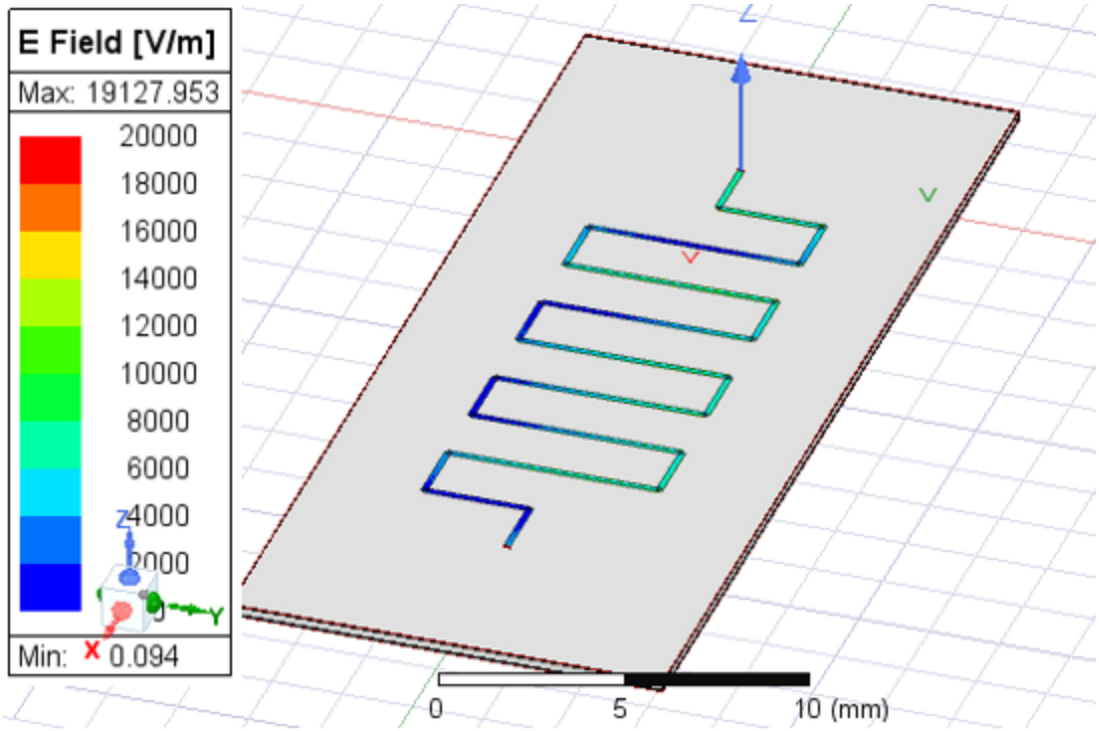
For example, you could select a metal trace from a layout component in the Nets and Layers table as shown below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Model window will show the field plot as follows.

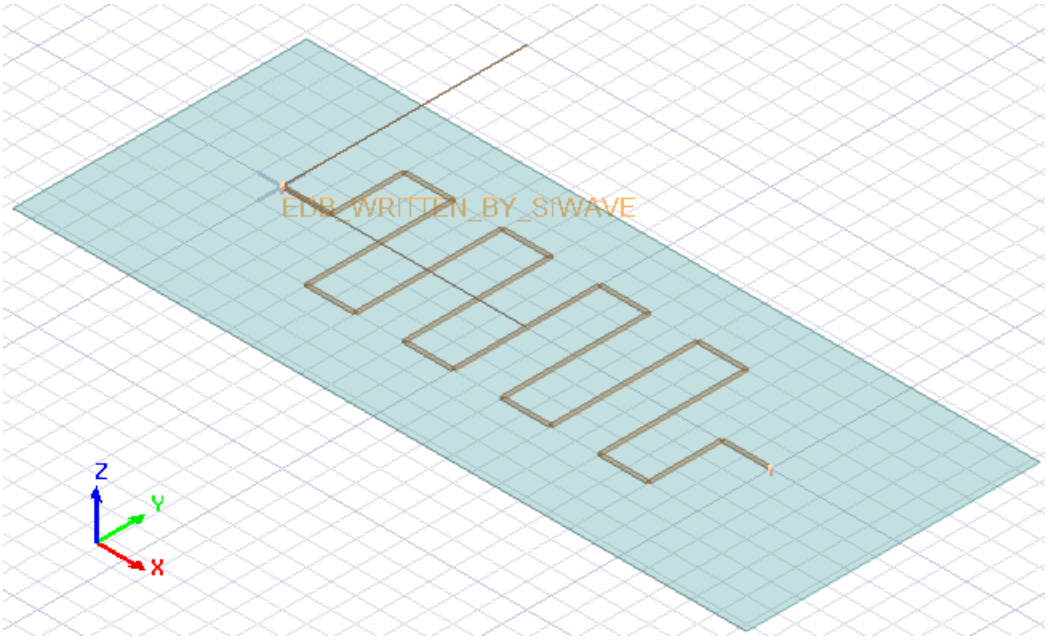
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Using Fields Calculator with Layout Component in a Design

If there are layout components in the design, users can create volume or surface geometry operators in Fields Calculator using any objects from layout components.

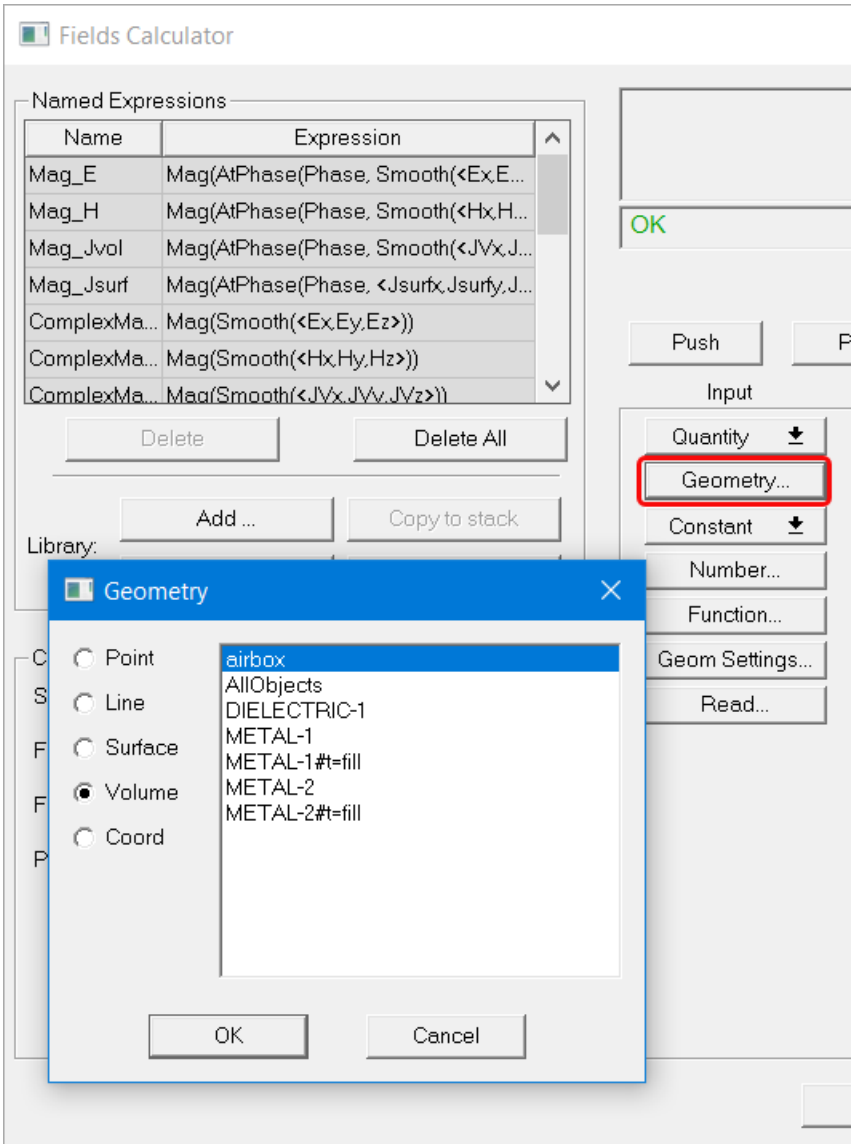
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



First, analyze the design. Designs analyzed before release 2023R2 must be analyzed again to extract the necessary mesh and fields information for the Fields Calculator.

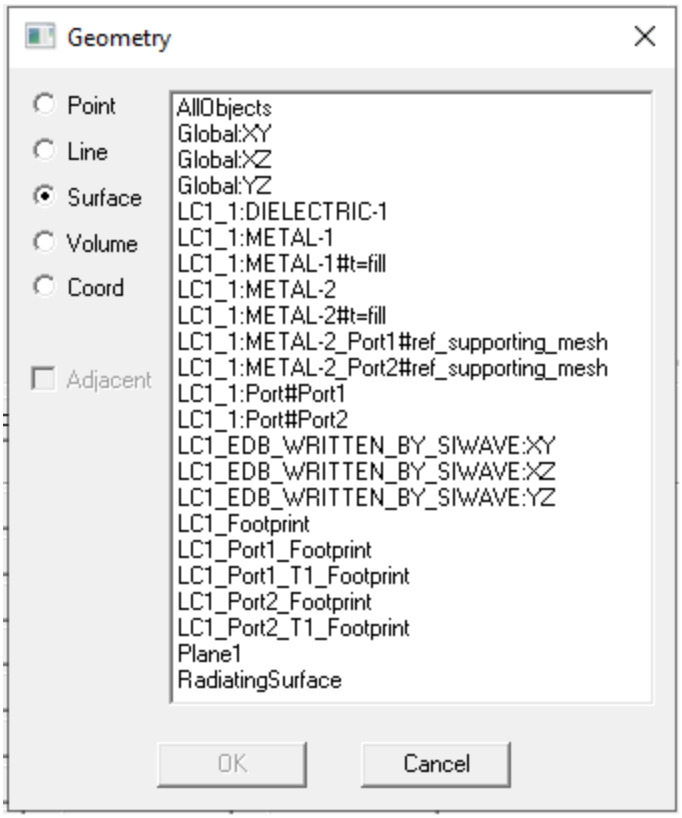
Open the Fields Calculator and create a Geometry operator. The following figure shows the **Geometry** dialog with Volume selected, showing Layout Component objects.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following **Geometry** dialog shows layout components added for the Surface section.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the modeler window, these Layout Component objects are drawn as visualizations without actual geometry but can now be selected and used to calculate field quantities as shown below.

These geometry operators can be used to perform computations just like one constructed from regular 3D geometry. Remember, these Layout Component objects are generated only after running a simulation, so they won't show up if there is no solution.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Related Topics

Using a 3D LayoutComponent

Plotting Field Overlays

Using the Fields Calculator

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

17 - Coupling with the Maxwell Application

Coupling is provided in specific contexts between the Maxwell application and the products listed below.

Ansys Workbench™ Application

Coupling of the Maxwell 2D/3D solvers with the Ansys Workbench application is supported via the Workbench schematic. The following types of coupling with Workbench are supported:

- Thermal feedback is supported for Maxwell magnetostatic, AC magnetic, and transient solution types.
- Stress feedback is supported for Maxwell magnetostatic, AC magnetic, and electrostatic solution types.
- Harmonic force is supported for Maxwell transient and AC magnetic solutions.
- Partial Model Coupling

For more information, see Maxwell to Ansys Thermal Coupling and Maxwell to Ansys Stress Feedback Coupling, and Coupling Maxwell with Both Ansys Thermal and Structural via Workbench.

> **Note:** To use couple Maxwell with Ansys Workbench, you must integrate the Maxwell application with Ansys: from the Start menu, select **Ansys EM Suite 2025 R2** > **Modify Integration with Ansys 2025 R2**.

Ansys HFSS™ Application

There are two possible coupling mechanisms between the HFSS and Maxwell applications:

- Near field coupling
- Magnetic bias source coupling

While they serve different application objectives, they are similar in neither requires the geometry in the coupled designs to be identical.

For more information, see Near Field Coupling between Maxwell and HFSS and Magnetic Bias Source Coupling between Maxwell and HFSS.

Ansys Twin Builder® Application

Coupling between the Maxwell and Twin Builder applications can take the following forms (initiated through the Twin Builder user interface):

- Export a parametric matrix from Maxwell to an ECE model file for placement in Twin Builder.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Dynamically link a Maxwell capacitance or inductance project to a component in Twin Builder.
- Dynamically link a Maxwell Transient design to Twin Builder to perform Transient-Transient cosimulation.
- Dynamically link a Maxwell design to Twin Builder to perform State Space cosimulation.
- Dynamically link to RMxprt through Dynamic SML.

Because the thermal and/or stress solutions are derived from the distributed power loss and force density, they are likely to be more accurate than analyses based on average losses/force.

For more information, see "Coupling Maxwell to a Twin Builder Component."

Ansys Fluent® Application

Coupling without using the Ansys Workbench application is supported between the Ansys Fluent application and Maxwell 3D magnetostatic and Eddy current designs in the Maxwell application. A transient Ansys Fluent simulation can use a Maxwell solution to simulate the magnetic effect in the electric arc of contactors.

For more information, see Coupling Maxwell Magnetostatic and AC Magnetic Designs with Ansys Fluent.

LMS Virtual.Lab™

The Maxwell application supports one-way coupling with Siemens® LMS Virtual.Lab™.

Maxwell to Ansys Thermal Coupling

Coupling Maxwell 2D/3D with Ansys applications is supported via the Workbench schematic. Thermal feedback is supported for Maxwell magnetostatic, AC magnetic, AC and DC conduction, and transient types. Users also need to set up the design and geometry appropriately. An appropriate design should be temperature-dependent, and have one or more solve setups that are enabled for thermal feedback.

> **Note:** A Maxwell system can also be coupled with both thermal and stress systems via Workbench.

Maxwell-Specific Considerations

- If there is thermal feedback, each frequency point in an AC magnetic sweep will be solved with the same feed-back data.
- Temperature distribution data that has already been exported from Ansys Thermal will be invalidated when users continue the Maxwell solution in a way such that the Maxwell mesh is being regenerated.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Once the Maxwell solution has been integrated with a thermal effect, the adaptive process of a static simulation will be disabled. Users must then manually invalidate the temperature data in the existing solution before the adaptive process will be re-enabled.
- Maxwell solutions can incorporate feedback from both thermal and stress.
- When setting up loss transfers from Maxwell to WorkBench, if the user wants to map loss density on a face shared by two objects, this face only has to be used once in the setup.

Maxwell 2D Coupling Considerations

For Maxwell 2D in RZ mode, geometry either must be exported into the XY plane, or the geometry must be rotated into the XY plane after it is imported into **Ansys Mechanical**. This is because Mechanical 2D requires geometry to be in XY plane.

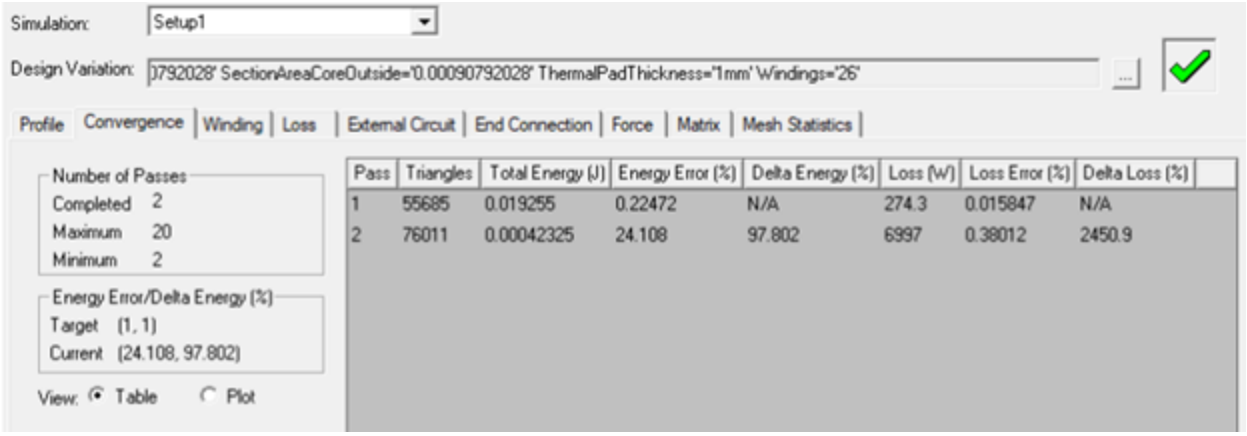
There are two modes in Ansys Thermal 2D. The default mode is called **Planed Stress** where zero stress and non-zero strain are assumed in the z-direction. This is similar to the XY mode in Maxwell 2D. The other mode is called **Axisymmetric**. This mode will be used when a 3D model can be generated by revolving a 2D section 360 degree. This is similar to the RZ mode in Maxwell 2D. Users must ensure that they use the correct geometry mode in Thermal 2D. Invalid geometry mode selection is prohibited by the Thermal2D interface.

There are two types of **Import Load** in Mechanical: **Heat Generation** and **Heat Flux**. **Heat Generation** is used when mapping losses from *objects* in Maxwell 2D/3D and **Heat Flux** should be used to map the loss from the *edges* of objects in Maxwell 2D/3D. **Heat Flux** is supported only for AC magnetic designs with impedance boundary. Users can insert multiple **Heat Generation** or **Heat Flux** loads via the **Imported Load (Maxwell 2D/3DSolution)** objects.

Convergence of Maxwell-Mechanical Two-Way Thermal Coupling

In two-way thermal coupling between the Maxwell static solver (magnetostatic, eddy, AC, DC) and the Mechanical thermal solver, the user may be confused by the **Loss** value reported in the solution profile (see figure below): when two-way coupling is finished, the difference of the Loss value between the last pass and previous pass is sometimes huge and is NOT aligned with **Loss Error(%)**. The uncertainty arises as to whether the two-way coupling process has truly converged.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The convergence reported in the solution profile primarily reflects the convergence history in mesh refinement, aiming for a field solution independent of mesh size. This mesh refinement process occurs within the AEDT environment.

Instead of relying on the convergence reported in the Maxwell solution profile, users should focus on examining properties of the **Feedback Iterator** in the Workbench:

Properties of Schematic D2: Feedback Iterator

	A	B
1	Property	Value
2	General	
3	Component ID	FeedbackIterator
4	Directory Name	FeedbackIteratorComponent
5	Update Condition Parameter (Beta)	None
6	Notes	
7	Notes	
8	Used Licenses	
9	Last Update Used Licenses	Ansys Mechanical Enterprise [12]
10	Iterations	
11	Iterations Completed	4
12	Max Iterations	15
13	Callback	
14	Script	
15	Temperature Convergence	
16	Target Delta Temperature %	1
17	Latest Delta Temperature %	0.00194419
18	Displacement Convergence	
19	Target Delta Displacement %	5
20	Latest Delta Displacement %	Not Available

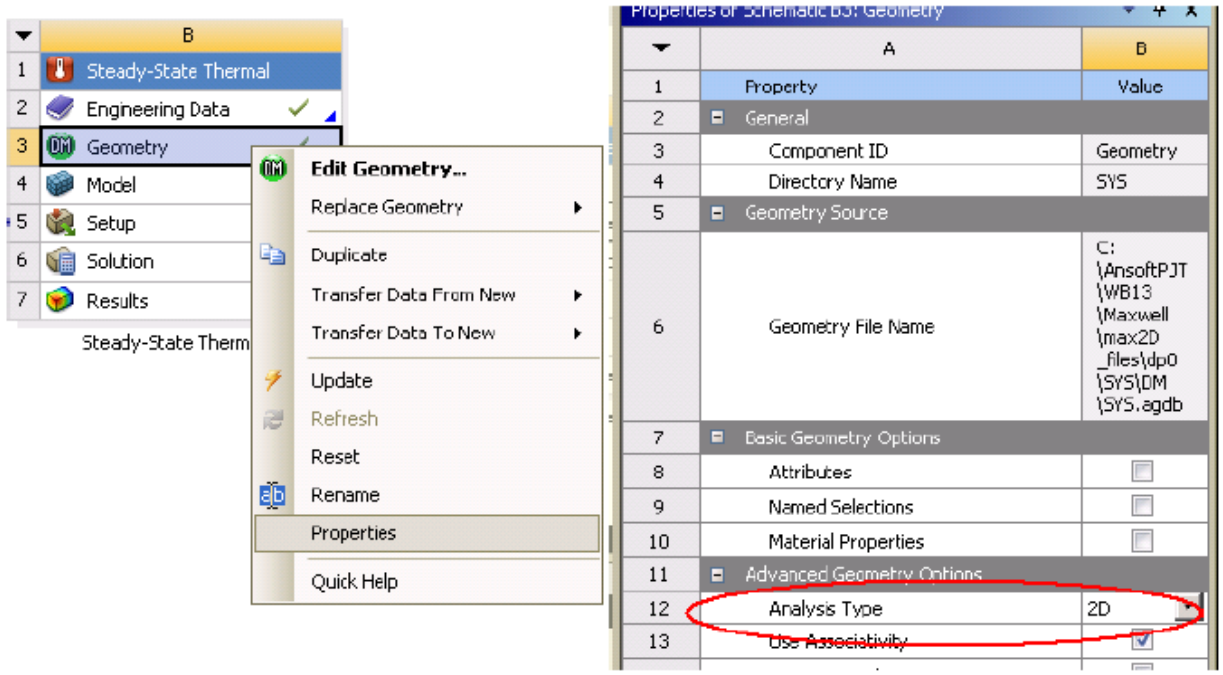
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The true convergence of the two-way thermal coupling process implies the attainment of a steady temperature field and steady loss distribution in Maxwell. The mesh refinement process serves as a preprocessing step for the Maxwell static solvers. In two-way thermal coupling feedback iterations, the mesh in Maxwell solver remains unchanged, but the Maxwell field solution is changing because the material properties are temperature dependent, which affect the solution profile of the last pass.

Coupling Maxwell Designs with Ansys Thermal via Workbench

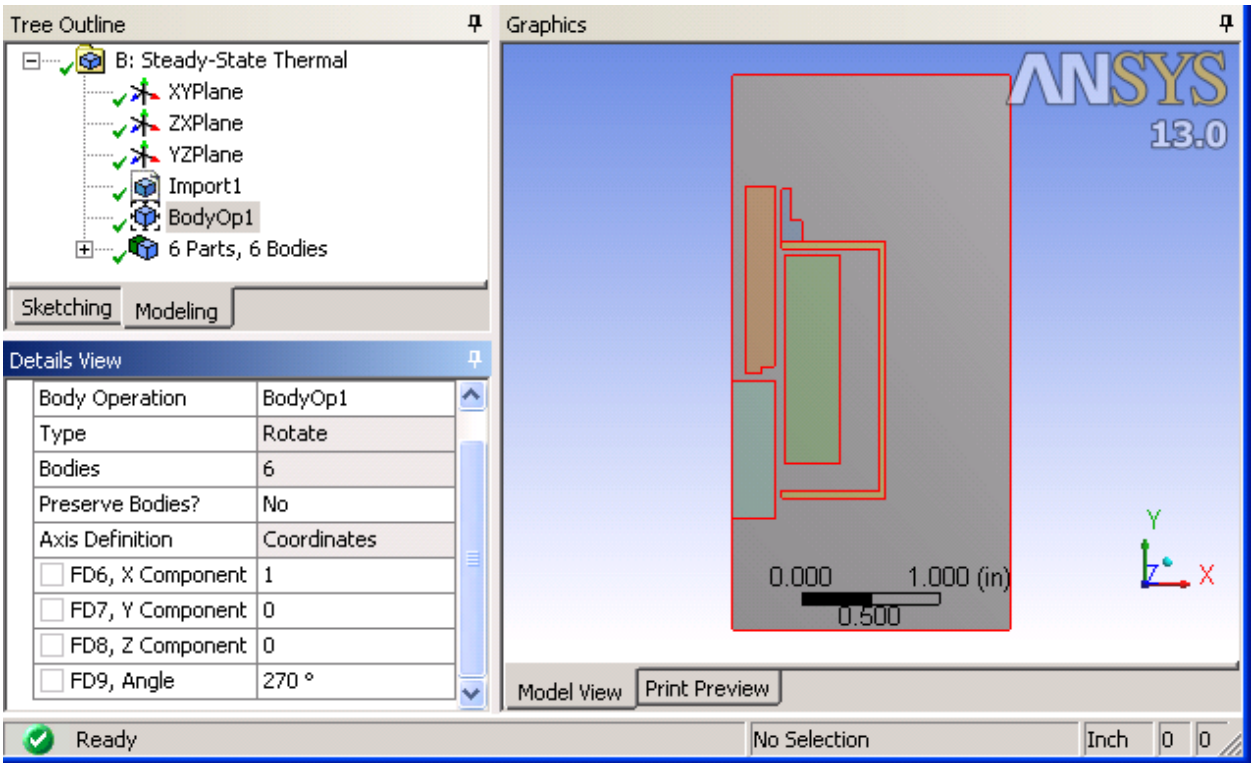
Coupling Maxwell 2D/3D with Ansys applications is supported via the Workbench schematic. Thermal feedback is supported for Maxwell magnetostatic, AC magnetic, AC and DC conduction, and transient types. Users also need to set up the design and geometry appropriately. An appropriate design should be temperature-dependent, and have one or more solve setups that are enabled for thermal feedback.

1. The easiest way to add a Maxwell 2D or 3D design to a Workbench schematic is to import a working design via **Workbench File** > **Import**. The imported design is placed in the **Workbench** schematic after it is successfully imported.
2. Next, insert a **Steady-State Thermal** system and change its **Analysis Type** to **2D** or **3D**, (depending on the Maxwell design type) by right-clicking on the **Geometry** cell and selecting **Properties**. It is important to change the **Steady-State Thermal** system's analysis type *before* setting up its geometry.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. To set up the **Steady-State Thermal** system's geometry, you must first export the Maxwell geometry using **sat** or **step** format as follows:
    a. Select the **Modeler** > **Export** menu item.
    b. Select the desired model geometry format (**sat** or **step**), and the save location in the dialog box and save the file for use by Ansys Workbench.
4. Import the file via the **Geometry** module of the **Steady-State Thermal** system.
    a. To access the **Geometry** module, double-click on the **Geometry** cell in the **Steady-State Thermal** system to launch **Design Modeler**.
    b. Select **File** > **Import External Geometry File** and browse for the geometry file exported from Maxwell.
    c. After the geometry file is imported, right-click on the root folder of the modeler project tree and select **Generate**. (When the geometry file is of a Maxwell 2D RZ design, users can rotate the geometry in **Design Modeler** by creating a body operation.)

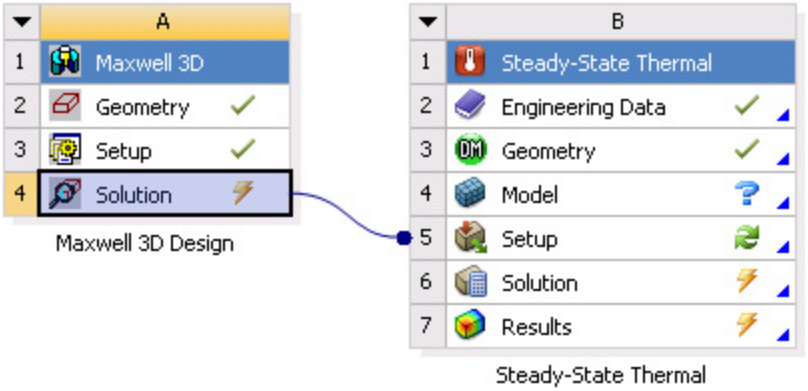


5. Close **Design Modeler** and refresh the **Model** cell of the **Steady-State Thermal** system by right-clicking the **Model** cell and selecting **Refresh**.
6. The geometry mode of the **Steady-State Thermal** system can be changed via the Ansys **Mechanical** user interface.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

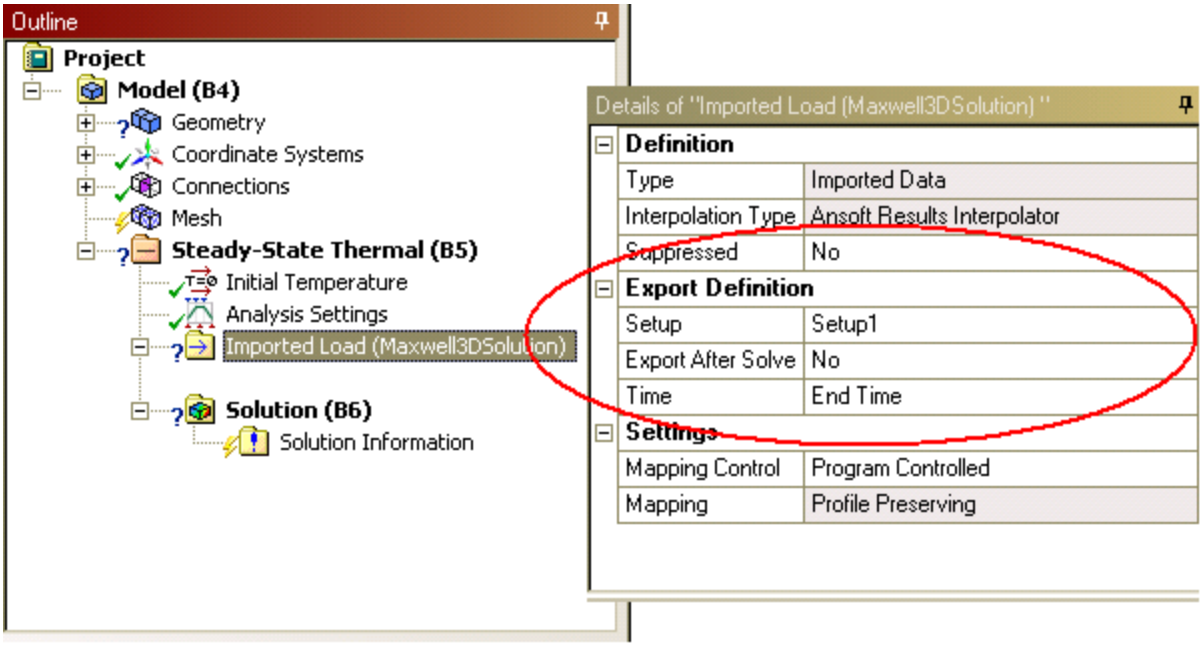
a. Launch **Mechanical** by double-clicking the **Setup** cell of the **Steady-State Thermal** system.
b. Select **Geometry** in the project tree and the **Definition** of **Geometry** will be shown in the **Detail** window.
c. Select either **Plane Stress** or **Axisymmetric** as the value for the property 2D (or 3D) **Behavior**.

7. To set up the coupling, drag the **Solution** cell of the Maxwell system and drop it on the **Setup** cell of the **Steady-State Thermal** system.



8. Note that the Maxwell **Solution** cell is tagged with a “Lighting Bolt” symbol. Right-click on the Maxwell **Solution** cell and select **Update**. This will initiate a Maxwell simulation if it is not already solved. Once Maxwell's solution is available, the “Lighting Bolt” changes to a “Green Check” symbol.
9. To “push” the coupling into **Steady-State Thermal**, right-click on the **Steady-State Thermal Setup** cell and select **Refresh**.
10. After refresh is finished, you can launch Ansys **Mechanical** by double-clicking the **Setup** cell to finish the coupling setup.
11. In the Ansys **Mechanical** application project tree, an **Imported Load (Maxwell2DSolution)** or **(Maxwell3DSolution)** item should already be inserted. Select the **Imported Load** folder to view its details. Because the inserted Maxwell 2D (or 3D) system supports thermal feedback, the **Details** window shows information regarding how the temperature result should be exported, and what type of mesh mapping should be used.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

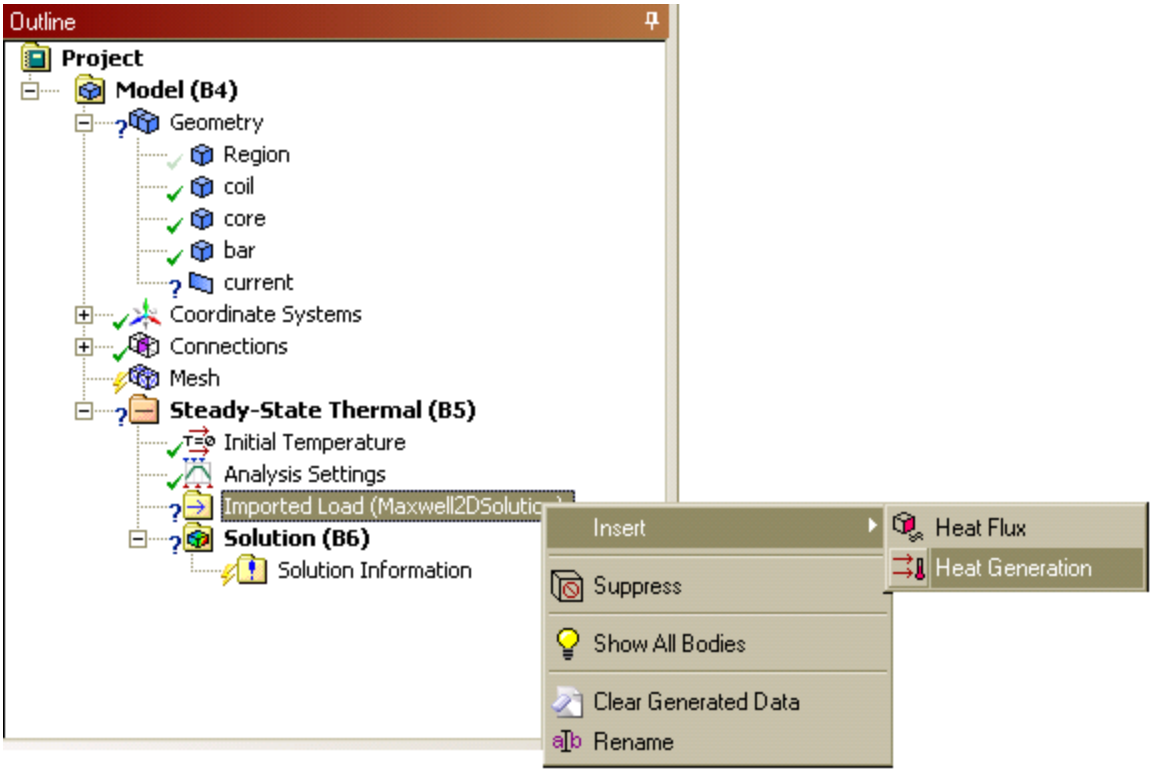


12. To finish the coupling setup, you must insert either an **imported Heat Generation** or an **imported Heat Flux** boundary condition. **Heat Generation** is used when mapping loss from objects in Maxwell; and **Heat Flux** should be used to map the loss from the edges of objects in Maxwell. Users can insert multiple **Heat Generation** or **Heat Flux** loads via the **Imported Load (Maxwell2DSolution)**, or **Imported Load(Maxwell3DSolution)** objects.

    > **Note:** When setting up loss transfers from Maxwell to any other product within the WorkBench framework, if the user wants to map loss density on a face shared by two objects, this face only has to be used once in the setup.

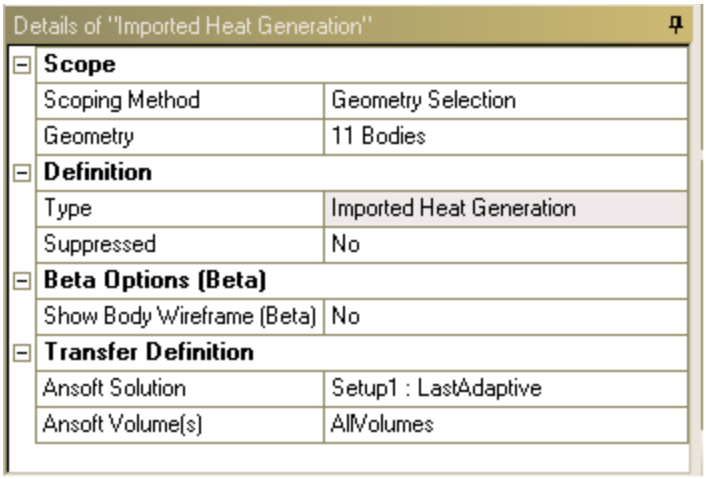
13. To insert a **Heat Generation**, use **Body** select by clicking the icon in the **Mechanical Toolbar**. Then click on the objects where the EM loss should be imported. After all the desired objects are selected, right-click on **Imported Load (Maxwell2DSolution)** > **Insert** > **Heat Generation**, or **Imported Load (Maxwell3DSolution)** > **Insert** > **Heat Generation**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A sub-item named **Imported Heat Generation** will appear in the project tree.

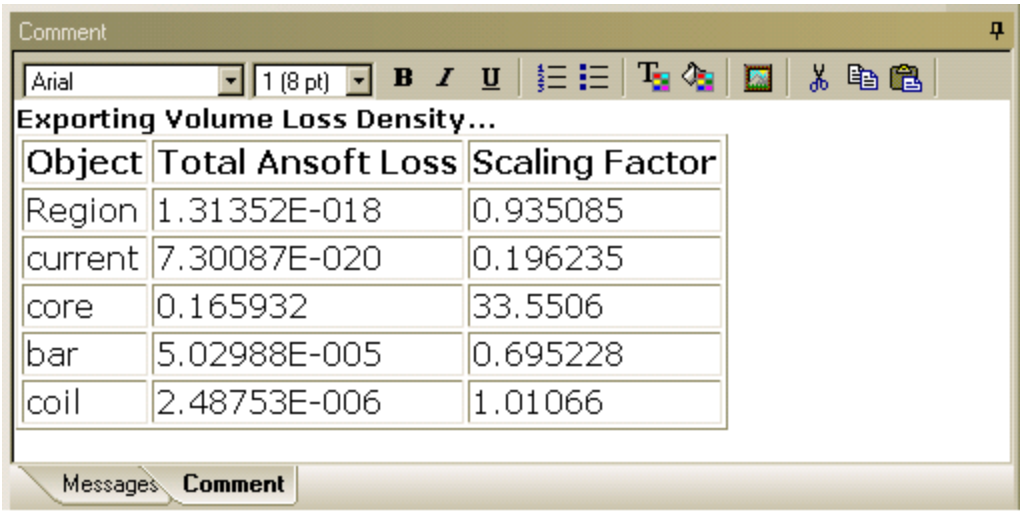
14. Click on the **Imported Heat Generation** tree item to view its details.



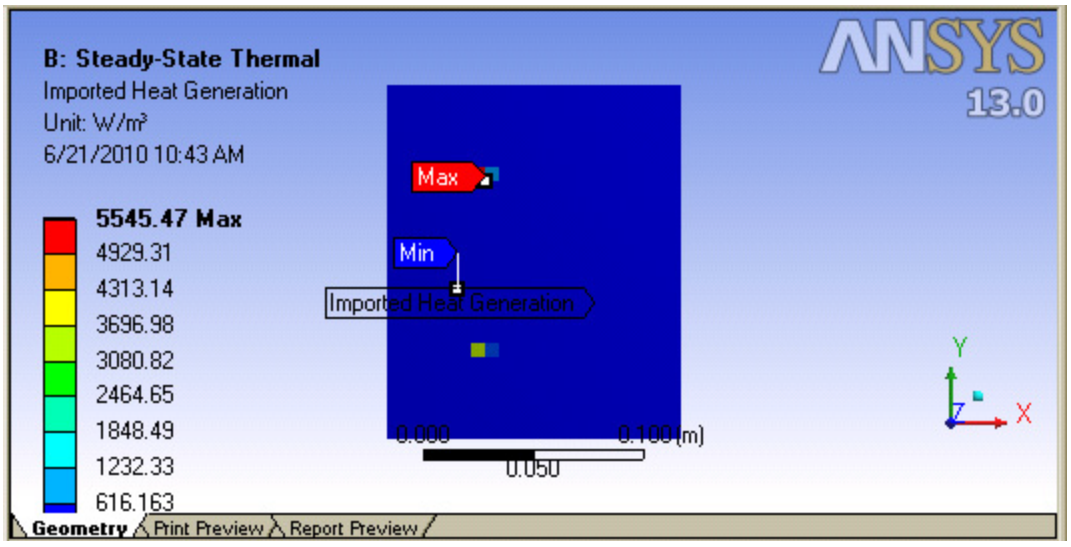
15. In the **Transfer Definition** section, users can set up the source Maxwell solution by pulling down the **Ansoft Solution** combo box and select one of the listed Maxwell solutions.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16. Right-clicking on **Imported Heat Generation** > **Import Load** will import loss from the **Ansoft Solution** selected for this load. After import has completed, the **Import Heat Generation** item becomes a folder, and an entry called **Imported Load Transfer Summary** is listed. Select the **Imported Load Transfer Summary** entry and the scaling factors used to export the load from Maxwell will be displayed in the **Comment** window.

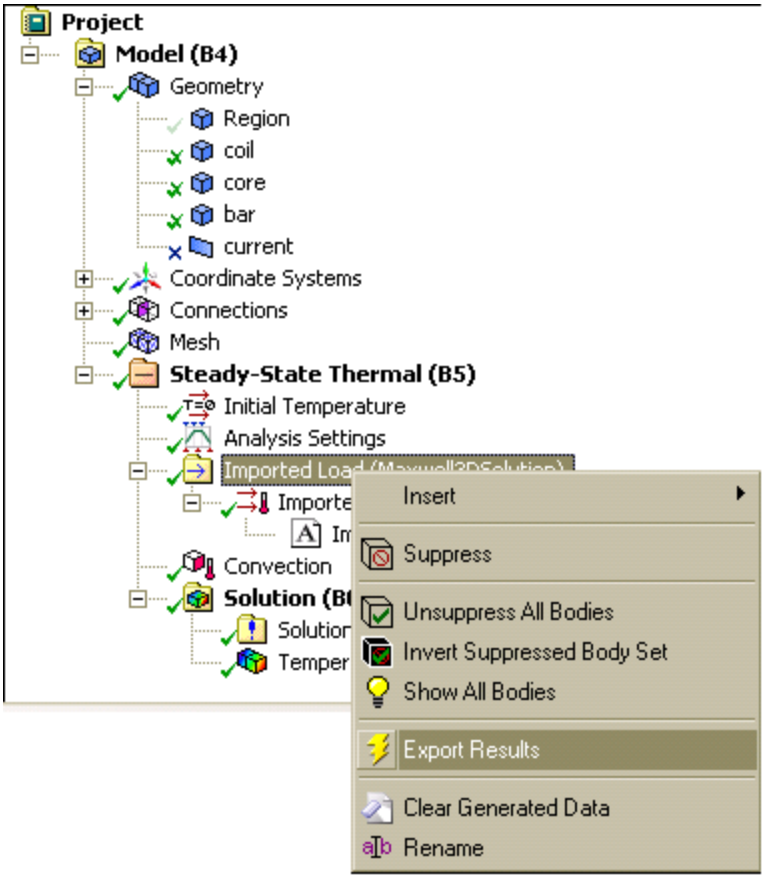


17. Selecting **Import Heat Generation** should show an overlay plot of the imported load. The loss mapping from Maxwell can be verified by comparing this overlay plot with an Ohmic-Loss field overlay plot in Maxwell.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

18. Create a **Convection** boundary to complete the thermal setup:
    a. Use **Edge** select by clicking the icon at the **Mechanical Toolbar** and then **Edit** > **Select All**.
    b. With the edges selected, right-click on the **Steady-State Thermal** project tree item and insert a **Convection**.
    c. With the **Convection** item selected, change its **Film Coefficient** to **5 W/m2** via the **Detail** window.
    d. Right-click on the **Solution** tree item and select **Solve**.
    e. After the thermal solution is finished, insert a **Temperature** plot by right-clicking on **Solution** and selecting **Insert > Thermal** > **Temperature**.
    f. Right-click on the newly inserted **Temperature** item and select **Evaluate All Results**.
19. To export the thermal result to Maxwell, right-click on the **Imported Load (Maxwell2DSolution)**, or **Imported Load(Maxwell3DSolution)** and select **Export Results**.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20. To fully utilize the automation capabilities provided in **Ansys Workbench**, select **Imported Load (Maxwell2DSolution)**, or **Imported Load(Maxwell3DSolution)**; and in its **Detail** window, select **Yes** for **Export after Solve**. With this option selected, users can continue the iteration between Maxwell/Thermal simulations from the Workbench schematic.

    To “push” the exported thermal results back to Maxwell, right-click on Maxwell's **Solution** cell on the Workbench schematic and select **Enable Update**. Then, right-click again on Maxwell's **Solution** cell and select **Update**. This will trigger Maxwell to re-simulate its solution with thermal results.

    To continue the solve iterations, repeat the following steps as needed:

    a. Right-click on Thermal's **Setup** cell and select **Refresh**.
    b. Right-click on Thermal's **Setup** cell and select **Update**.
    c. Right-click on Maxwell's **Solution** cell and select **Enable Update**.
    d. Right-click on Maxwell's **Solution** cell and select **Update**.

Maxwell to Ansys Stress Feedback Coupling

Maxwell supports stress feedback coupling for 2D/3D transient, magnetostatic, AC magnetic, and electrostatic designs via the Ansys Workbench schematic.

> **Note:** A Maxwell system can also be coupled with **both** thermal and stress systems via Workbench.

Maxwell-Specific Considerations

- Users must select object/s (i.e., geometry) in Maxwell upon which displacement data from Ansys Mechanical should be considered. This is done to avoid applying displacement to the whole model while only certain part(s) are sensitive to stress. This is a per-design setting similar to the Set Object Temperature setting for thermal feedback.
- Each frequency point in an AC magnetic sweep will be solved with the same feedback data.
- Displacement data that has already been exported from Ansys Mechanical Stress is invalidated when the Maxwell design is edited such that the previously simulated solution is considered as invalid or non-current. These edits include
    - Modifying the geometries or selecting “Revert To Initial Mesh.”
    - Modifying the boundary/excitation settings, such as changing the value of a “Current excitation” in a magnetostatic design.
- Maxwell solutions can incorporate feedback from both thermal and stress.
- Once the Maxwell solution has been integrated with displacement/temperature, the adaptive simulation process of a static simulation is disabled. Users must then manually

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

invalidate the displacement/temperature data in the existing solution before the adaptive simulation process is re-enabled.

- Simulation iteration is controlled by user.

Maximum Force Density (AC) Components Calculation

In Maxwell, there are two options to couple the eddy solver with the Mechanical structure static solver. You can choose either DC force density or Maximum force density. The maximum force density includes the contribution from AC components.

In Maxwell stress tensor,

$$\sigma_{ij} = B_i B_j - \frac{1}{2\mu_0} B^2 \delta_{ij}$$

The DC (averaged) components of B_iB_j is calculated by

$$B_i B_j|_{DC} = \frac{1}{2\mu_0} Re\left[\dot{B}_i \grave{B}_j\right]$$

and the AC (fluctuation) components

$$B_i B_j|_{AC} = \frac{1}{2\mu_0} Re\left[\dot{B}_i \grave{B}_j\right]$$

where

$$\dot{B} = B_{Re} + jB_{im}$$

And $\grave{B}$ is the complex conjugate of $\dot{B}$. The maximum force is calculated as

$$\vec{F} = \frac{\vec{F}_{DC}}{\left|\vec{F}_{DC}\right|}\left(\left|\vec{F}_{DC}\right| + \left|\vec{F}_{AC}\right|\right)$$

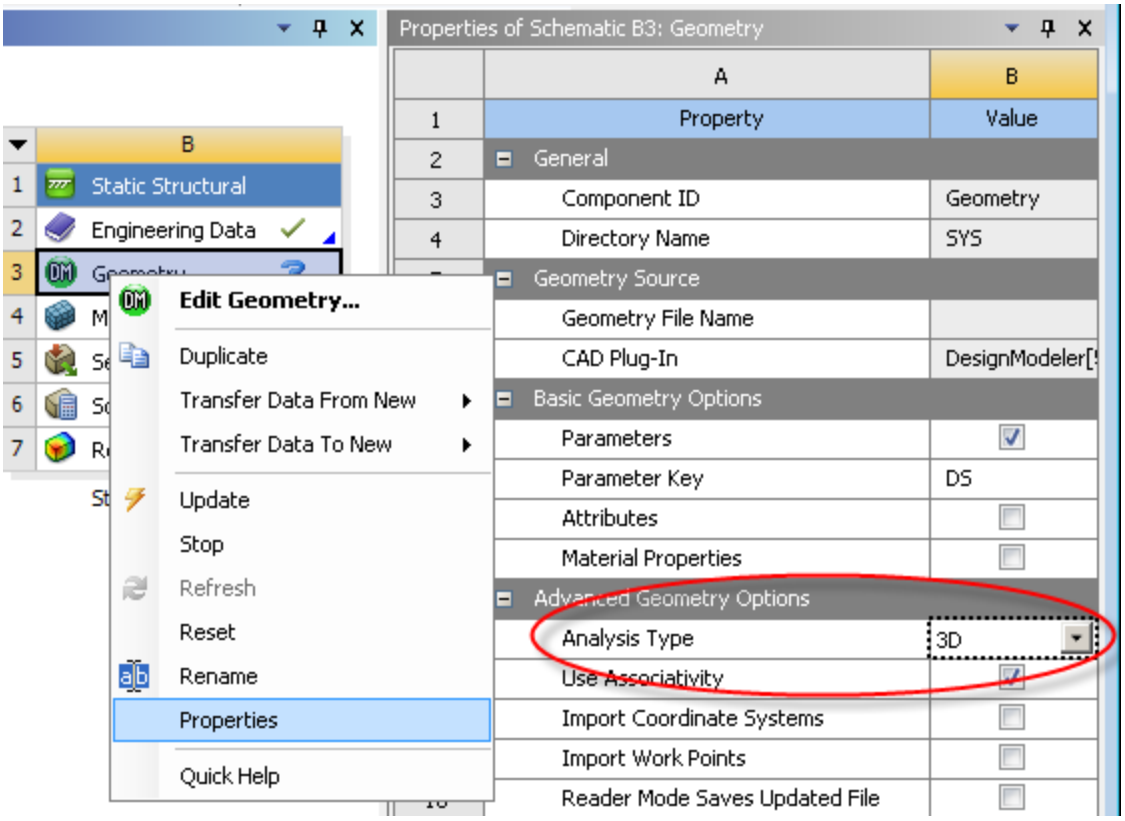
Coupling Maxwell Designs with Ansys Structural via Workbench

Stress feedback coupling between Maxwell 2D/3D and Ansys Structural is supported via the Workbench schematic. Stress feedback is supported for Maxwell magnetostatic, AC magnetic, and transient types. Users also need to set up the design and geometry appropriately. An appropriate design has one or more solve setups that are enabled for stress feedback. The process for stress feedback coupling between Maxwell and Ansys Structural is similar to that described in Coupling Maxwell Designs with Ansys Thermal via Workbench.

1. The easiest way to add a Maxwell 2D or 3D design to a Workbench schematic is to import a working design via **WorkbenchFile** > **Import**. The imported design is placed in the **Workbench** schematic after it is successfully imported.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

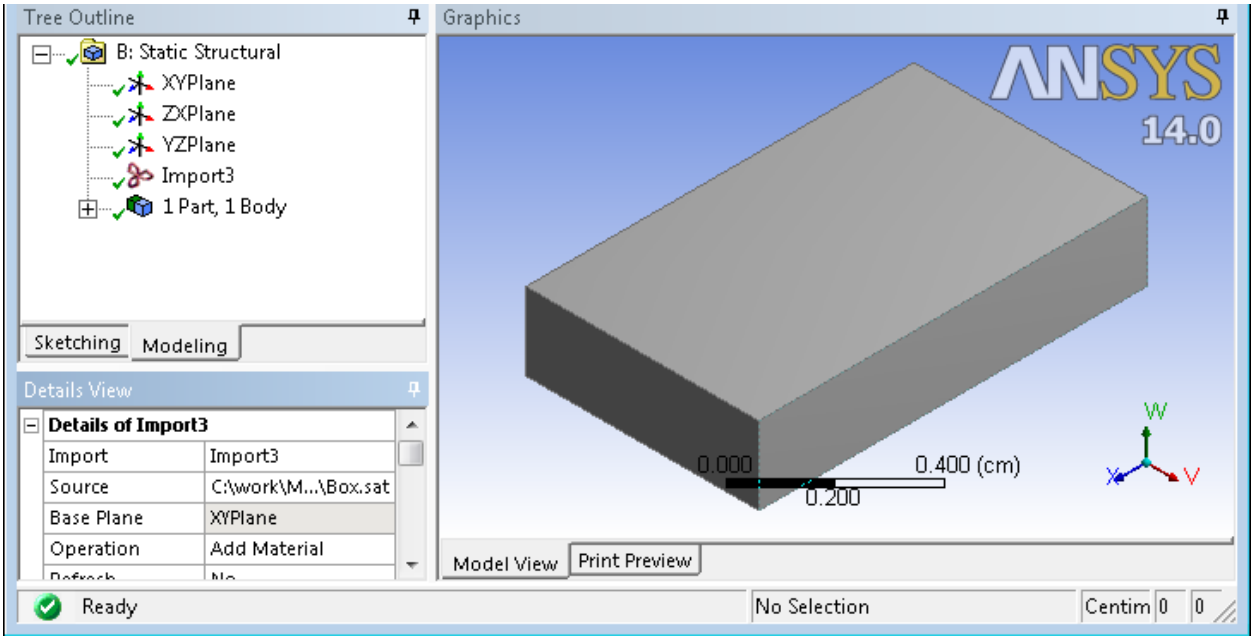
2. Next, insert a **Static Structural** system and change its **Analysis Type** to **2D** or **3D**, (depending on the Maxwell design type) by right-clicking on the **Geometry** cell and selecting **Properties**. It is important to change the **Steady-State Thermal** system's analysis type *before* setting up its geometry.



3. To set up the **Static Structural** system's geometry, you must first export the Maxwell geometry using **sat** or **step** format as follows:
    a. Select the **Modeler** > **Export** menu item.
    b. Select the desired model geometry format (**sat** or **step**), and the save location in the dialog box and save the file for use by Ansys Workbench.
4. Import the file via the **Geometry** module of the **Static Structural** system.
    a. To access the **Geometry** module, double-click on the **Geometry** cell in the **Static Structural** system to launch **Design Modeler**.
    b. Select **File** > **Import External Geometry File** and browse for the geometry file exported from Maxwell.
    c. After the geometry file is imported, right-click on the root folder of the modeler project tree and select **Generate**. (When the geometry file is of a Maxwell 2D RZ design, users can rotate the geometry in **Design Modeler** by creating a body

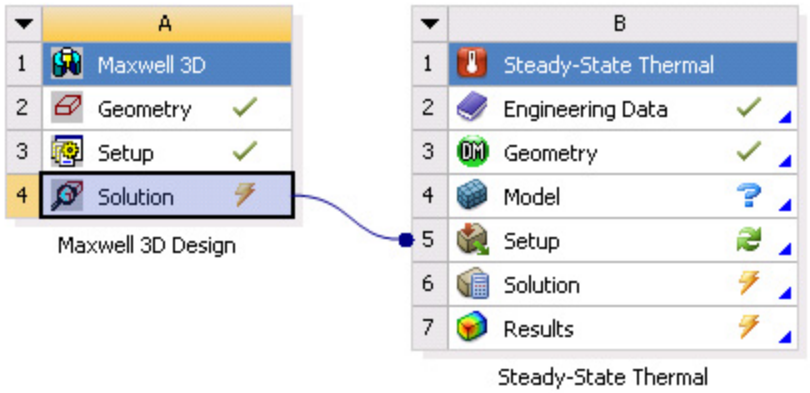
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

operation.)



5. Close **Design Modeler** and refresh the **Model** cell of the **Static Structural** system by right-clicking the **Model** cell and selecting **Refresh**.
6. The geometry mode of the **Static Structural** system can be changed via the Ansys **Mechanical** user interface.
    a. Launch **Mechanical** by double-clicking the **Setup** cell of the **Static Structural** system.
    b. Select **Geometry** in the project tree and the **Definition** of **Geometry** will be shown in the **Detail** window.
    c. Select either **Flexible** or **Rigid** as the value for the property 2D (or 3D) **Stiffness Behavior**.
7. To set up the coupling, drag the **Solution** cell of the Maxwell system and drop it on the **Setup** cell of the **Static Structural** system.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



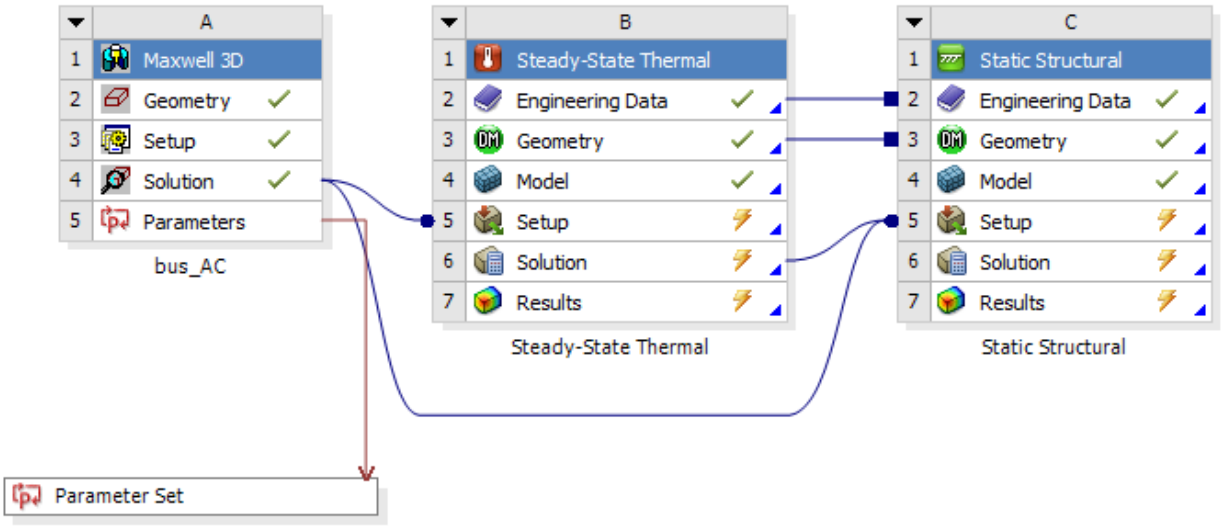
8. Note that the Maxwell **Solution** cell is tagged with a “Lighting Bolt” symbol. Right-click on the Maxwell **Solution** cell and select **Update**. This will initiate a Maxwell simulation if it is not already solved. Once Maxwell's solution is available, the “Lighting Bolt” changes to a “Green Check” symbol.
9. To “push” the coupling into **Static Structural**, right-click on the **Static Structural Setup** cell and select **Refresh**.
10. After refresh is finished, you can launch Ansys **Mechanical** by double-clicking the **Setup** cell to finish the coupling setup, which is similar to that described in Coupling Maxwell Designs with Ansys Thermal via Workbench.

Coupling Maxwell with Ansys Thermal and Structural via Workbench

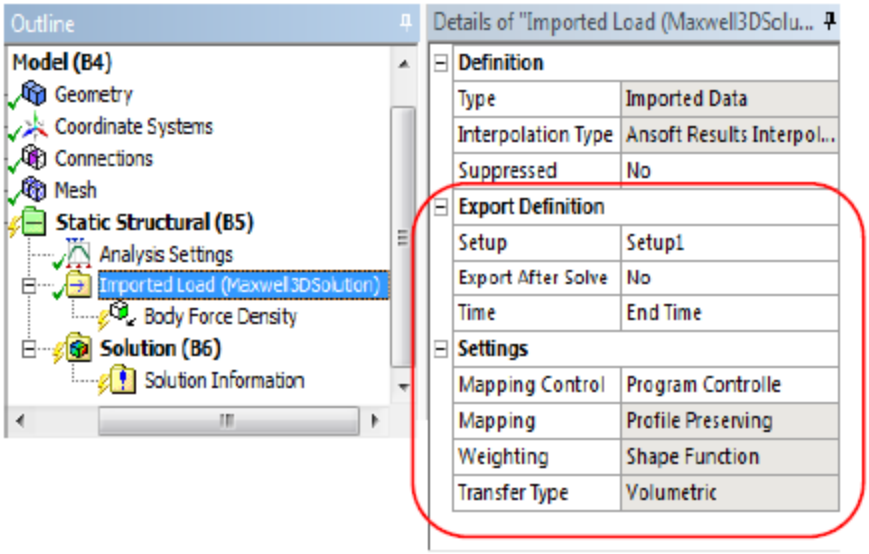
A Maxwell system can also be coupled with *both* thermal and stress systems via Workbench. The thermal system then serves as an “up-stream” system for the stress system. This means that the stress analysis takes both electromagnetic and thermal forces into consideration. In this scenario the user must select the same object(s) in the thermal and stress systems to apply the respective imported loads.

To set up the coupling, drag the **Solution** cell of the Maxwell system and drop it at the **Setup** cell of a Thermal and Static Structural (Stress) system. The user also needs to couple the Thermal and Static Structural systems to capture the effect of thermal force.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



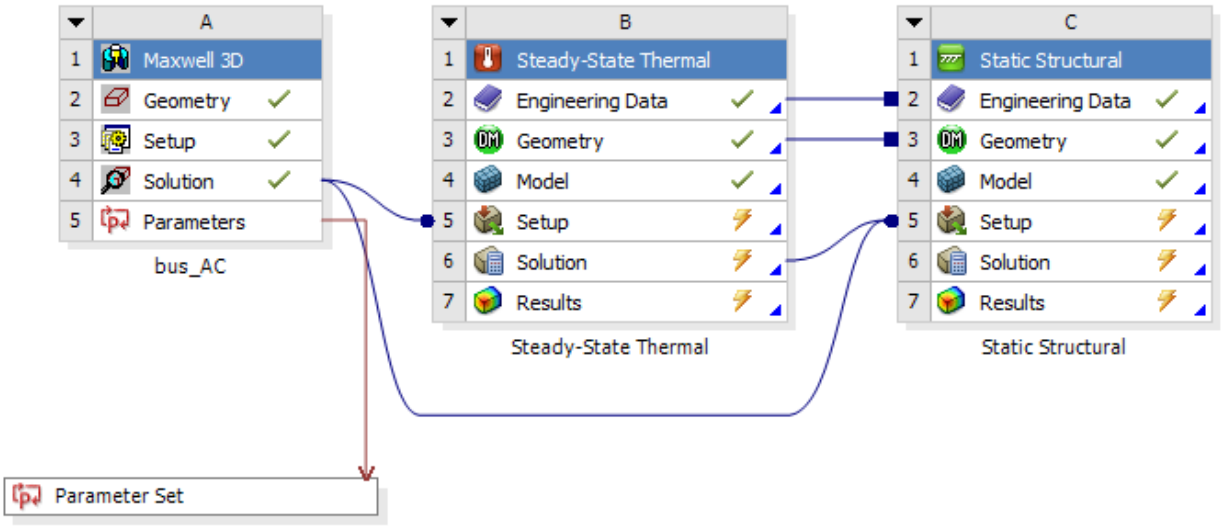
The coupling framework is built on top of the existing Thermal Feedback. The Stress system will have additional Export Definition and Mapping Settings like Thermal.



Example 1: One Iteration

The following image illustrates a coupling setup where both the Thermal and Stress system are ready to be updated. The Maxwell adaptive solution has converged in three passes. The Maxwell system has both thermal and stress feedback enabled.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

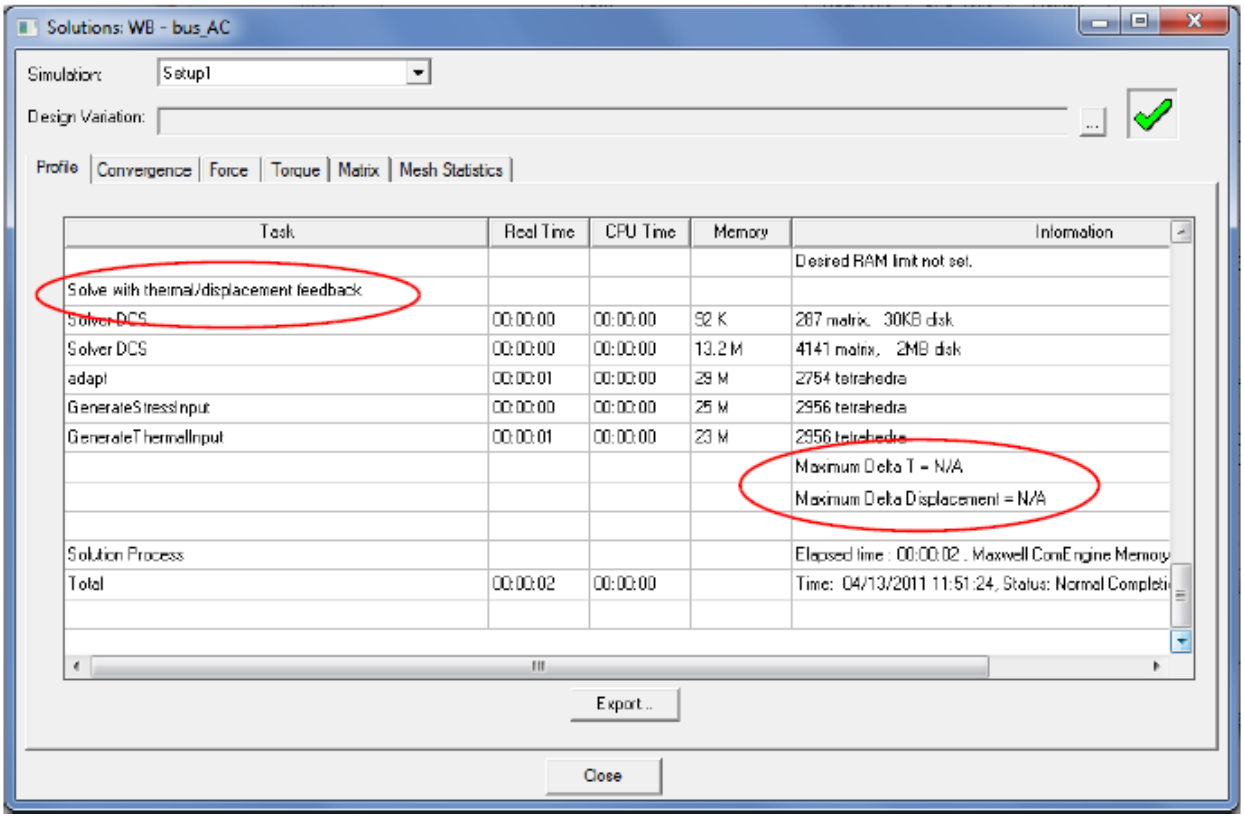


1. Right-click on the **Solution** cell of Stress (Static Structural) and select **Update**.
   a. The **Setup** of Thermal will be Updated with “em” loss from Maxwell.
   b. The **Solution** of Thermal will be Updated and temperature will be exported to Maxwell.
   c. The **Setup** of Stress will be Updated with thermal force from Thermal and force density from Maxwell.
   d. Displacement will be exported to Maxwell after Stress finishes simulation.
2. Right-click on the **Solution** cell of Maxwell and select **Enable Update**.
3. Right-click on the **Solution** cell of Maxwell and select **Update**.

Maxwell resimulates the third pass with its mesh and the temperature/displacement feedback.

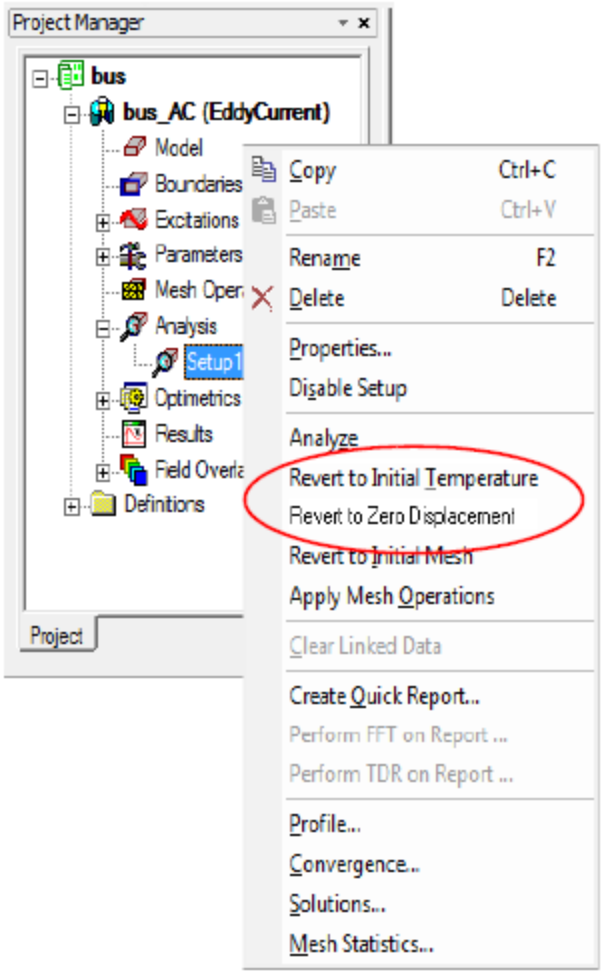
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The profile will show information about the feedback:



Both **Revert to Initial Temperature** and **Revert to Zero Displacement** menus are present in the Analysis Setup context menu.

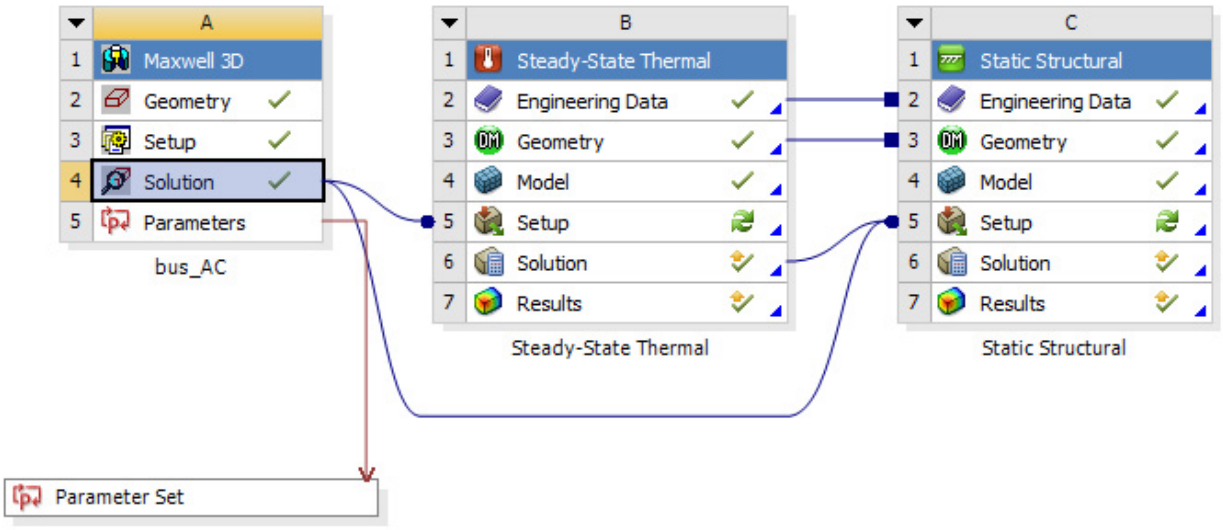
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Example 2: Manual Iteration

The following image illustrates the Workbench schematic after Example 1, where the first iteration has completed.

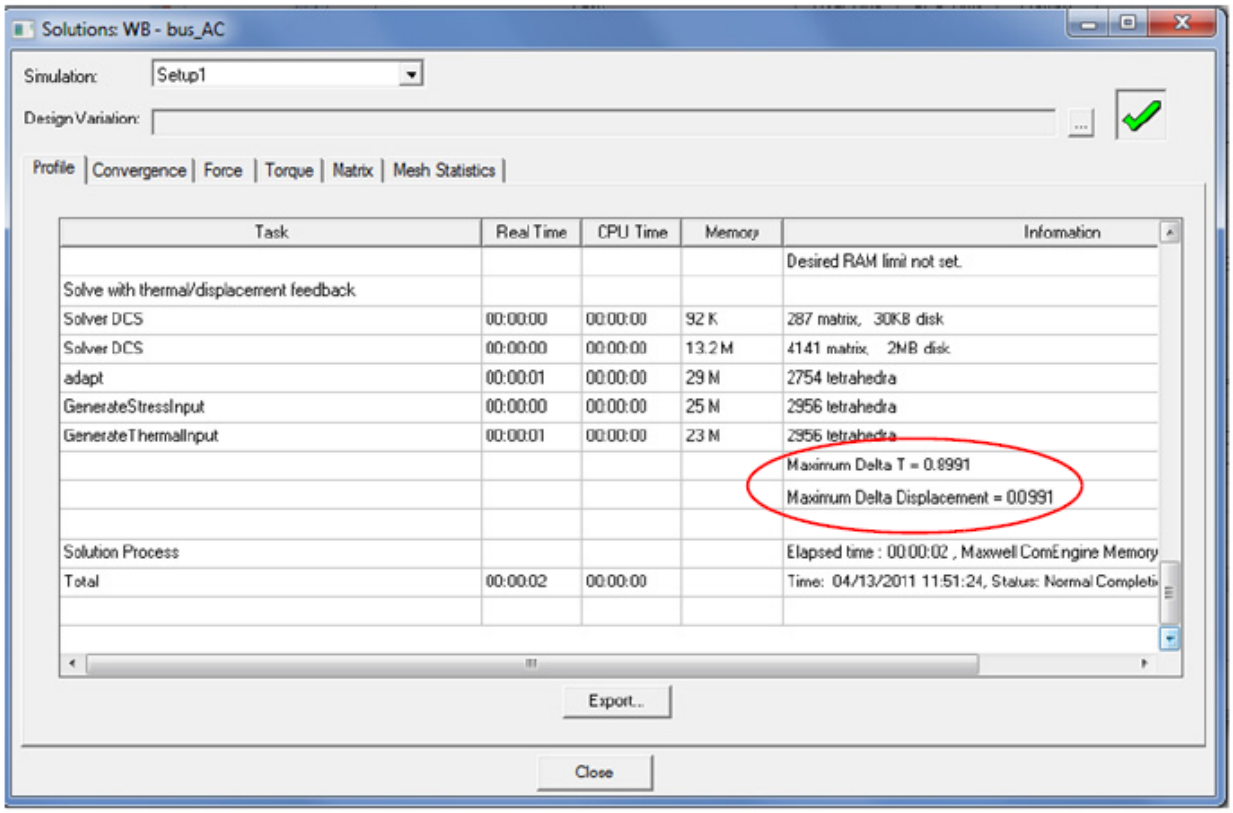
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



1. Right-click on the **Solution** cell of Stress and select **Update**.
   - The **Setup** of Thermal will be Refreshed and Updated with “em” loss from Maxwell.
   - The **Solution** of Thermal will be Updated and temperature will be exported to Maxwell.
   - The **Setup** of Stress will be Refreshed and Updated with thermal force from Thermal and force density from Maxwell.
   - Displacement will be exported to Maxwell after Stress finishes simulation.
2. Right-click on the **Solution** cell of Maxwell and select **Enable Update**.
3. Right-click on the **Solution** cell of Maxwell and select **Update**.

Maxwell resimulates the third pass with its mesh and *new* temperature/displacement feedback. The profile will show information about the feedback:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



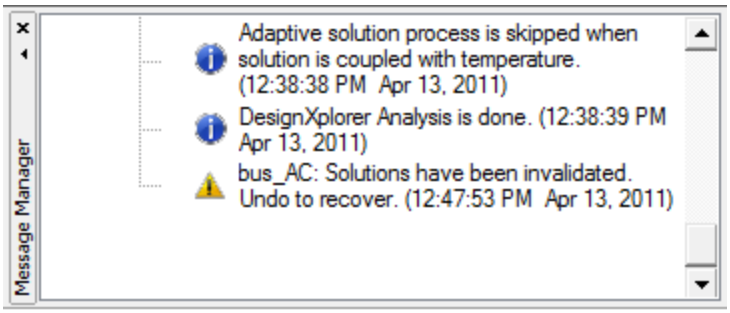
Note that the delta temperature and displacement is being reported in the profile.

Example 3: Revert Maxwell Solution

The temperature and displacement in the Maxwell solution can be reverted separately.

1. Select **Revert to Initial Temperature** in Maxwell.

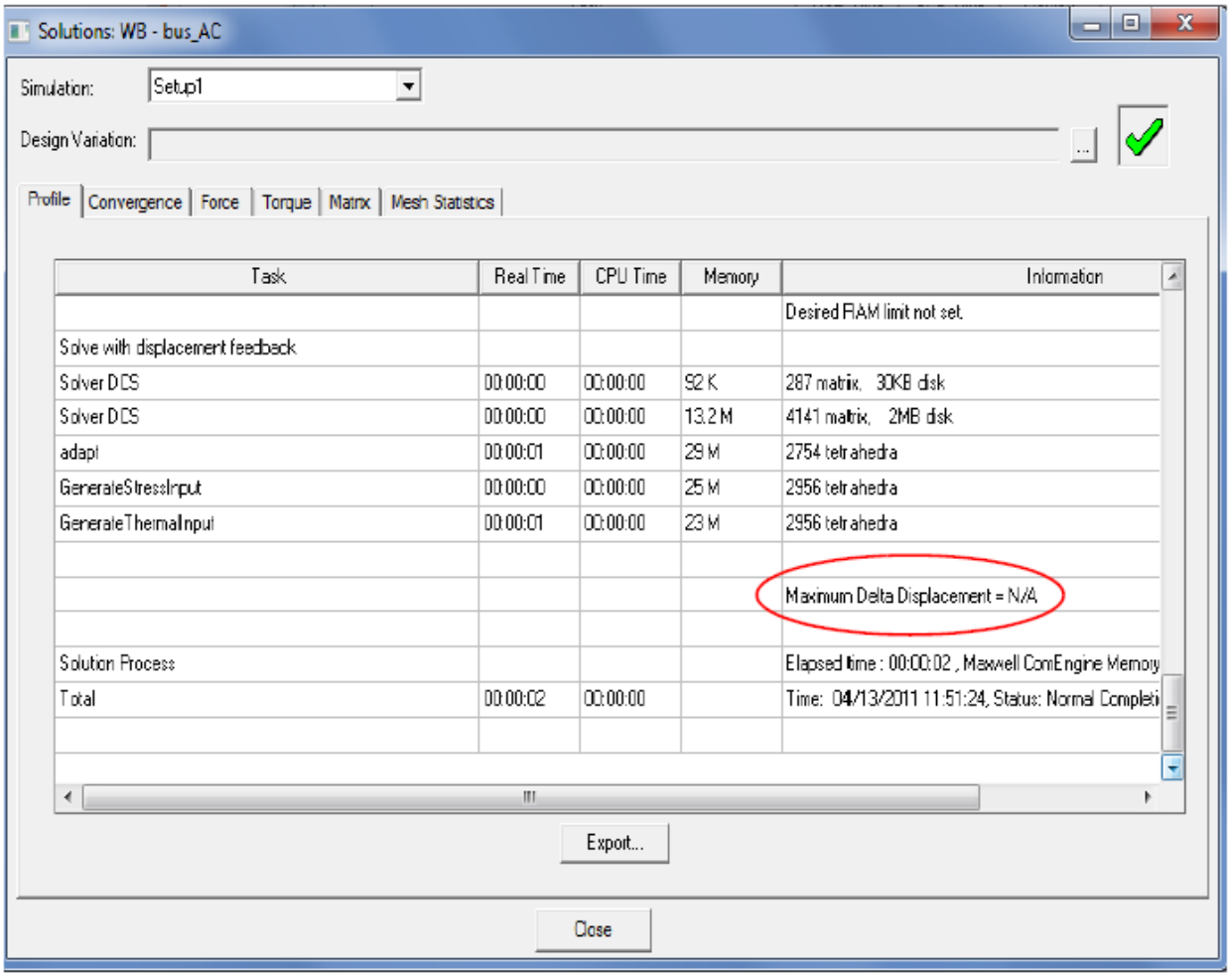
   A warning message displays notifying users about the invalidation of solution.



2. Right-click on Maxwell analysis setup and select **Analyze**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

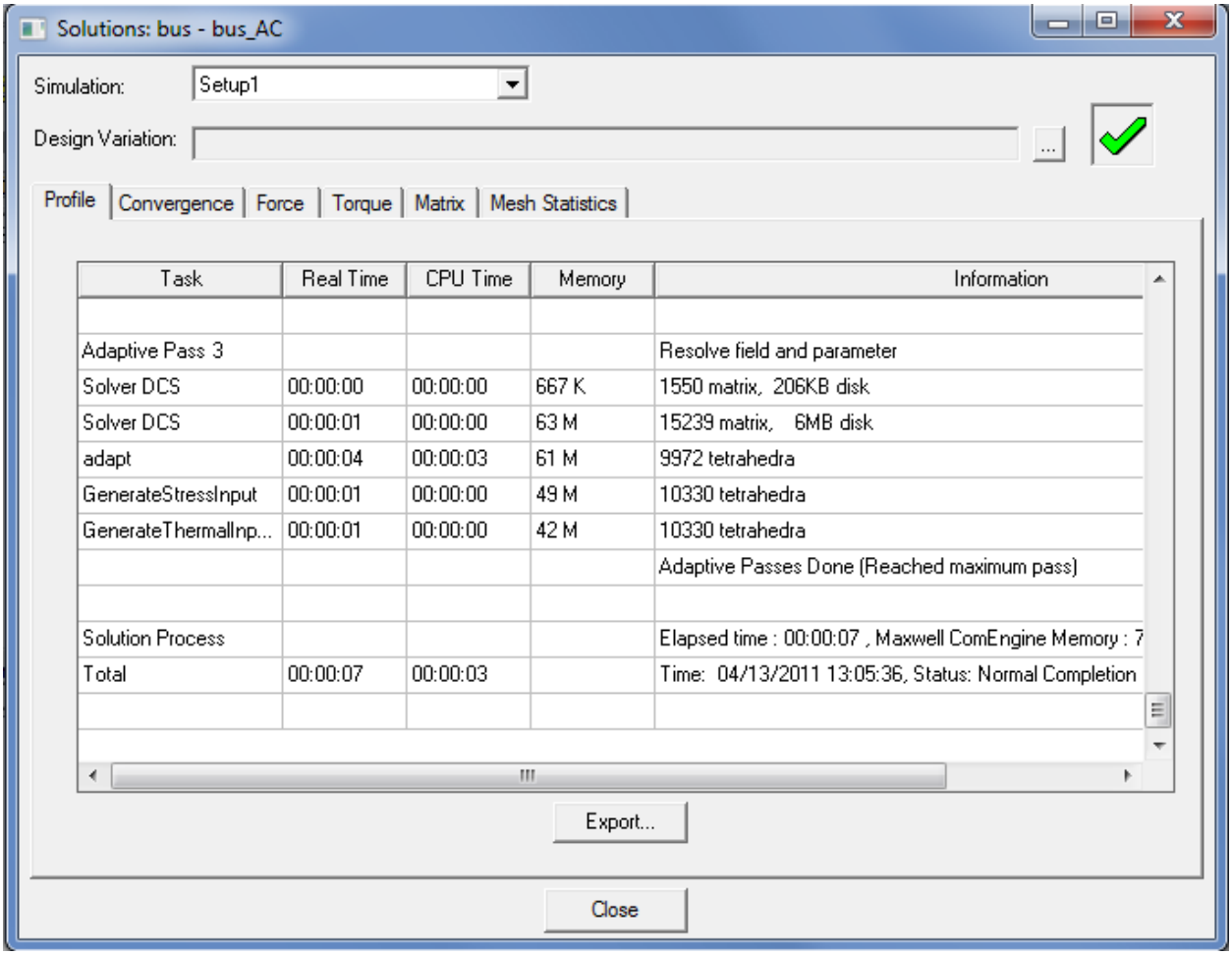
Maxwell resimulates the third pass with its mesh and displacement that was previously exported from Mechanical, but without temperature.



3. Select **Revert to Zero Displacement** in Maxwell, then right-click on Maxwell analysis setup and select **Analyze**.

Maxwell resimulates the third pass with its mesh without either temperature nor displacement.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



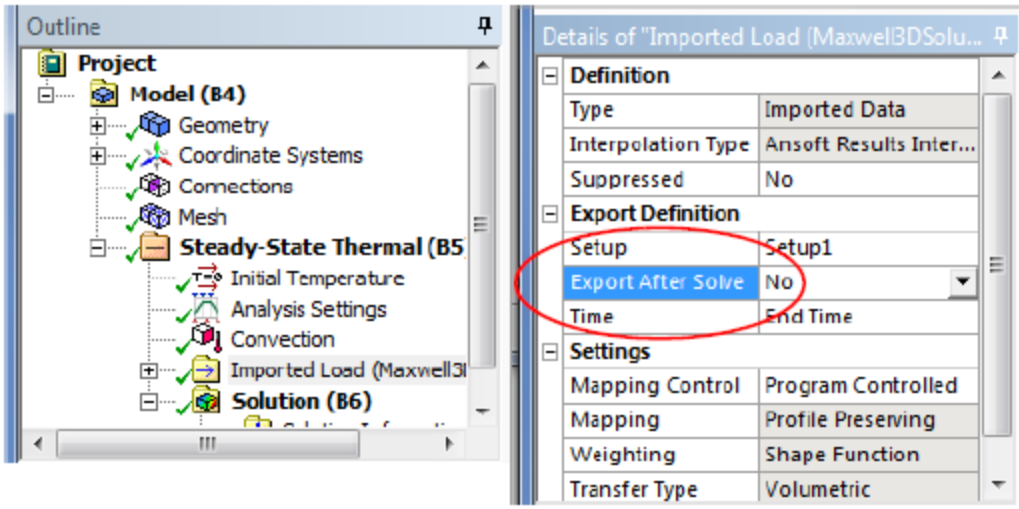
Example 4: Only Stress Feedback

The coupling setup is the same as in Example 1, with either one of the following differences:

- The Maxwell system is not enabled for thermal feedback (via SetObjectDisplacement). Note that because Maxwell is not enabled to support feedback, the **Export Result** properties should be available in the thermal system.
- The Maxwell system is enabled for thermal feedback, but users choose not to **Export**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Result in the thermal system, which disables the automatic **Export Results**.

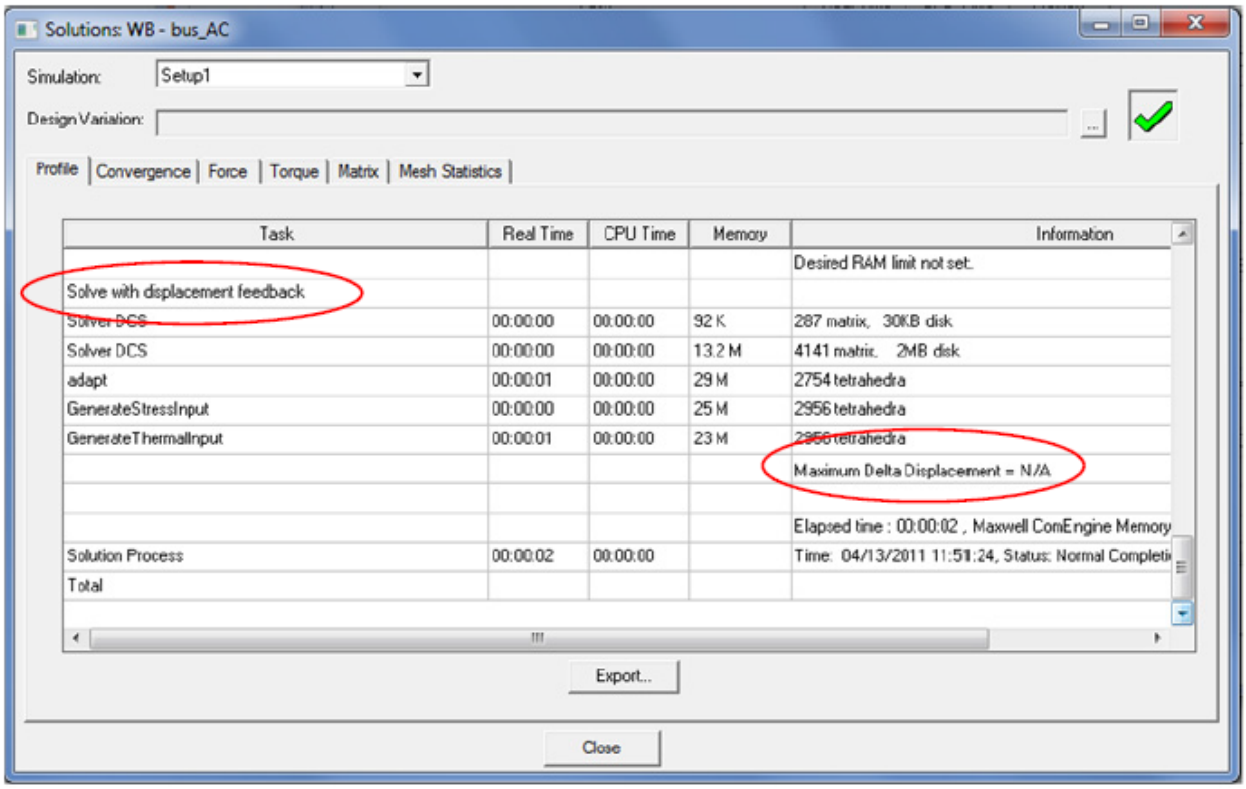


1. Right-click on the **Solution** cell of Stress and select **Update**.
   - The **Setup** of Thermal will be updated with EM loss from Maxwell.
   - The **Solution** of Thermal will be updated.
   - The **Setup** of Stress will be updated with thermal force from Thermal and force density from Maxwell.
   - Displacement will be exported to Maxwell after Stress finishes simulation.
2. Right-click on the **Solution** cell of Maxwell and select **Enable Update**.
3. Right-click on the **Solution** cell of Maxwell and select **Update**.

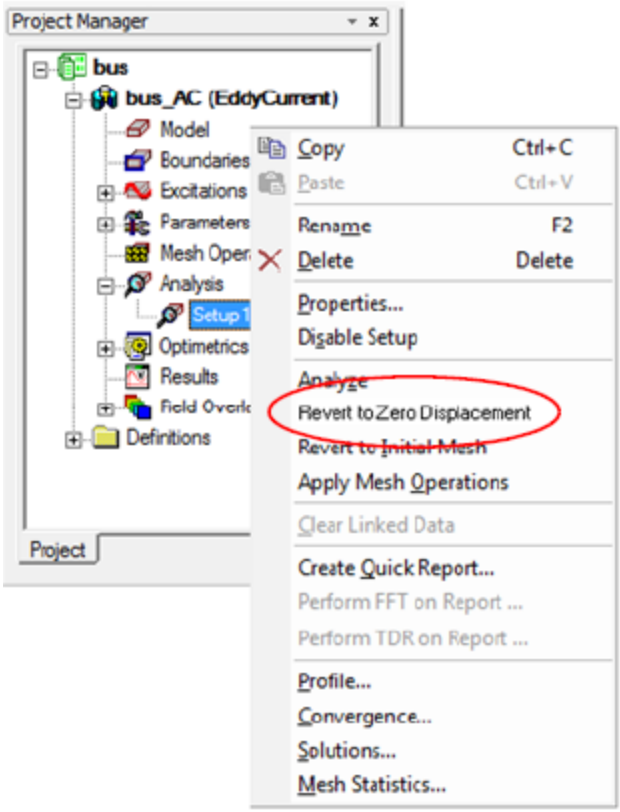
Maxwell resimulates the third pass with its mesh and the displacement feedback.

The profile shows information about the feedback and **Revert to Zero Displacement** will be available in the solve setup context menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

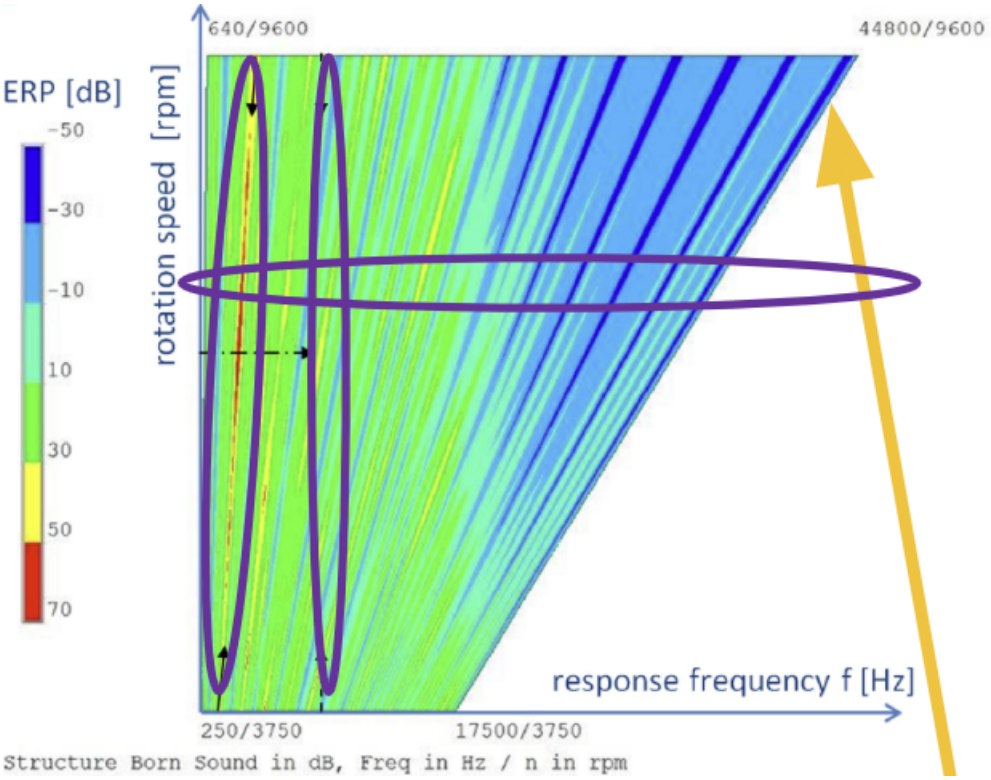


Maxwell Multiple RPM Harmonic Force Coupling

Waterfall analysis is a tool that is often used to evaluate the acoustic and/or vibratory performance of rotating machinery. Maxwell multiple RPM coupled with Ansys Harmonic Response provides a powerful tool to generate acoustic performance map of electric machines. The process involves calculating harmonic force using discrete Fourier transform (DFT) methods at each rpm (revolutions per minute), and performing harmonic response analysis at every selected frequency with calculated harmonic force loads.

Waterfall computation is given by ERP=ERP(f, rpm) making use of load interpolation between computed RPM.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Waterfall contour map shows the full acoustic fingerprint of the machine, where:

- Vertical Line: Given by one fixed eigenfrequency excited at different speeds of rotation.
- Horizontal Line: Single rpm load case yielding one frequency response function.
- Inclined Line: Excitation order of the electromagnetic force-density revealing a large ERP zone with its origin in single eigenfrequency that is met by both frequency and spatial load pattern of excitation and system mode.

To set up a Maxwell design for multiple RPMs coupling:

1. In the **Mechanical** tab of the **Motion Setup** window, specify angular velocity (RPM) with a design variable.
2. In the **General** tab of the **Solve Setup** window, specify the appropriate stop time and timestep using the same design variable.
3. Add a parametric setup to sweep the variable.
4. Select the parametric setup as **Embedded Parametric Analysis** in the DesignXplorer setup.

> **Note:** For multiple RPM harmonic force interpolation of inverter-fed electric motors, it is best to use the technique outlined in Harmonic Force Coupling for Inverter-Fed Motors.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Harmonic Force Coupling for Inverter-Fed Machines

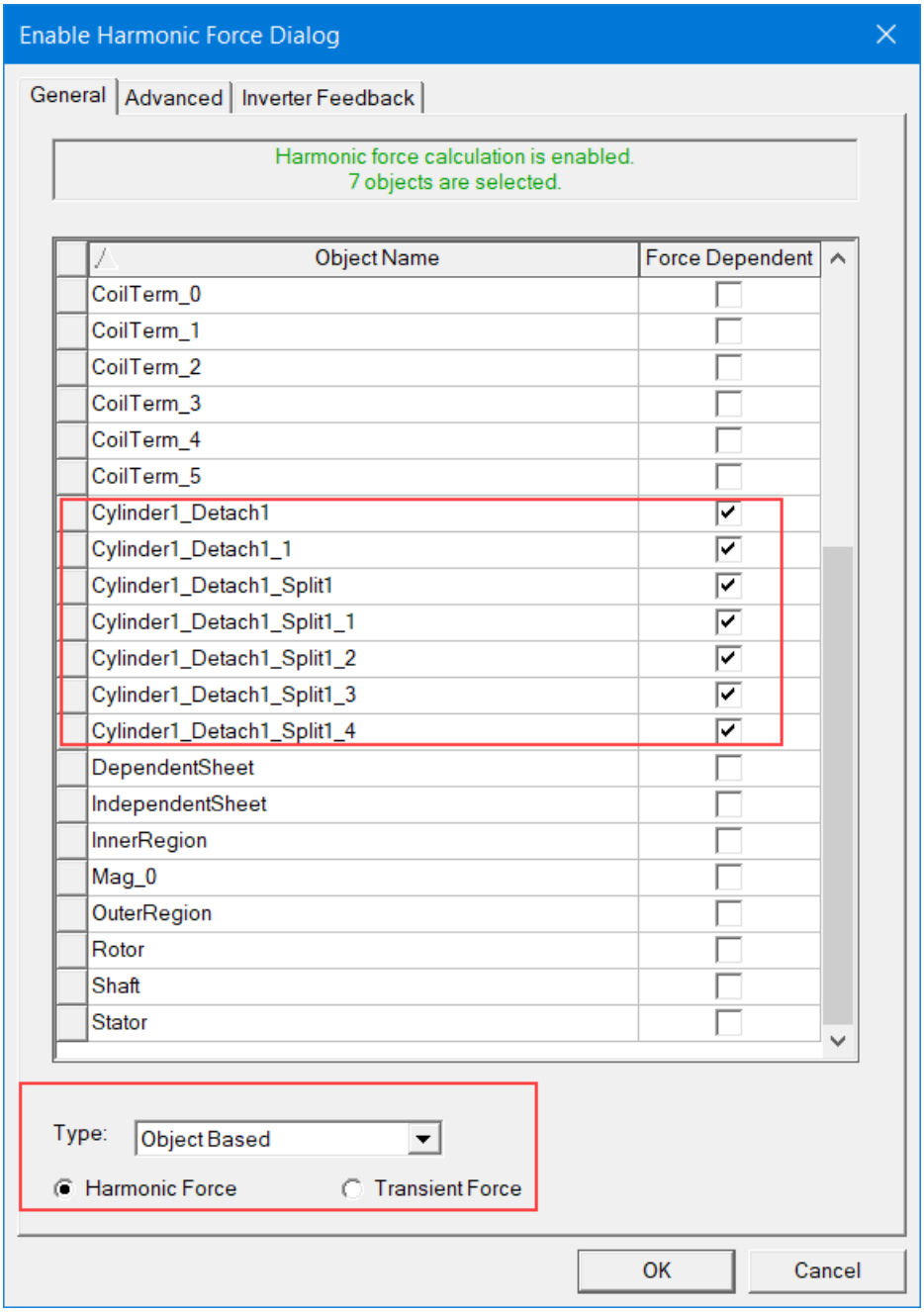
While the approach described in Maxwell Multiple RPM Harmonic Force Coupling is a good technique for E-motors with sinusoidal feeding, it does not work well for inverter-fed machines. The linear interpolation as done in Maxwell Multiple RPM Harmonic Force Coupling assumes that ALL the harmonics of force and of the response quantities are proportional to the speed. But for an inverter-fed machine, the motor's excitation, and thus the forces, have a frequency content that is independent of the speed.

This topic outlines how to use inverter feedback to generate a 2D interpolation of quantities in the frequency-speed plane. It will derive excitation forces in the frequency domain for speeds not directly calculated in Maxwell and will generate a continuous plot from discrete result data in the mechanical domain. It is applicable to Maxwell 2D and 3D transient T-Omega designs that include object-based harmonic force.

Generate Inverter-Fed Harmonic Force Results

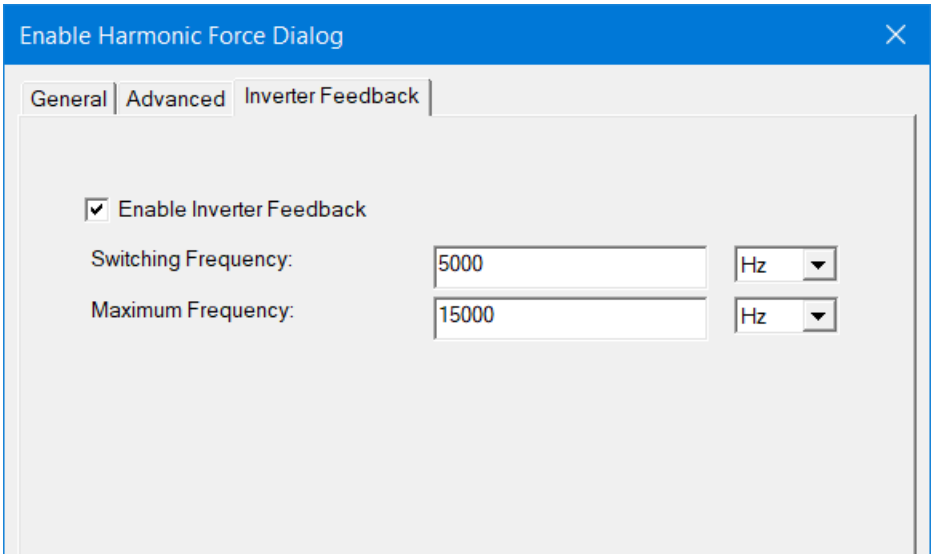
1. For a Maxwell 2D or 3D transient design, select **Maxwell** > **Enable Harmonic Force Calculation**.
2. In the window that opens, select the objects that are force dependent, and make sure the Type is **ObjectBased** and **Harmonic Force** is selected.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. From the **Inverter Feedback** tab, enable feedback, and specify the switching frequency and the maximum frequency. The switching and maximum frequencies are only available if **Enable Inverter Feedback** is checked; their default values are 4000 Hz and 8000 Hz, respectively.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

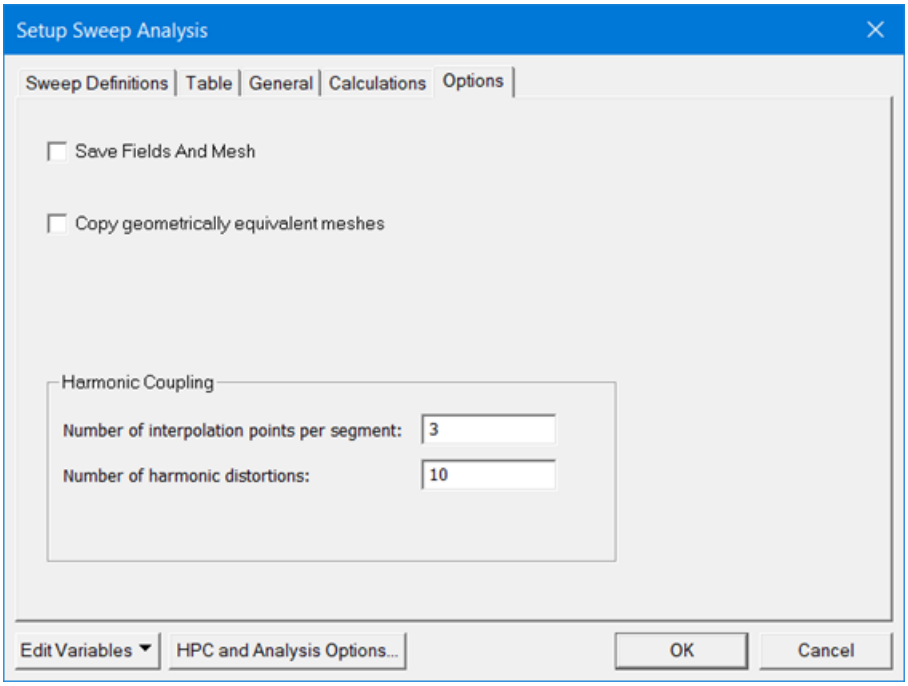


> **Note:** The Switching Frequency should be greater than 0 and can be defined as a variable as long as it does not depend on intrinsic variables.

> **Note:** The Maximum Frequency should be greater than the Switching Frequency, and it **cannot** be defined as a variable.

4. In the Project Manager tree, right-click on the **Optimetrics** entry and select **Add** > **Parametrics and Designs of Experiments** > **Parametric** to create a parametric analysis. In addition to the desired sweep setup, under the **Options** tab, specify number of harmonic distortions; the default value is 10.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

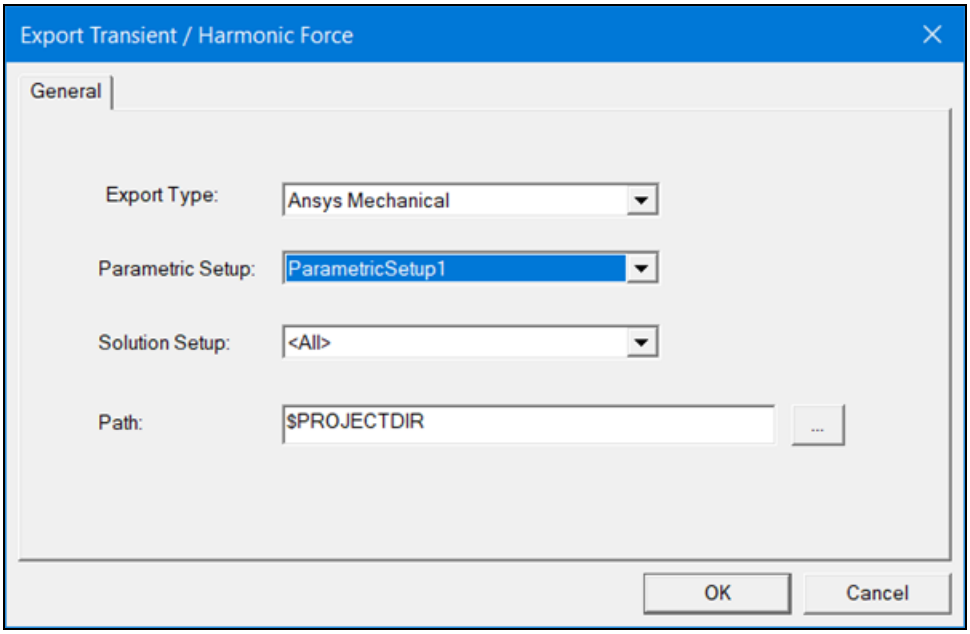


Note: If **Enable Inverter Feedback** is not checked in Enable Harmonic Force dialog box, the **Number of harmonic distortions** will not be available. The **Number of harmonic distortions** should be an integer equal or greater than 2 and **cannot** be defined with a variable.

Note: When setting up harmonic coupling in Ansys Workbench, make sure to use the same **Number of interpolation points per segment** and **Number of harmonic distortions** as you specified in the Setup Sweep Analysis; otherwise, the values specified in DesignXplorer will override the setting of the embedded parametrics setup.

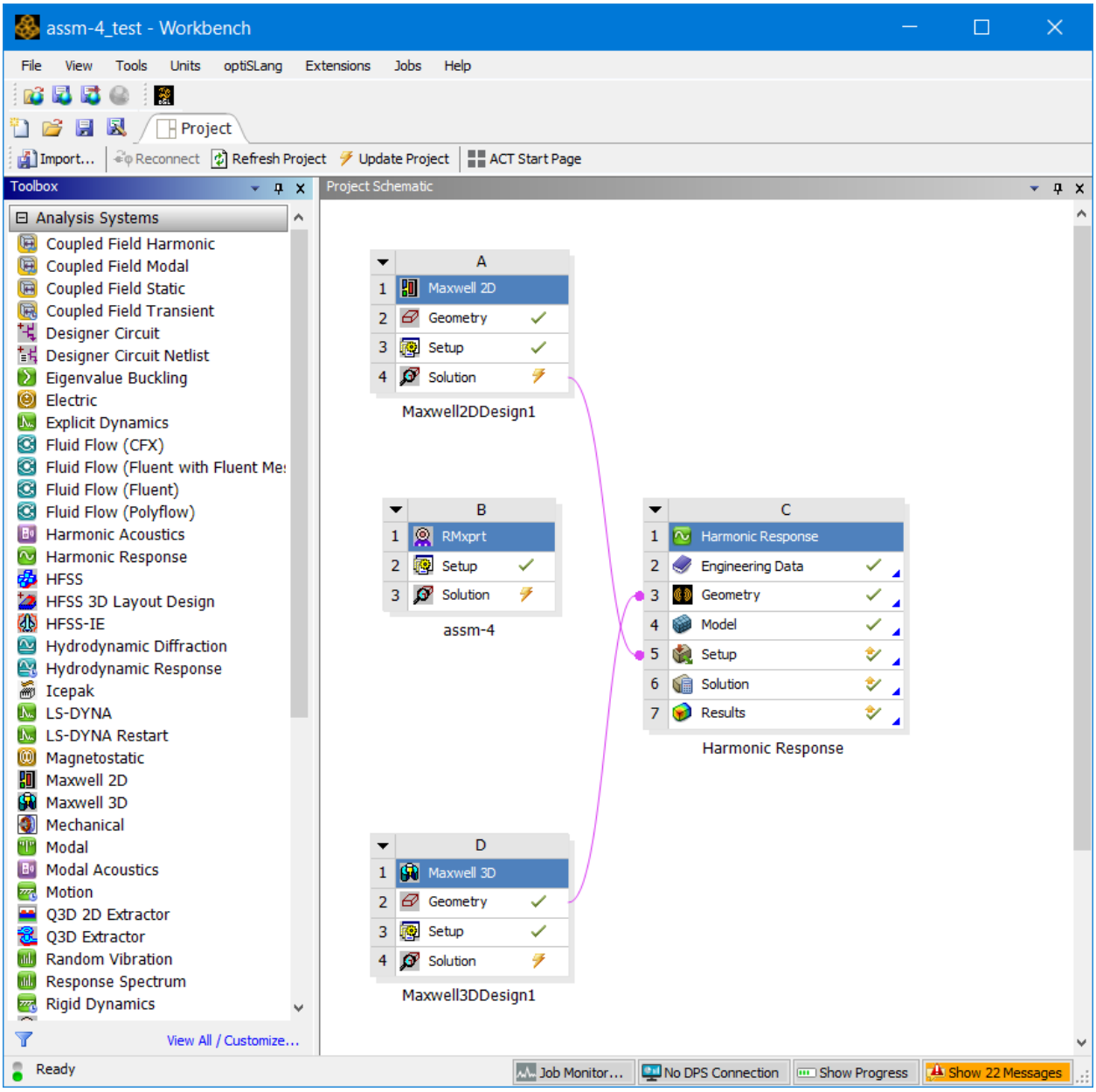
5. After running the analysis in Maxwell, select **Maxwell** > **Export Transient / Harmonic Force** to export harmonic force files to Ansys Workbench:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note: To use Maxwell results in Ansys Workbench, the two modules must be integrated. If you have not already done so, expand the Ansys EM **Suite** entry in the Start Menu, and select **Modify Integration with Ansys 2025 R2.**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



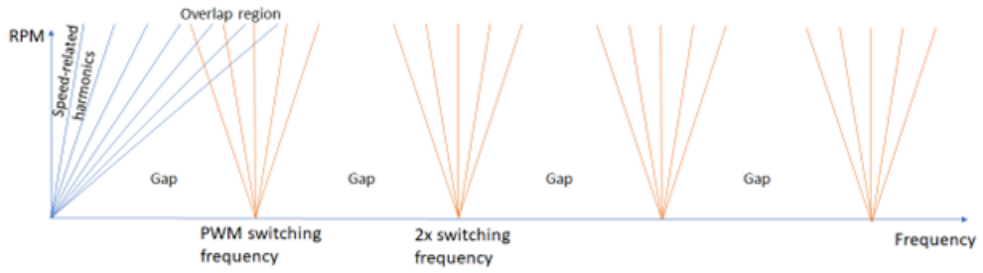
Note: For more information on Maxwell integration with Workbench, see Ansys Workbench Integration Overview.

Limitations

For this feature, there are some limitations, as explained below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the machine is excited by an inverter with constant PWM switching frequency, harmonic forces can be categorized into two groups: The frequency of harmonic forces excited directly by the machine are directly proportional to the machine's speed (speed-related harmonics). In contrast, harmonic forces excited by the inverter feeding are dependent on both the PWM frequency and machine's speed. This leads to "so-called" forward and backward spectral lines in the frequency-speed plane. This pattern of forward and backward lines occurs around the switching frequency and multiples of it.

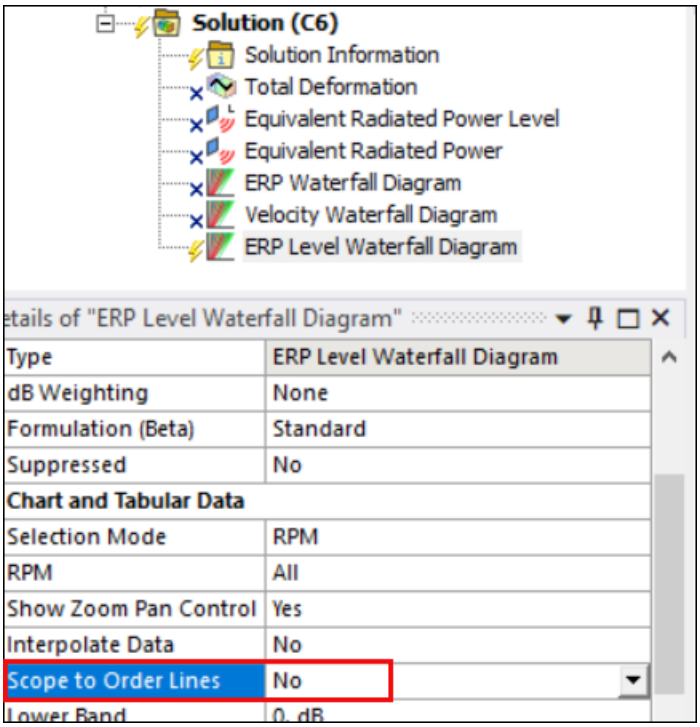


The force interpolation method for inverter-fed electrical machines is based on identifying which forces are excited by the machine or by the inverter feeding; different interpolations are used for these two. So when using this feature, keep in mind the following:

- For the speed-related harmonics, only the first 50 orders are used, and higher frequency data is not used. (This order is based on the number of periods run in the model.)
- When lines of speed-related harmonics intersect with the harmonic lines from the switching frequency, shown as "Overlap region" in the image above, the interpolation for inverter-excited forces will overwrite the interpolation performed for speed-related harmonics. Thus, the interpolated force will be based on the inverter-excited forces only. This can lead to numerical errors when the region of overlap is significant; for example, when a low switching frequency is used.
- In the "Gap" region between different components, no interpolated forces are calculated (a small value is assigned for numerical reasons).
- When generating waterfall diagrams in Harmonic Response analysis in Ansys Mechanical, please set **Scope to Order Lines** to **No** as this option is only valid for speed-

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

related harmonics.



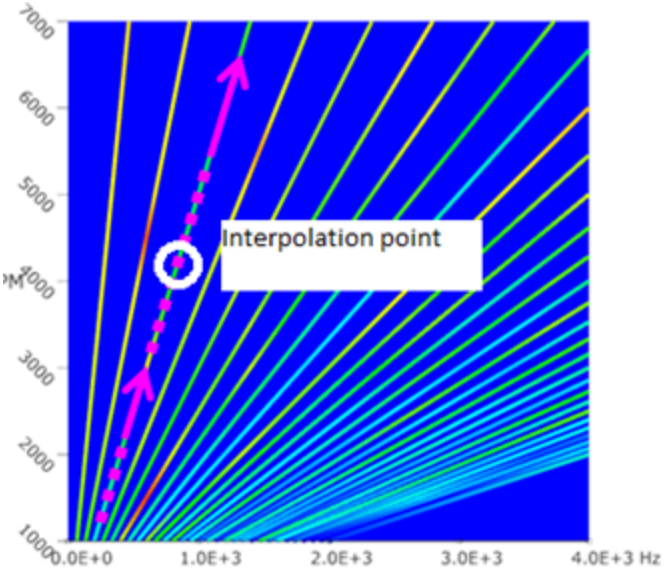
Harmonic Force Interpolation in Frequency-Speed Plane

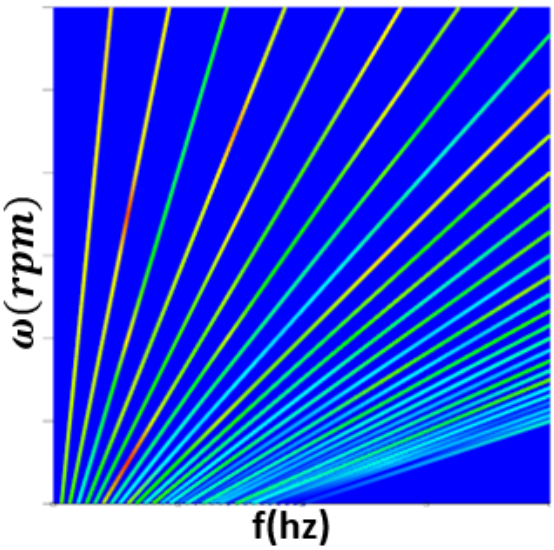
For Electrical Machines

In order to generate the ERP waterfall diagram of electrical machines in Ansys Mechanical Harmonic Response, harmonic force distribution at different rotation speeds from Maxwell solver is provided as excitations of Mechanical. In general, the quality of the ERP diagram depends on the number of speed points that are solved. To reduce the workload in Maxwell, an option is provided to interpolate harmonic force in the frequency-speed plane with a limited number of solved speed points.

In electrical machines, harmonic forces only exist at certain frequencies (harmonics) at a given speed (as shown below):

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Force components and corresponding frequencies at any given speed can be obtained by interpolation along those lines of harmonics:

$$F_k = F_{i-1,k} + \frac{(\omega-\omega_{i-1})}{(\omega_i-\omega_{i-1})}\left(F_{i,k} - F_{i-1,k}\right)$$

$$f_k = f_{i-1,k} + \frac{(\omega-\omega_{i-1})}{(\omega_i-\omega_{i-1})}\left(f_{i,k} - f_{i-1,k}\right)$$

$$i = 2, \cdots\cdots M$$

where

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

F : force vector

f : frequency

ω : speed of rotation

k : index of harmonics

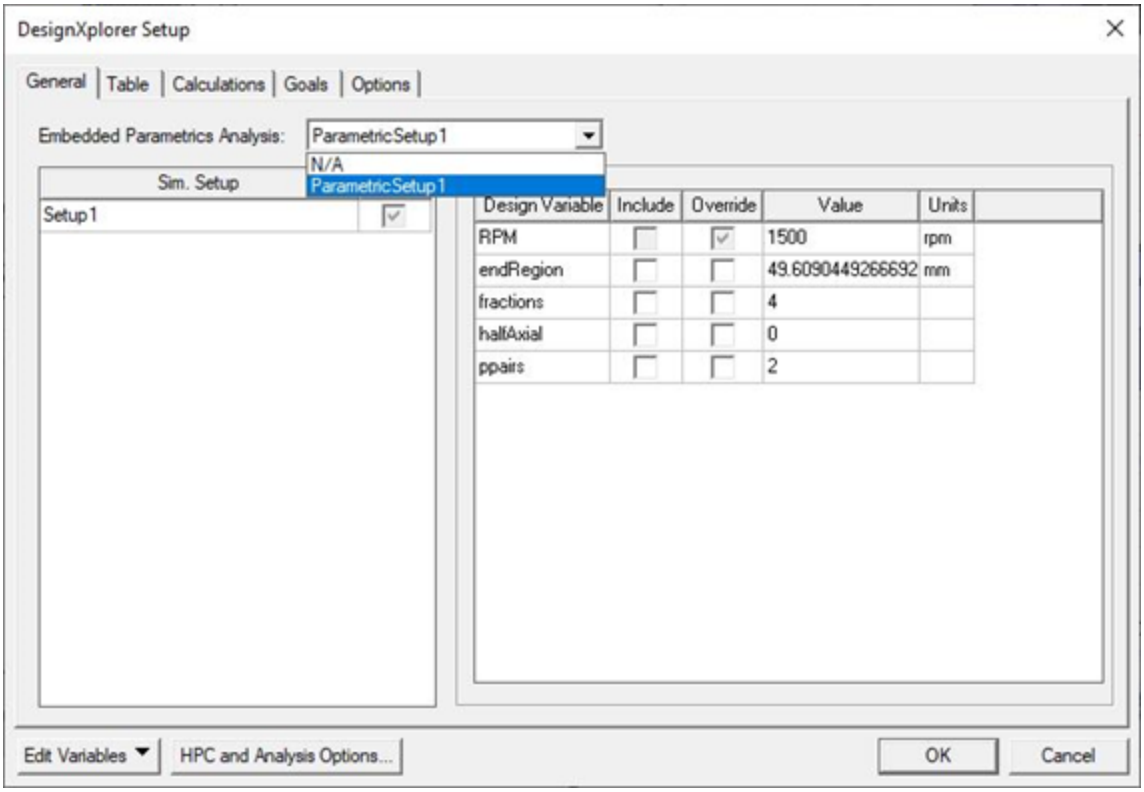
i : index of solved speed points

M : total number of solved speed points

Note: To use this feature you must open Maxwell from within Workbench. For detailed information on setting up and using harmonic force coupling in Workbench, refer to the Workbench help topic: *Importing Data into a Harmonic Analysis*.

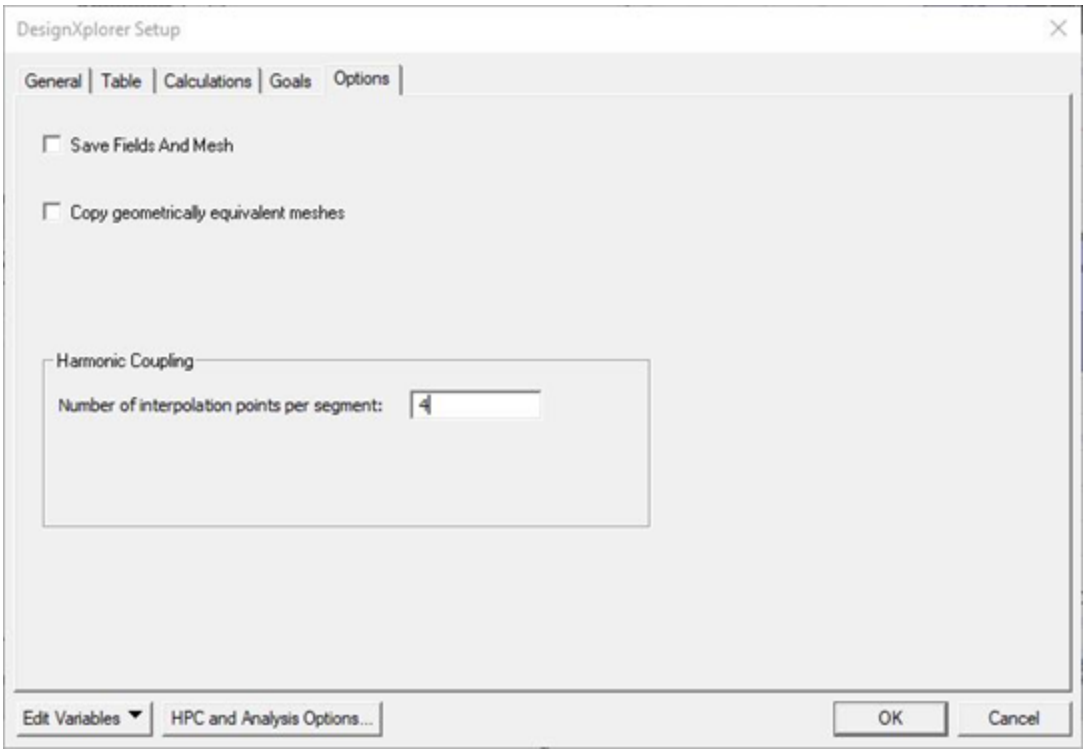
To set up the number of interpolation points per segment:

1. Open the **DesignXplorer Setup** panel and set Parametric Sweep (RPM sweep) as the **Embedded Parametric Analysis**.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

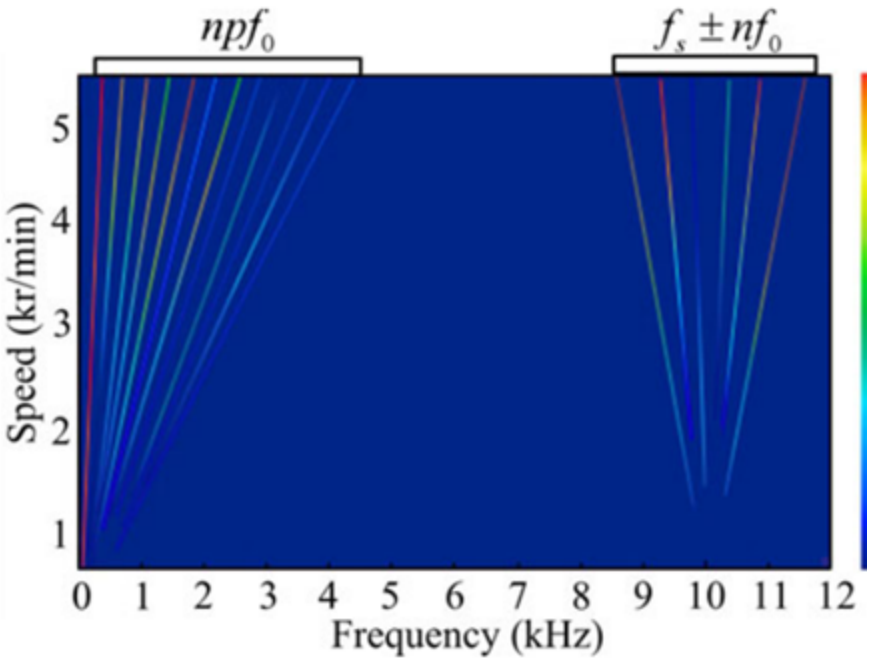
2. On the **Options** tab, enter the **Number of interpolation points per segment**. (This value must be a non-negative integer.)



For Inverter-Fed Motors

The interpolation scheme described for electrical machines is valid for harmonics related to mechanical rotation speed of electric machines. When the electric machine is fed from an inverter with a fixed switching frequency, the high-frequency harmonics due to the switching frequency have to be interpolated differently. Below is a typical waterfall phase current for inverter-fed electrical machine:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The diagram shows that there are two types of current harmonics: One is the current order harmonics npf_0. The other is the switching-frequency sideband harmonics $f_s \pm nf_0$. The harmonic force resulting from the speed-related part can be interpolated with RPM by using algorithms described for electrical machines. The current harmonics related to switching frequency will generate harmonic force components at frequencies around switching-frequency and its multiples:

$$f = mf_s + nf_0$$

Where f_s is switching frequency, f_0 is the mechanical rotating frequency, m and n are integers. In the Maxwell solver, m is calculated by:

$$m = \frac{f_{max}}{f_s}$$

Here f_{max} is the input maximum frequency by user, and n is the number of harmonic distortions.

The harmonic force interpolation scheme for inverted-fed machines is similar to the one described for electrical machines:

1. The speed-frequency lines are constructed at and around switching-frequency and its multiples based on the calculated harmonic force data at specified RPM sweep.
2. Linear interpolation takes place along speed-frequency lines.

In order to distinguish different zones of harmonic force interpolation, the harmonics in the speed-related region is set to a maximum of 50.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: The interpolation for electrical machines is order-based, related to mechanical speed. The interpolation for inverted-fed machines is related to inverter switching. Interpolation results from switching frequency will overwrite the results obtained by order-based interpolation because the order-based interpolation is done first.

Maxwell Harmonic Force Calculations and Coupling

- For 2D/3D *transient* and *AC magnetic* designs Maxwell supports *object-based* one-way harmonic force coupling via Ansys Workbench. Maxwell enables you to select multiple objects for force calculations for harmonic coupling with Workbench projects.
- For 2D/3D *AC magnetic* designs, Maxwell provides *element-based* solutions for one-way harmonic force coupling via Ansys Workbench. For AC magnetic element-based solutions, there is no need to select objects in Maxwell, because the harmonic surface force is a default output in the solver.
- For 2D/3D *transient* designs, Maxwell provides *element-based* solutions for one-way harmonic force coupling via Ansys Workbench. Element-based harmonic force calculations can also be enabled for stand-alone transient designs.
- Maxwell supports one-way coupling with various third-party software for transient designs.
- Maxwell supports one-way coupling with Ansys motion for transient designs.

Maxwell to Ansys Object-Based Harmonic Force Coupling

Maxwell 2D/3D supports object-based one-way harmonic force coupling for *transient* and *AC magnetic* designs via the Workbench schematic. Maxwell enables the user to select multiple objects for force calculations for harmonic coupling with Workbench. You can also setup multiple RPM Harmonic Force Coupling.

Note: For object-based harmonic force calculation, a full-axial model is required.

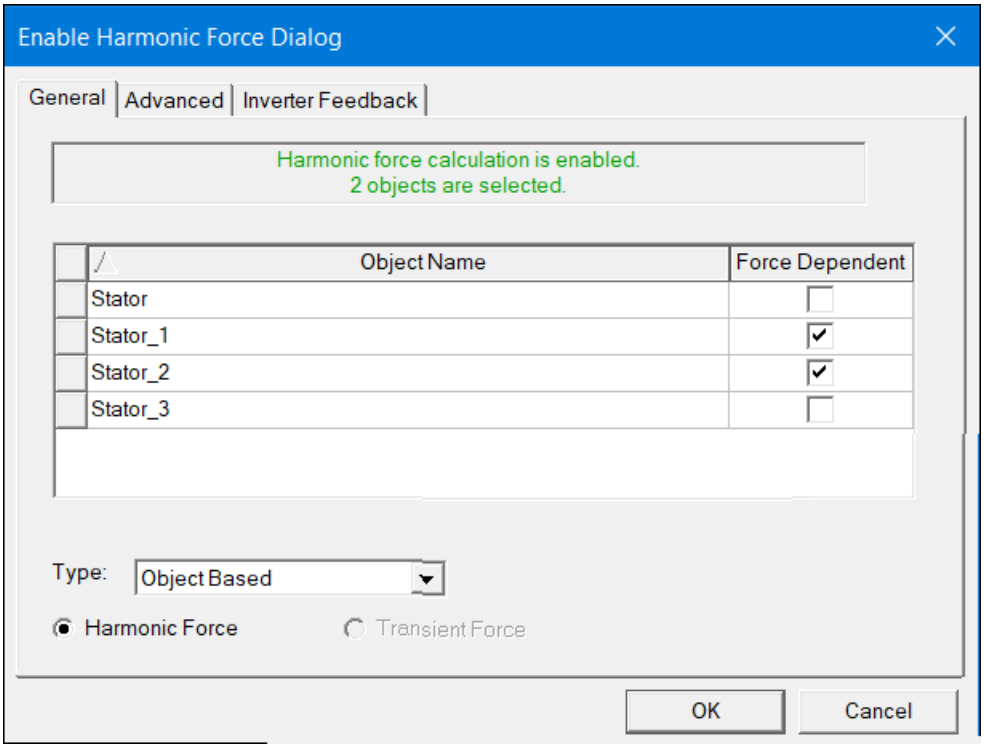
For detailed information on setting up and using harmonic force coupling in Workbench, refer to the Workbench help topics: *Importing Data into a Harmonic Analysis* and *Importing Data into a Thermal or Structural (Static or Transient) Analysis*.

To enable object-based harmonic force calculations in Maxwell:

1. Either select **Maxwell 2D** (or **Maxwell 3D**) > **Enable Harmonic Force Calculation**, or right-click on the design in the Project Manager tree and select **Enable Harmonic Force Calculation** to open the **Enable Harmonic Force Dialog** window. The window has two tabs: **General** and **Advanced**.
2. Do one of the following:

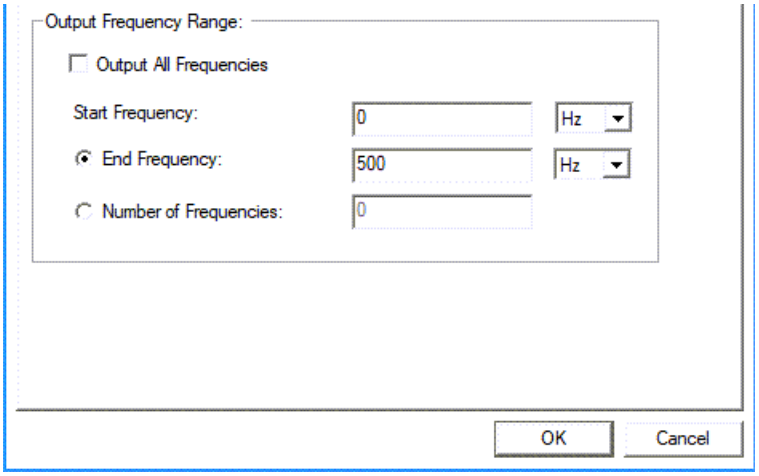
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For *AC magnetic* designs, on the **General** tab, check the **Force Dependent** objects for which force calculations are desired.
- For *transient* designs, on the **General** tab, select **Object Based** as the harmonic force **Type**. Select the **Harmonic Force** radio button. Then check the **Force Dependent** objects for which force calculations are desired.

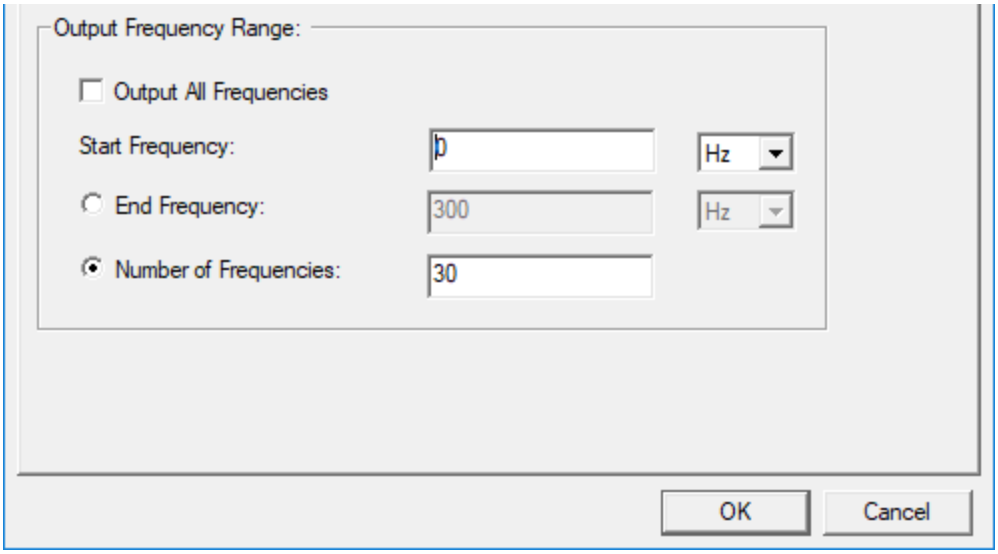


3. The **Advanced** tab provides additional settings to allow you to control collection of data from the force calculations.
   - For *2D and 3D transient* designs (either with rotational motion or with non-rotational motion), the **Output Frequency Range** panel allows you to specify the frequency range of calculated harmonic force components. The default is to **Output All Frequencies** that are calculated. There are two options to set the range of output frequencies:
     - **Use Range** – specify the **Start Frequency** and **End Frequency**. Parameterization of **Start Frequency** and **End Frequency** is supported.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Use Number** – specify the **Start Frequency** and the **Number of Frequencies**; the latter defines the harmonic content where an upper frequency isn't necessarily the limit of interest. The Number of Frequencies is based on multiples of the fundamental frequency, which is defined by the duration of the transient dataset created by the user during pushback. Note that parameterization of **Number of Frequencies** is supported.



- For *transient* designs that include rotational motion:

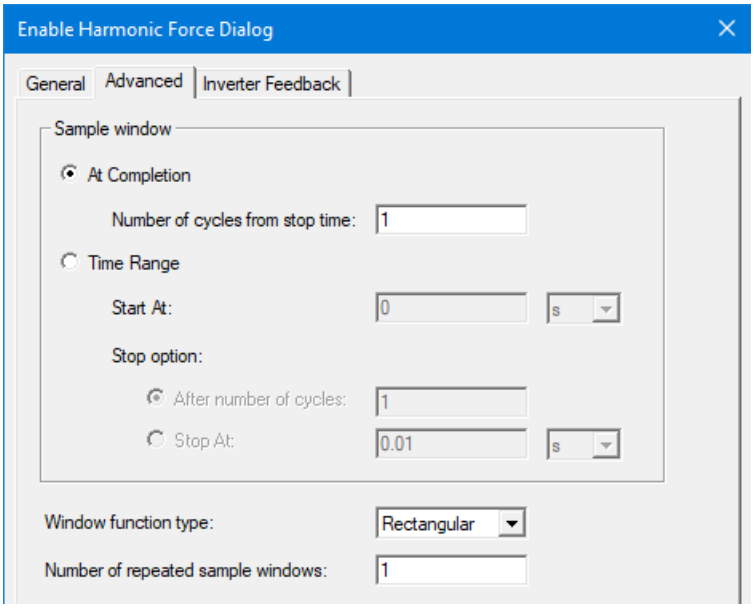
> **Note:** When using Partial Simulation for Full Rotational Machine Models, select *all* teeth tips as objects for harmonic force calculation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **Sample Window**'s collect samples **At completion** setting allows you to choose the **Number of cycles from stop time** for which data is collected. For example, if one cycle equals 1 second, and you solve for 3.5 seconds, setting the **Number of cycles from stop time** to 2 results in data being collected for the period from 1.5 seconds to 3.5 seconds. Alternatively, user can specify a **Time Range** to collect data, for which user can set a data collection **Start At** and a data collection **Stop option** – either stop data collection after a user-specified number of cycles (**After number of cycles**) or at a user-specified **Stop At** time.
- The **Window function type** options setting is used for Discrete Fourier Transform (DFT) of harmonic forces. Refer to Window Function Options for DFT of Harmonic Forces for details on this setting.
- In the frequency domain of harmonic force, the base frequency is determined by the range of time window of data collection ($f=1/\Delta T$). In order to get lower base frequencies, one must increase the time window of data collection, which means increasing the simulation time. It is waste of resources to do so when the solution reaches steady state/periodic in a transient simulation. Instead, you can set the **Number of repeated sample windows** to expand the range of time window of data collection without increasing simulation time by repeating the collected data *n* times before doing DFT. By applying this option, you can get a lower base frequency and higher frequency resolution, but the magnitudes of force components are close to zero at the introduced frequencies.

> **Note:** The **Number of repeated sample windows** setting is enabled only for the **Rectangular** (the default) **Window function type**.

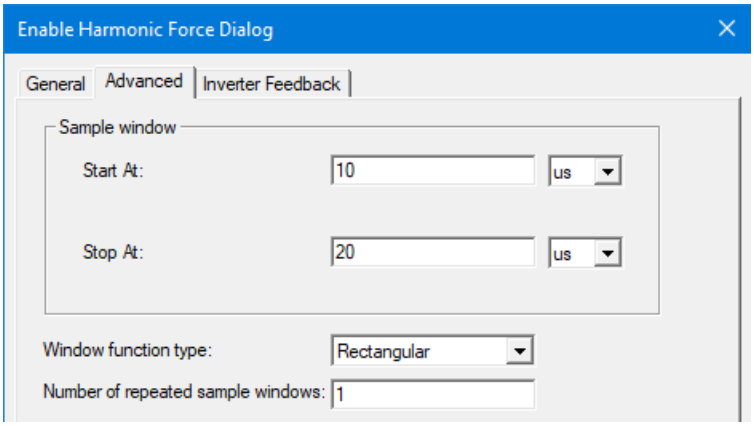
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



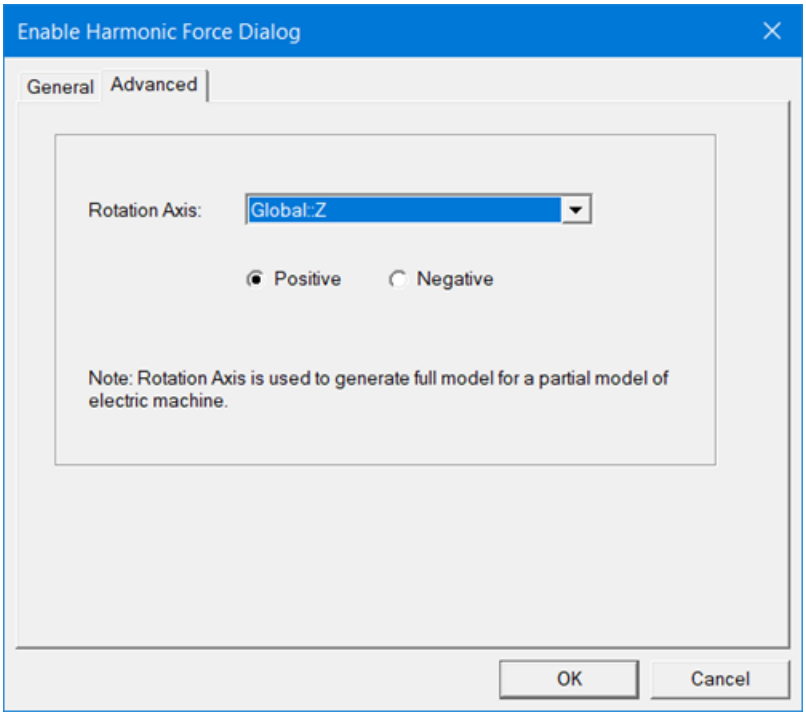
- For *transient* non-rotational motion designs:
  - The **Sample Window** of the **Advanced** tab allows you to set the data collection **Start At** and **Stop At** times.
  - The **Window function type** options setting is used for Discrete Fourier Transform (DFT) of harmonic forces. Refer to Window Function Options for DFT of Harmonic Forces for details on this setting.
  - You can set the **Number of repeated sample windows** to expand the range of time window of data collection.

    > **Note:** The **Number of repeated sample windows** setting is enabled only for the **Rectangular** (the default) **Window function type**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- For *AC magnetic* designs, the **Advanced** tab allows you to set the Rotation Axis and the axis direction so that the Maxwell solver can generate a full model if a partial model is used in the design. You can create relative coordinate systems to add more axes.



Harmonic Eddy force solutions are kept as part of Maxwell design solutions. If Workbench harmonic design settings are changed, the Maxwell solutions will be invalidated.

4. Click **OK** to close the window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Inverter Feedback Tab

See Inverter Feedback for Harmonic Force Coupling for information on this tab.

Force Densities in Maxwell

Let $T_{\alpha\beta}$ be the Maxwell stress tensor (refer to reference [1] for detail of the Maxwell stress tensor). Then the force acting on an object can be calculated by

$$F_\alpha = \sum_\beta \int_V \frac{\partial}{\partial x_\beta} T_{\alpha\beta} d^3x \qquad (1)$$

Application of the divergence theorem to volume integral gives

$$F_\alpha = \oint_S \sum_\beta T_{\alpha\beta}\eta_\beta ds \qquad (2)$$

Note that $\sum_\beta T_{\alpha\beta}n_\beta$ is the force per unit area transmitted across the surface. Namely, it is the surface force density, which is denoted as

$$f_\alpha^S = \sum_\beta T_{\alpha\beta}\eta_\beta \qquad (3)$$

Also, from (1), you can define a volume force density by stress tensor as

$$f_\alpha^v = \sum_\beta \frac{\partial}{\partial x_\beta} T_{\alpha\beta} \qquad (4)$$

For a magnetostatic field (similar forms for electric field), the explicit form of the Maxwell stress tensor is

$$T_{\alpha\beta} = H_\alpha B_\beta - \frac{1}{2}\vec{B} \bullet \vec{H}\delta_{\alpha\beta} \qquad (5)$$

So the surface force density can be calculated as

$$f_\alpha^S = \left(\vec{B} \bullet \vec{n}\right)\vec{H} - \tfrac{1}{2}\vec{B} \bullet \vec{H}\vec{n} \qquad (6)$$

The volume force density can be calculated by (4) and using the virtual work principle (refer to reference [2] for details).

Reference [1] J.D. Jackson, *Classical Electrodynamics*, 3rd edition, Wiley, 1999.

Reference [2] A. Bossavit: "Eddy-currents and forces in deformable conductors", in *Mechanical Modelings of New Electromagnetic Materials* (Proc. IUTAM Symp., Stockholm, April 1990, R.K.T. Hsieh, ed.), Elsevier (Amsterdam), 1990, pp. 235-42.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Force Density Calculation in Maxwell

Maxwell solvers can be coupled with Ansys Mechanical either one-way or two-way. Maxwell solvers provide volumetric force density/surface force density as load to the mechanical solvers. Force density is calculated by Maxwell stress tensor.

Maxwell stress tensor for magnetic fields:

$$\overline{S} = \frac{1}{\mu_0}\begin{bmatrix} B_x^2 - B^2/2 & B_x B_y & B_x B_z \\ B_x B_y & B_y^2 - B^2/2 & B_y B_z \\ B_x B_z & B_y B_z & B_z^2 - B^2/2 \end{bmatrix}$$

where

$$B^2 = B_x^2 + B_y^2 + B_z^2$$

The force on a surface can be calculated as

$$\vec{F} = \frac{1}{\mu_0}\iint dA\overline{S}\cdot n$$

where *n* is outward normal direction of the face.

The surface force density is calculated from

$$\vec{f} = \frac{\vec{F}}{A}$$

where ***A*** is the area of the surface.

The volumetric force density in an element is calculated from

$$\vec{f} = \frac{1}{V}\sum_{i=0}^{N}\overrightarrow{F_i}$$

where ***V*** is volume of the element, and ***N*** is the number of surfaces (sides in 2D) of this element.

Harmonic Force Calculation from Maxwell

Maxwell can calculate the harmonic force on each tooth of a rotating machine. This feature enables the one-way coupling between Maxwell and Harmonic Response of Ansys. The force is object based, not element based. The harmonic momentum reference to the geometric center of the object is also provided at the same time. This feature is also extended to any individual object.

There are two options to calculate the force of an individual tooth:

- Split the stator/rotor to form separate tooth object, and integrate forces on all out surfaces of this tooth except the splitting surfaces.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Draw an arc (2D) or a cylindrical face (3D) near the tooth in the air gap, and integrate forces on this face.

Discrete Fourier Transform (NDFT) with Non-Uniform Time Interval

In the general case, the definition of the Non-uniform Discrete Fourier Transform (NDFT) is the same as the definition with constant time interval, taking into consideration that the samples can be taken at irregular intervals in time (t_n). That is to say that the samples x(k) of the irregular Fourier transform are taken at multiples of a quantity $\Delta\omega$, which is a fixed quantity in the Fourier domain. The fixed quantity, $\Delta\omega$, in the regular case corresponds to 2π/T. The extension from regular to irregular sampling, therefore, depends on the duration of the signal *x(t)* and not on the fact that the samples are taken at regular or irregular intervals.

A finite sample measured at ***N*** points in a period of T:

$$x(k) = \begin{cases} 0, & k < 0 \\ F(t_k), & 0 \le k \le (N-1) \\ 0, & k \ge N \end{cases}$$

The definition of the non-uniform discrete Fourier transform (NDFT)is as follows:

$$X(k) = \frac{1}{N} \sum_{n=0}^{N-1} x\left(n\right) e^{-j\left(t_n - t_0\right) k \frac{2\pi}{T}}$$

The inverse NDFT is given by

$$x(t_n) = \sum_{k=0}^{N-1} X(k)\, e^{j2\pi \frac{k(t_n - t_0)}{T}}$$

Data Collection

The solver records the components of force and momentum at all teeth at every timestep:

$$\begin{array}{lllll} F_x : F_x(0), & F_x(\Delta t), & F_x(2\Delta t), & \cdots\cdots, & F_x(K\Delta t) \\ F_y : F_y(0), & F_y(\Delta t), & F_y(2\Delta t), & \cdots\cdots, & F_y(K\Delta t) \\ F_z : F_z(0), & F_z(\Delta t), & F_z(2\Delta t), & \cdots\cdots, & F_z(K\Delta t) \\ M_x : M_x(0), & M_x(\Delta t), & M_x(2\Delta t), & \cdots\cdots, & M_x(K\Delta t) \\ M_y : M_y(0), & M_y(\Delta t), & M_y(2\Delta t), & \cdots\cdots, & M_y(K\Delta t) \\ M_z : M_z(0), & M_z(\Delta t), & M_z(2\Delta t), & \cdots\cdots, & M_z(K\Delta t) \end{array}$$

If $t_{end} - t \le \frac{2\pi}{period * rotationSpeed}$, then the solver keeps recording.

Generate Harmonic Force from Partial Model to Full Model

With symmetry-multiplier (period in 3D) > 1, Maxwell solvers automatically generate harmonic force for the full model from the partial model $r^* = Ar$.

where r is the vector to be rotated, r^* is the vector after rotation. The rotation matrix in 3D:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$= \begin{bmatrix} \cos\theta + u_x^2(1-\cos\theta) & u_x u_y(1-\cos\theta) + u_z\sin\theta & u_x u_z(1-\cos\theta) + u_y\sin\theta \\ u_x u_y(1-\cos\theta) + u_z\sin\theta & \cos\theta + u_y^2(1-\cos\theta) & u_y u_z(1-\cos\theta) + u_x\sin\theta \\ u_x u_z(1-\cos\theta) + u_y\sin\theta & u_y u_z(1-\cos\theta) + u_x\sin\theta & \cos\theta + u_z^2(1-\cos\theta) \end{bmatrix}$$

The rotation matrix in 2D:

$$A = \begin{bmatrix} \cos\theta & -\sin\theta \\ \sin\theta & \cos\theta \end{bmatrix}$$

Vectors that need to be rotated: position (**r**), force (**F**) and moment (**M**)

Spatial and Temporal 2D Discrete Fourier Transform (DFT)

Electromagnetic force components on teeth inside an electric machine are spatial (in circumferential direction) and temporal (transient). The magnetic force density acting on the stator along the air gap is dependent on time and on space. This spatial and temporal force distribution can be expanded by means of Fourier analysis to calculate its time and space harmonic waves. This analysis can be used to improve NVH (noise, vibration, and harshness) performance.

The magnetic force density acting on the stator along the air gap is dependent on time and on space. This spatial and temporal force distribution can be expanded by means of Fourier analysis to calculate its time and space harmonic waves.

The force components on teeth can be expressed as function f(x, t), where x represents spatial, and t denotes temporal. The function f (x, t) is decomposed to a linear combination of harmonic (sines and cosines, more generally orthogonal) functions.

Direct Transformation

$$F(u,v) = \frac{1}{MN}\sum_{m=0}^{M-1}\sum_{n=0}^{N-1} f(m,n) exp\left[-2\pi i\left(\frac{mu}{M} + \frac{nv}{N}\right)\right]$$

where:

u = 0, 1, …… M-1, wave number in spatial,

v = 0, 1, ……. N-1, wave number in temporal.

Window Function Options for DFT of Harmonic Forces

Window function options for Discrete Fourier Transform (DFT) of harmonic forces are available for Maxwell 2D and 3D transient solvers and support both rotational and non-rotational models.

The Discrete Fourier Transform (DFT) operates on a finite-length time sequence to compute its spectrum. For a continuous signal like a sine wave, you need to capture a segment of the signal in order to perform the DFT. Usually, you also need to apply a window function to the captured signal before taking the DFT. A window function (also known as an apodization function or tapering function) is a mathematical function that is zero-valued outside of some chosen interval,

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

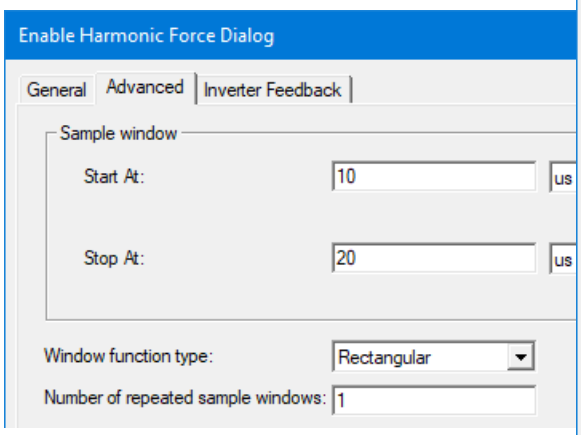
normally symmetric around the middle of the interval, usually near a maximum in the middle, and usually tapering away from the middle. The mismatch in amplitude between the two ends of the signal distorts the spectrum, a phenomenon called spectral leakage. Spectral leakage can be caused by discontinuities in the non-integer number of periods in sampled signals that usually happen in asynchronous machines. This can be improved using windowing. Maxwell provides several window functions for the DFT of harmonic forces. These include

- Rectangular (default)
- Triangular
- Van Hann
- Hamming
- Blackman
- Lanczos
- Welch

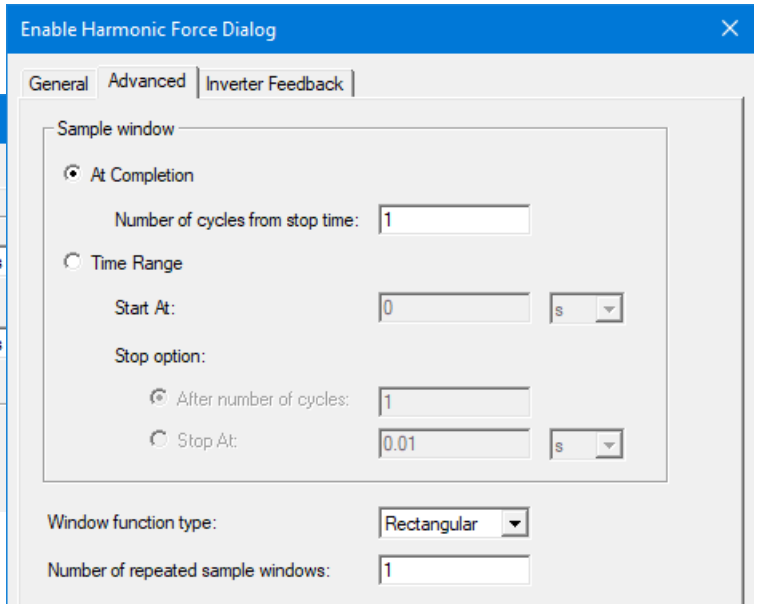
Refer to FFT Window Functions for detailed information on each of these window function types.

To select the various window function options:

1. Either select **Maxwell 2D** (or **Maxwell 3D**) > **Enable Harmonic Force Calculation**, or right-click on the design in the Project Manager tree and select **Enable Harmonic Force Calculation** to open the **Enable Harmonic Force Dialog** box.
2. Specify the **Window function type** using the drop-down menu on the **Advanced** tab.



Non-Rotational Setup



Rotational Setup

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

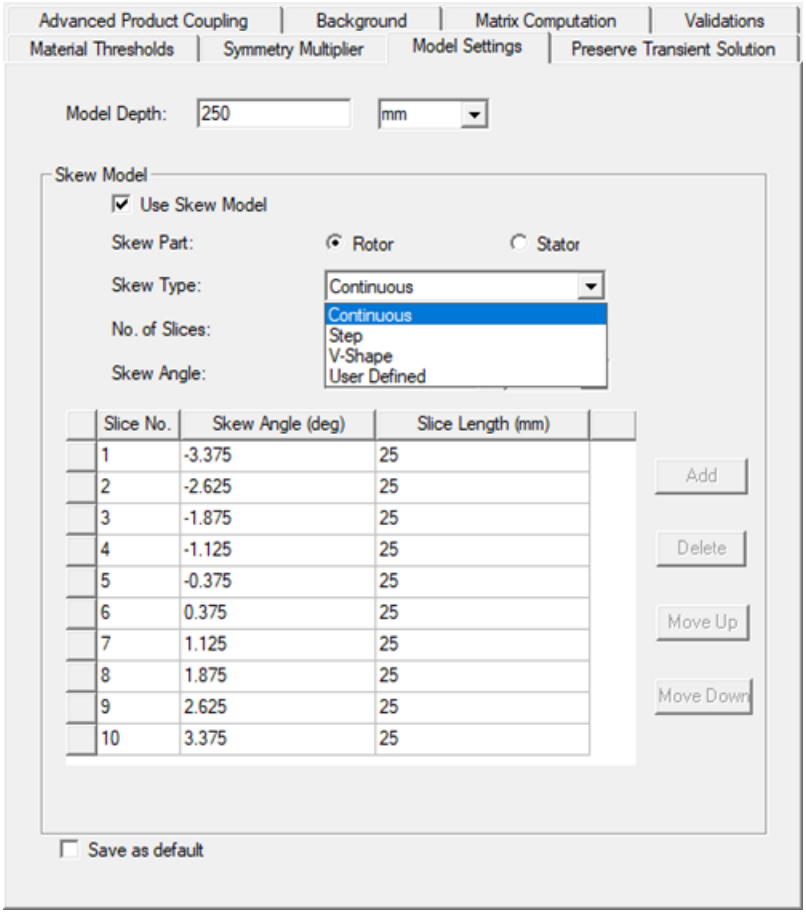
Harmonic Force Calculation in Maxwell 2D with Skew Model

Maxwell 2D supports object-based harmonic force calculation using a multi-slice skew model along the axial direction of rotating machines. (Element-based harmonic force calculation is not applicable to 2D multi-slice skew model.)The multi-slice skew model can improve the accuracy of Maxwell 2D simulation when the slot/teeth on the stator/rotor are skewed. By using a skew model, 2D vibration modeling in Ansys Mechanical is no longer valid; thus, 3D simulation must be performed instead by applying object-based harmonic force calculation slice-by-slice.

Skew Model Setup

> **Note:** For detailed information on Skew Model settings, refer to the Model Settings Tab topic.

Skew model setup in Maxwell 2D is defined on the **Model Settings** tab of the **2D Design Settings** window as described in the Model Settings Tab topic.



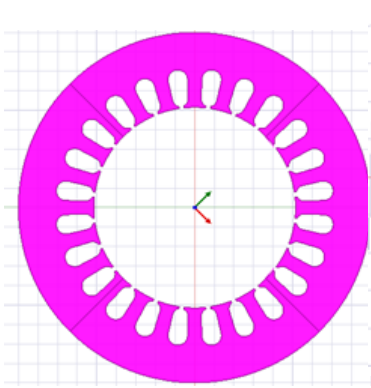
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To create 3D geometry, the center position of each slice must be set properly:

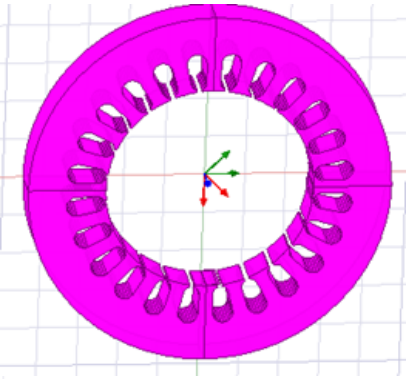
1. Create a 2D full model in Maxwell (do not unite half teeth at the M-S boundary), and generate 3D geometry of one slice; then create a Maxwell 3D design, and duplicate one slice in the Maxwell 3D.
2. Move the center of the 3D geometry to the origin of the global coordinate system; then set the Z axis as the rotating axis.
3. Rotate each slice to its individual circumferential position: slice angle is taken from the skew model settings (see figure above). The slice number indices are arranged from least to greatest, corresponding to Z coordinates of slice position (from least to greatest).

   For object-based harmonic force calculation, only one group of harmonic force components is generated for each selected object when skew model in not enabled in Maxwell 2D transient solver; and the Z coordinate of the object is set to zero by default. The “zero” Z position is used to match the selected geometry in Mechanical if coupled with the Harmonic response 3D solver.

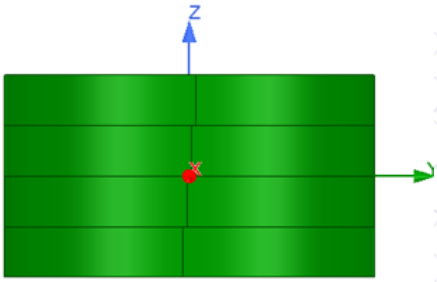
Maxwell 2D Partial Model



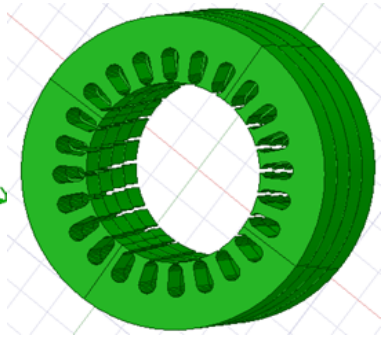
2D full model (stator only)



3D geometry of stator (one slice)



3D geometry after moving and rotating



3D geometry in harmonic response solver

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Harmonic Force Calculation Requirements for Maxwell 2D Slice Model

Because the harmonic force calculated from the 2D slice model will be applied to a 3D model in Ansys Mechanical, it is very important to have a rule to link the skew angle to the slice number, and the slice number to the z-position in the Maxwell 2D solver. The relationship between slice number and z-position will determine how the 3D model will be generated.

The general rule is as follows:

- The sequence of slice number: from least to greatest with z position (this is for customer, not in the solver)
- The z-position: from least to greatest
- The skew angle: one-to-one with slice number, from slice 1 to slice N

The user generates 3D model based on skew angle, slice by slice, from least z-position to greatest z-position.

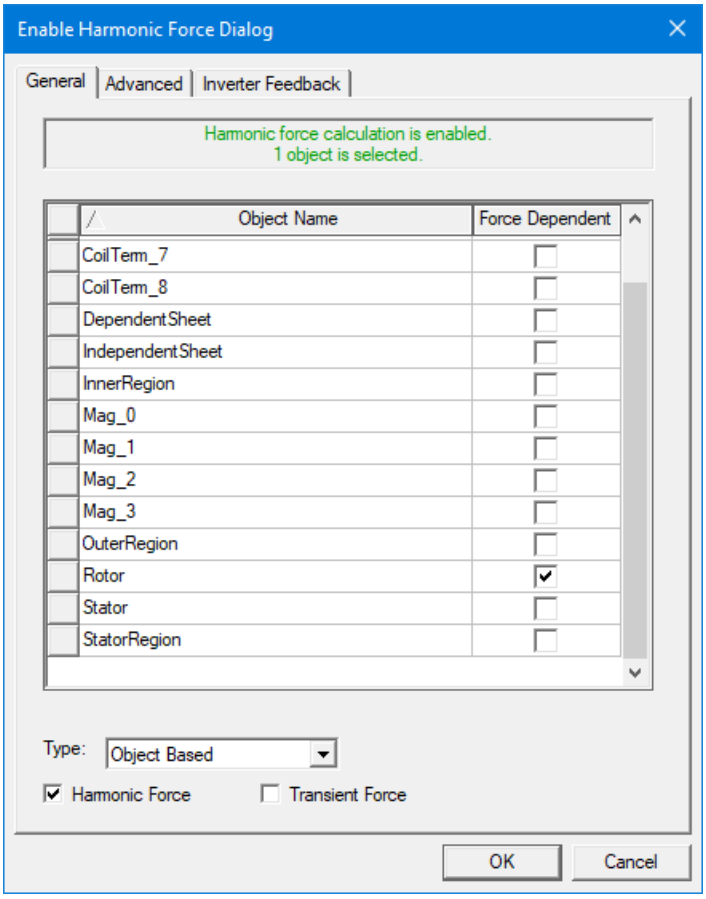
Transient Report of Object-Based Harmonic Force Components

When object-based harmonic force is activated, the transient force data of selected harmonic force objects are available for plot in Maxwell transient solvers. The force component data is recorded at every timestep, starting from time zero. To create a transient report of object-based harmonic force components:

1. Select **Maxwell 2D** or **3D** > **Enable Harmonic Force Calculation**.
2. In the **Enable Harmonic Force Calculation** dialog box, on the **General** tab, select **Object Based** as the harmonic force **Type**. Then check the **Force Dependent** objects for

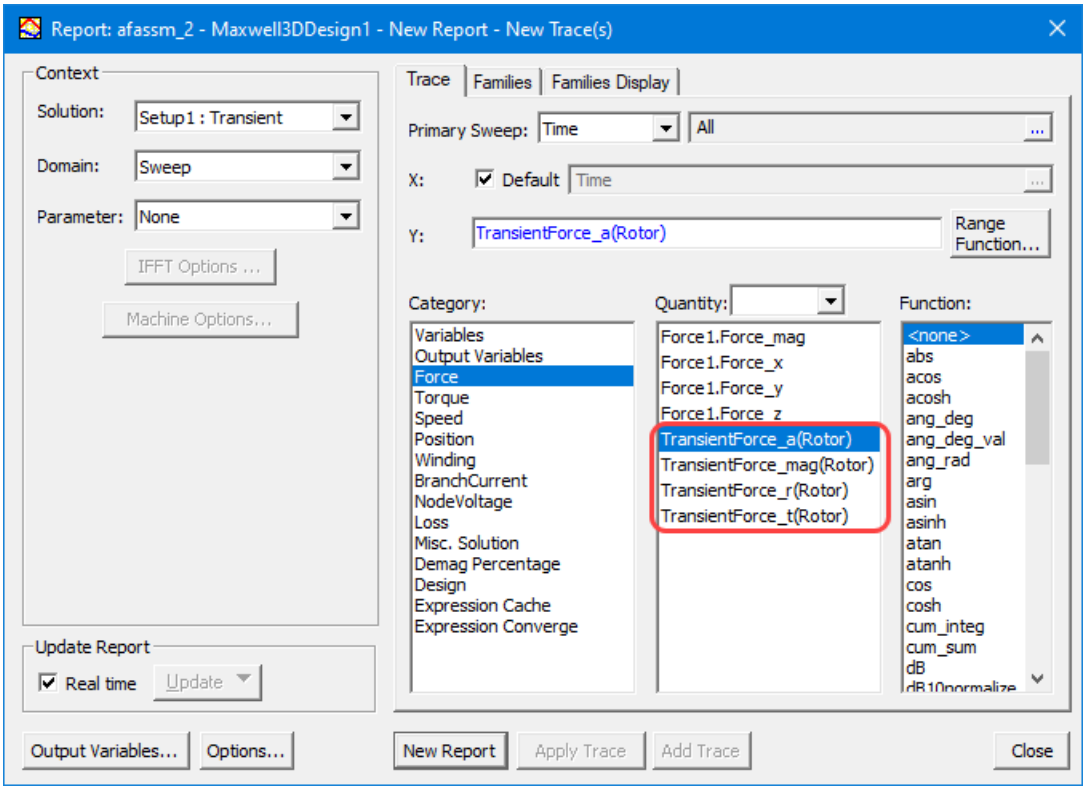
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

which force calculations are desired.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

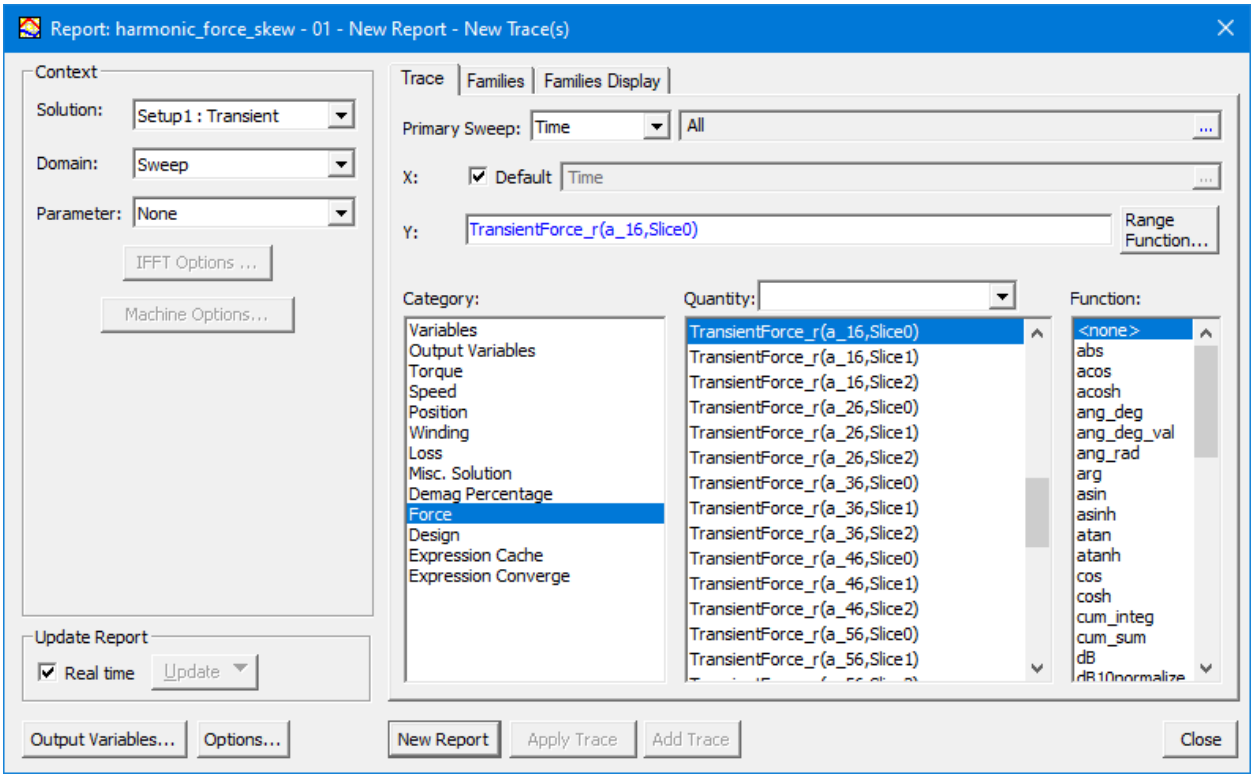
3. Create a **Transient Report** under **Results** in the Project Manager tree.



4. Select **Force** as the **Category**, and then select the harmonic force components you want to plot:
    - For 2D (XY) designs: **TransientForce_mag**, **TransientForce_r**, **TransientForce_t**.
    - For 3D designs: **TransientForce_mag**, **TransientForce_a**, **TransientForce_r**, **TransientForce_t**.
    - For axial flux, it is **transientForce_r**, **transientForce_t** and **transientForce_z**.

Transient force reports also support Maxwell 2D with skew slice model. The transient force is calculated on each slice, and there are *n* forces available to be plotted for each selected object.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note: For detailed information on Skew Model settings, refer to the *Model Settings Tab* topic.

Maxwell to Ansys Element-Based Harmonic Force Coupling

Maxwell provides element-based solutions for one-way harmonic force coupling via Ansys Workbench schematic for

- 2D/3D AC Magnetic design element-based harmonic force coupling
- 2D/3D Transient design element-based harmonic force coupling

For detailed information on setting up and using harmonic force coupling in Workbench, refer to the Workbench help topics: *Importing Data into a Harmonic Analysis* and *Importing Data into a Thermal or Structural (Static or Transient) Analysis.*

Element-Based Harmonic Force Calculations and Coupling for Transient Designs

Maxwell 2D/3D supports *element-based* one-way harmonic force coupling for *transient* designs via the Workbench schematic. Maxwell enables the user to select multiple objects for force

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

calculations for harmonic coupling with Workbench.

- For element-based (Surface) harmonic coupling, the force density is calculated at each surface element of the selected objects at every timestep in the specified time window. Then, Discrete Fourier Transform is performed to transfer data from the time domain to the frequency domain. In the WB Harmonic response solver, surface force density is applied as surface pressure to the selected surfaces. There is no restriction on surface selection. Mapping is involved and it is done by the post-processor. Magnetostriction is not included for surface element-based harmonic force.
- For element-based (Volumetric) harmonic coupling, the force density is calculated at each element of the selected objects at every timestep in the specified time window. Then, Discrete Fourier Transform is performed to transfer data from the time domain to the frequency domain. In the WB Harmonic response solver, volumetric force density is applied as body force to the selected objects. There is no restriction on object selection. Mapping is involved and it is done by the postprocessor. Magnetostriction can be included for volumetric element-based harmonic force.

You can also set up multiple RPM Harmonic Force Coupling.

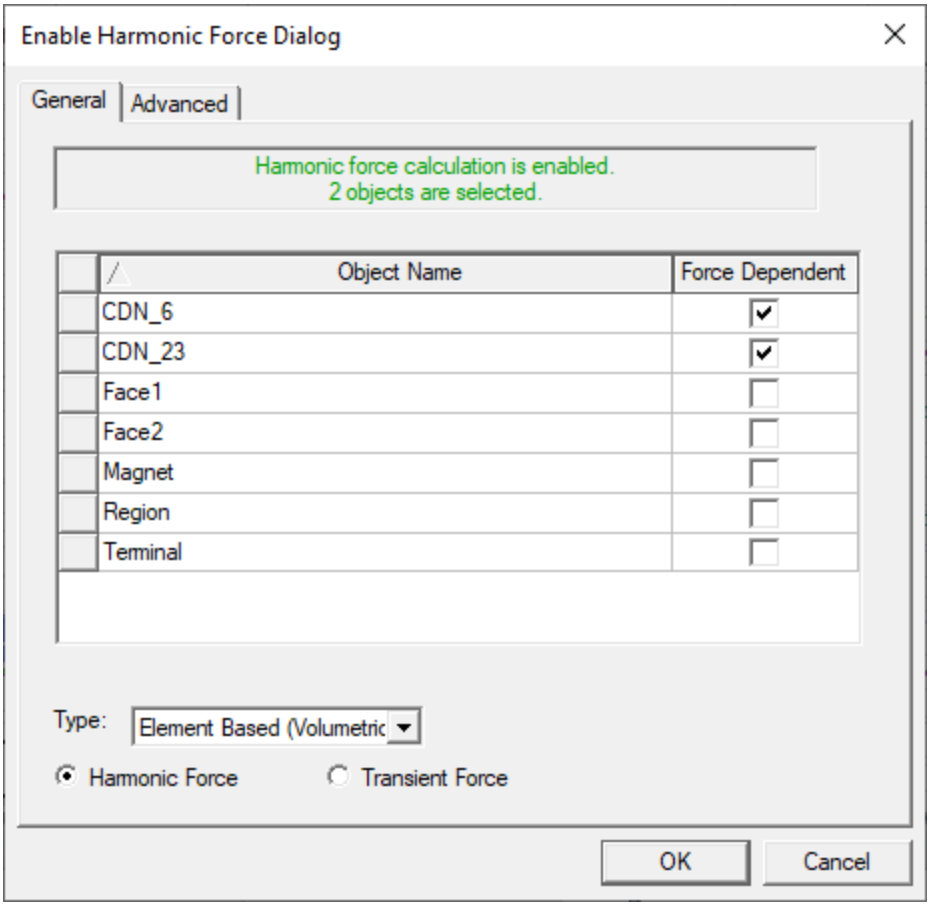
> **Note:** Element-based harmonic force calculations can also be enabled for stand-alone transient designs.

For detailed information on setting up and using harmonic force coupling in Workbench, refer to the Workbench help topics: *Importing Data into a Harmonic Analysis* and *Importing Data into a Thermal or Structural (Static or Transient) Analysis*.

To enable element-based harmonic force calculations for transient designs in Maxwell:

1. Either select **Maxwell 2D** (or **Maxwell 3D**) > **Enable Harmonic Force Calculation**, or right-click on the design in the project tree and select **Enable Harmonic Force Calculation** to open the **Enable Harmonic Force Dialog** window.
    - For *transient* designs, the dialog has two tabs: **General**, and **Advanced**.
2. On the **General** tab, select **Element Based (Surface)** or **Element Based (Volumetric)** as the harmonic force **Type**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Check the **Force Dependent** objects for which force calculations are desired.

3. The **Advanced** tab provides additional settings to allow you to control collection of data from the force calculations.

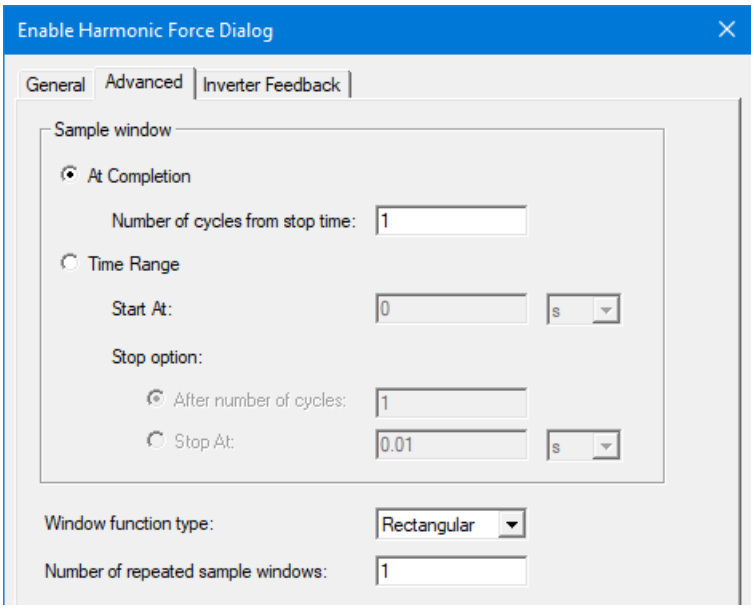
    - For *transient* designs that include rotational motion:
        - The **Sample Window**'s collect samples **At completion** setting allows you to choose the **Number of cycles from stop time** for which data is collected. For example, if one cycle equals 1 second, and you solve for 3.5 seconds, setting the **Number of cycles from stop time** to 2 results in data being collected for the period from 1.5 seconds to 3.5 seconds. Alternatively, user can specify a **Time Range** to collect data, for which user can set a data collection **Start At** and a data collection **Stop option** – either stop data collection after a user-specified number of cycles (**After number of cycles**) or at a user-specified **Stop At** time.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** The term "cycle" refers to the mechanical cycle, equivalent to 60/rpm/(Symmetry Multiplier) [in seconds].

- The **Window function type** options setting is used for Discrete Fourier Transform (DFT) of harmonic forces. See Window Function Options for DFT of Harmonic Forces for details on this setting.
- In the frequency domain of harmonic force, the base frequency is determined by the range of time window of data collection ($f = 1/\Delta T$). In order to get lower base frequencies, one must increase the time window of data collection, which means increasing the simulation time. It is waste of resources to do so when the solution reaches steady state/periodic in a transient simulation. Instead, you can set the **Number of repeated sample windows** to expand the range of time window of data collection without increasing simulation time by repeating the collected data *n* times before doing DFT. By applying this option, you can get a lower base frequency and higher frequency resolution, but the magnitudes of force components are close to zero at the introduced frequencies

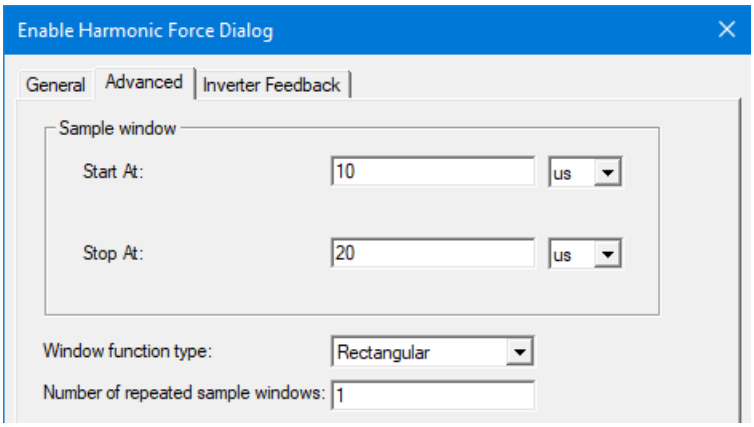
> **Note:** The **Number of repeated sample windows** setting is enabled only for the **Rectangular** (the default) **Window function type**.



- For *transient* non-rotational motion designs:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **Sample Window** of the **Advanced** tab allows you to set the data collection **Start At** and **Stop At** times.

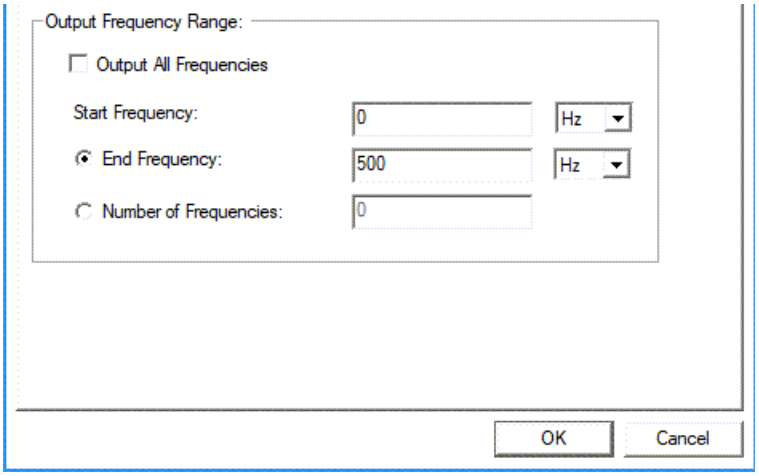


- The **Window function type** options setting is used for Discrete Fourier Transform (DFT) of harmonic forces. See Window Function Options for DFT of Harmonic Forces for details on this setting.
- You can set the **Number of repeated sample windows** to expand the range of time window of data collection.

> **Note:** The **Number of repeated sample windows** setting is enabled only for the **Rectangular** (the default) **Window function type**.

- For *2D and 3D transient* designs (either with rotational motion or with non-rotational motion), the **Output Frequency Range** panel allows you to specify the frequency range of calculated harmonic force components. The default is to **Output All Frequencies** that are calculated. If the **Output All Frequencies** is unchecked, there are two options to set the range of output frequencies:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Specify the **Start Frequency** and **End Frequency**. Parameterization of **Start Frequency** and **End Frequency** is supported.
- Specify the **Start Frequency** and the **Number of Frequencies**; the latter defines the harmonic content where an upper frequency isn't necessarily the limit of interest. The Number of Frequencies is based on multiples of the fundamental frequency, which is defined by the duration of the transient dataset created by the user during pushback. Note that parameterization of **Number of Frequencies** is supported.

4. Click **OK** to close the window.

Once Maxwell is linked to the Workbench harmonic solver, it can export the harmonic force results to the Workbench Harmonic design. Harmonic transient force solutions are kept as part of Maxwell design solutions. If Workbench harmonic design settings are changed, the Maxwell solutions will be invalidated.

Element-Based Harmonic Force Coupling for AC Magnetic Designs

For 2D/3D AC Magnetic designs, Maxwell provides *element-based* solutions for one-way harmonic force coupling via Ansys Workbench. For element-based solutions, there is no need to select objects in Maxwell, because harmonic surface force is a default output in the solver.

For element-based (surface) coupling, there is a harmonic force vector on each surface node for objects. The user must select surfaces on the Workbench Mechanical side before these forces can be mapped. This mapping is done by the Maxwell postprocessor.

For element-based (volumetric) harmonic coupling, the force density is calculated at each element of selected objects. The user must select bodies on the Workbench Mechanical side before these forces can be mapped. This mapping is done by the Maxwell post-processor.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For detailed information on setting up and using harmonic force coupling in Workbench, refer to the Workbench help topics: *Importing Data into a Harmonic Analysis* and *Importing Data into a Thermal or Structural (Static or Transient) Analysis*

Export Transient/Harmonic Force from Maxwell

There are four options to export transient forces and one option to export transient/harmonic force from Maxwell transient solvers:

- **LMS** – provides element-based transient surface node force and mesh to link to Virtual lab, only for stator of electric machine.
- **Romax** – provides object-based transient force for the stator of electric machines and torque information.
- **UNV** – UNV file format is a standard universal file format, and it can be read by most of third-party software. Element-based transient surface/volumetric node force will be outputted with mesh.
- **Ansys Motion** – provides object-based transient force, and element-based volumetric/surface transient force to Ansys-Motion for structure and acoustic simulation.

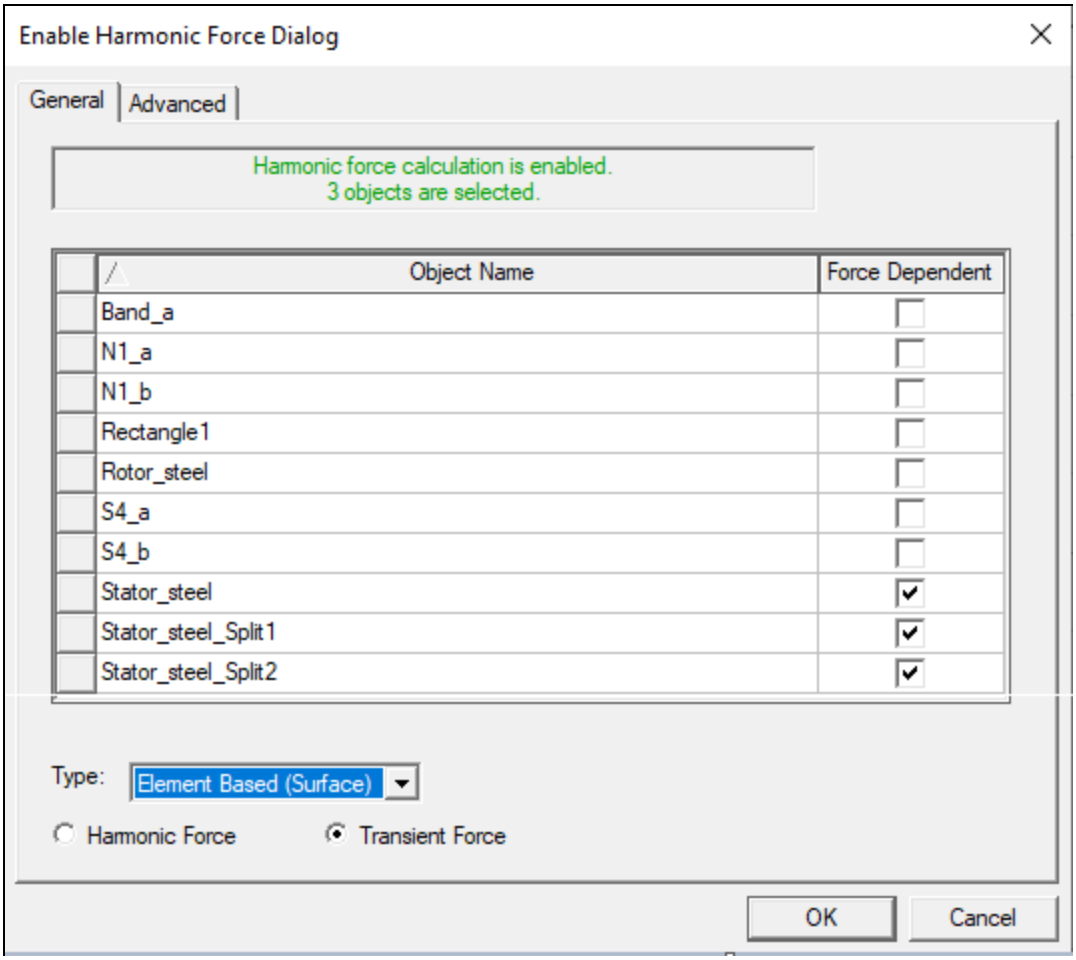
  > **Note:** Ansys Motion does not currently support harmonic force.

- **Ansys Mechanical**
  - For 3D transient solver, if **Element Based (Volumetric)** harmonic force selected, this option provides element-based volumetric harmonic force in `.csv` file format to *Ansys-Harmonic Response* for acoustic simulation.
  - If **Object Based** harmonic force is selected, this option provides object-based harmonic force with RPM parametric sweep to generate waterfall diagram in *Ansys Mechanical*.

All forces and mesh are output with the full model. To export transient/harmonic force from Maxwell, following steps must be done before solving the design:

1. Either select **Maxwell 2D** (or **Maxwell 3D**) > **Enable Harmonic Force Calculation**, or right-click on the design in the Project Manager tree and select **Enable Harmonic Force Calculation** to open the **Enable Harmonic Force Dialog** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. On the **General** tab, select the **Force Dependent** objects for which force calculations are desired. For LMS, select only one object (stator).
3. Select **Type** of force for output. Choices are: **Object Based**, **Element Based (Surface)**, or **Element Based (Volumetric)**.
4. Check **Transient Force** if you want to export transient forces. (**Harmonic Force** is for Workbench Harmonic Response Coupling.)
5. Check **Harmonic Force** if you want to export element-based volumetric harmonic force in .csv file format. This export option is only available for the Maxwell 3D magnetic transient solver.

> **Note:** Both Transient Force and Harmonic Force can be exported; check both options to export them for the same simulation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. The **Advanced** tab provides additional settings to allow you to control collection of data from the force calculations. For more details, see Element-Based Harmonic Force Coupling for Transient Designs.

After solving the design, select **Maxwell 2D** (or **Maxwell 3D**) > **Export Transient/Harmonic Force** to generate transient forces. For details on the export window, see Maxwell One-Way Coupling with Ansys Mechanical.

Maxwell to LMS Virtual.Lab Coupling

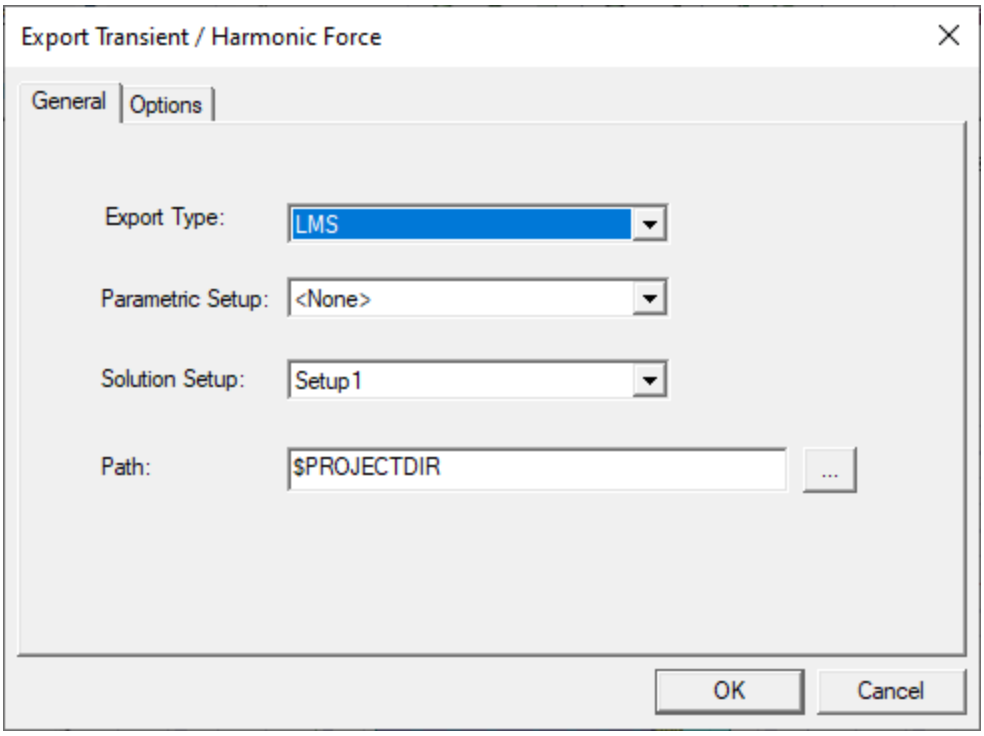
Maxwell supports one-way coupling with Siemens® LMS Virtual.Lab™.

To enable output of Maxwell solutions for use in Virtual.Lab:

1. Follow the steps in **Export Transient/Harmonic Force from Maxwell** to enable transient force output before solving the Maxwell design.
2. After solving, either select **Maxwell 2D** (or **Maxwell 3D**) > **Export Transient/Harmonic Force**, or right-click on the Design in the project tree and select **Export Transient/Harmonic Force** to open the **Export Transient/Harmonic Force Dialog** panel.
3. On the **General** tab:
    - Select **LMS** for the **Export Type**.
    - Specify a **Parametric Setup** if any (If **<None>** is selected, only the nominal results will be exported).
    - Select a **Solution Setup**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Specify the path of the location of output file (the default is the project directory).

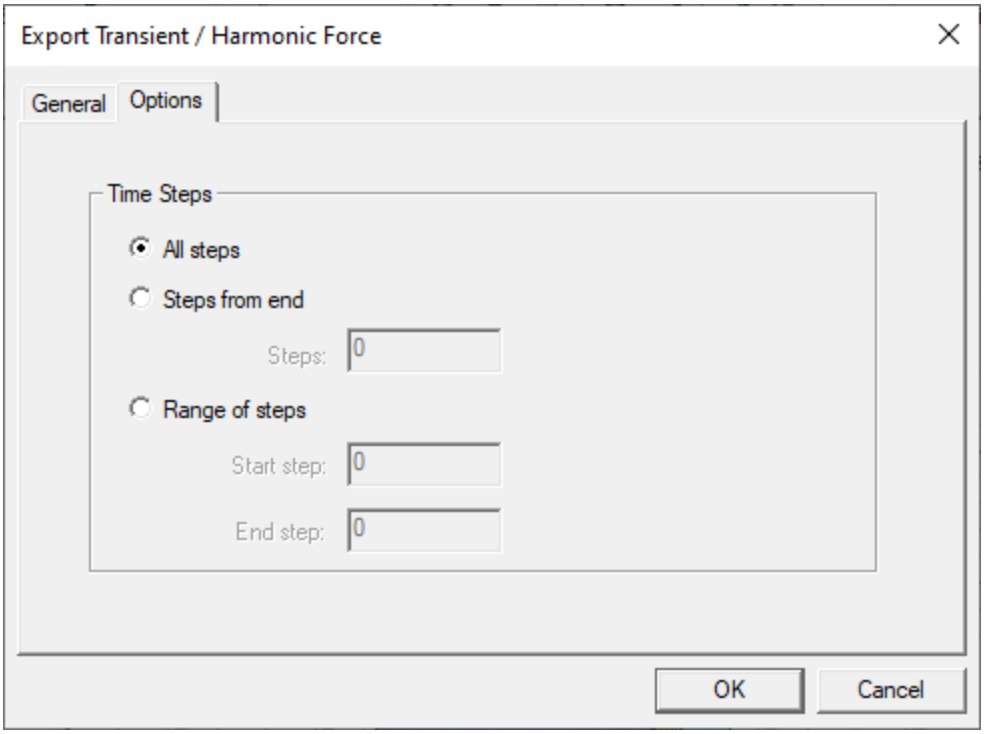


4. On the **Option** tab, settings allow you to control collection of data for export to Virtual Lab. You can choose to include **All steps**. The **Steps from end** setting allows you to choose the number of **Steps** from the end for which data is collected.

   > **Note:** If rotation motion is defined in the project, the input number of steps will be ignored. Instead the last cycle will be the time window.

   Alternatively, you can select a **Range of steps**, for which you can set a data collection **Start step** and a data collection **End step**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Click **OK** to close the window. When you click **OK**, all the results (nominal or parametric solution) will be exported to the specified folder.

Maxwell to Romax Coupling

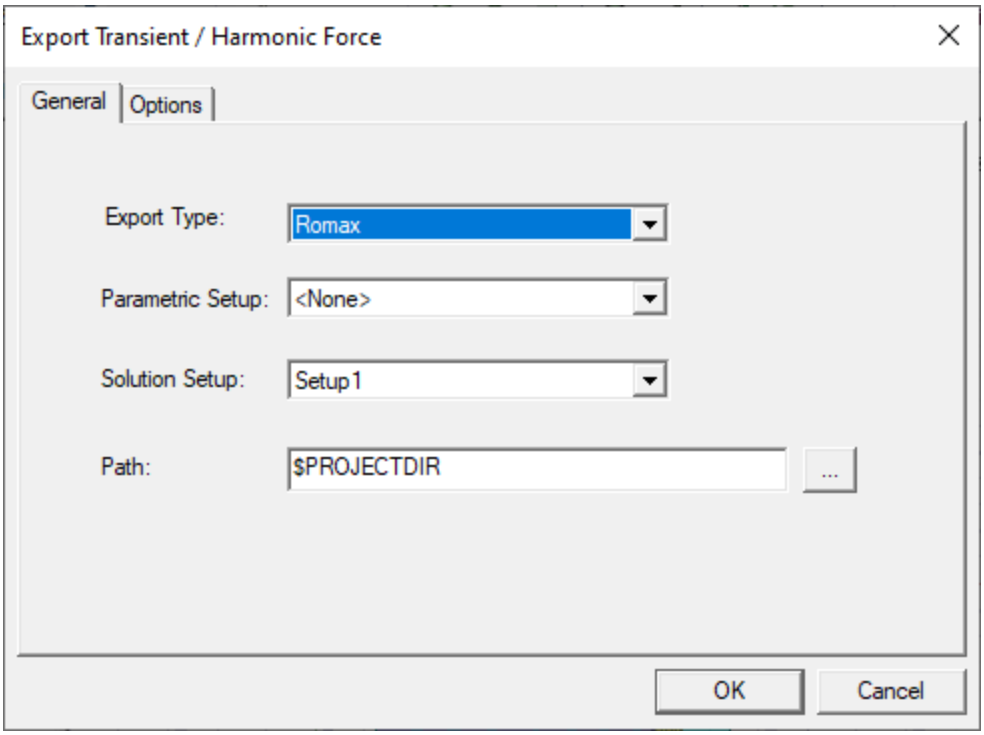
Maxwell provides object-based transient force and torque components on selected objects. Maxwell supports one-way coupling with Romax.

To enable output of Maxwell solutions for use with Romax:

1. Follow the steps in **Export Transient/Harmonic Force from Maxwell** to enable transient force output before solving the Maxwell design.
2. After solving, either select **Maxwell 2D** (or **Maxwell 3D**) > **Export Transient/Harmonic Force**, or right-click on the design in the Project Manager tree and select **Export Transient/Harmonic Force** to open the **Export Transient/Harmonic Force Dialog** panel.
3. On the **General** tab:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select **Romax** for the **Export Type**.
- Specify a **Parametric Setup** if any (If **<None>** is selected, only the nominal results will be exported).
- Select a **Solution Setup**.
- Specify the path of the location of output file (the default is the project directory).

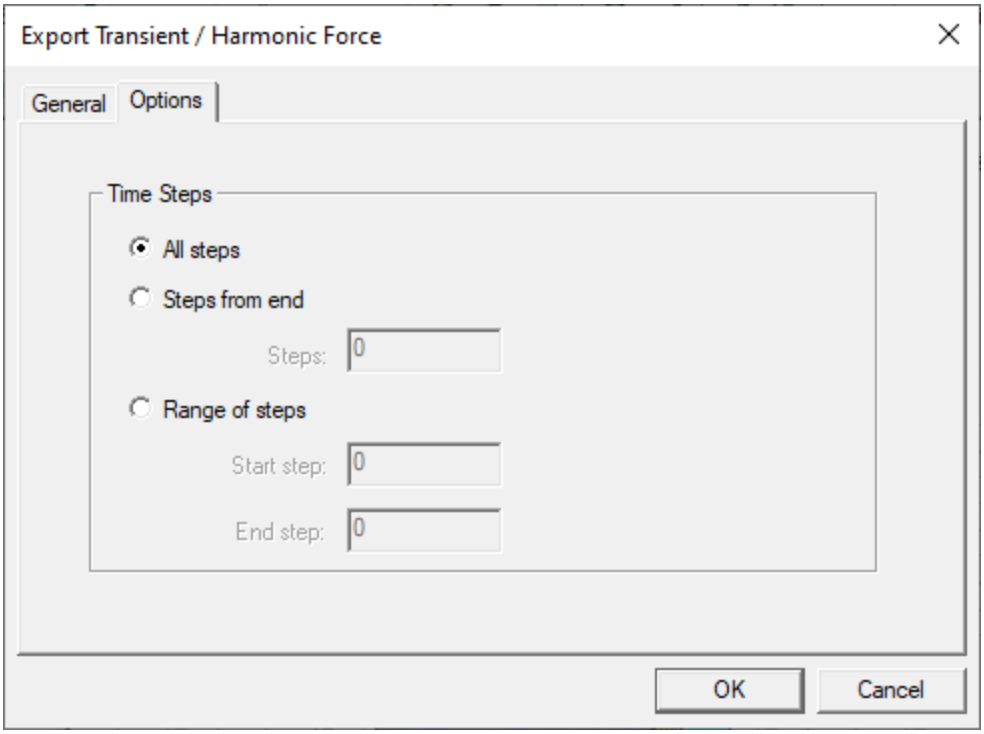


4. On the **Option** tab, settings allow you to control collection of data for export to Romax. You can choose to include **All steps**. The **Steps from end** setting allows you to choose the number of **Steps** from the end for which data is collected.

   > **Note:** If rotation motion is defined in the project, the input number of steps will be ignored. Instead the last cycle will be the time window.

   Alternatively, you can select a **Range of steps**, for which you can set a data collection **Start step** and a data collection **End step**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Click **OK** to close the window. When you click **OK**, all the results (nominal or parametric solution) will be exported to the specified folder.

Maxwell to Third-Party Coupling by UNV File

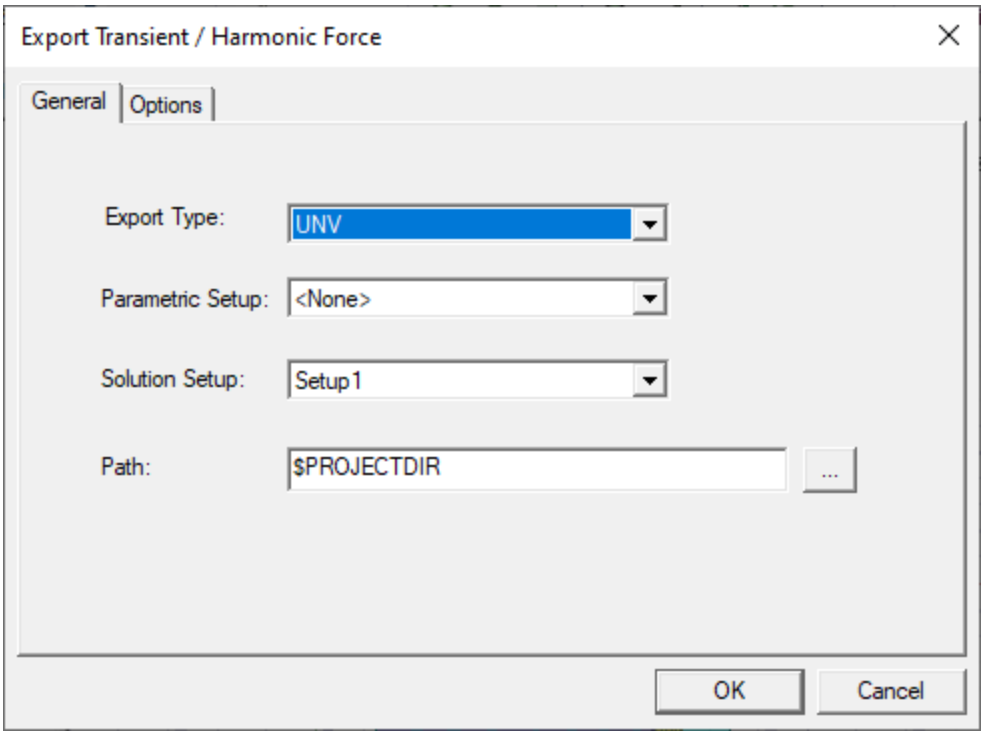
Maxwell provides option of one-way transient force coupling with any third party through standard universal (**UNV**) file format.

To enable output of Maxwell solutions for use with any third-party through UNV file format:

1. Follow the steps in **Export Transient/Harmonic Force from Maxwell** to enable transient force output before solving the Maxwell design.
2. After solving, either select **Maxwell 2D** (or **Maxwell 3D**) > **Export Transient/Harmonic Force**, or right-click on the Design in the Project Manager tree and select **Export Transient/Harmonic Force** to open the **Export Transient/Harmonic Force Dialog** panel.
3. On the **General** tab:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select **UNV** for the **Export Type**.
- Specify a **Parametric Setup** if any (If **<None>** is selected, only the nominal results will be exported).
- Select a **Solution Setup**.
- Specify the path of the location of output file (the default is the project directory).

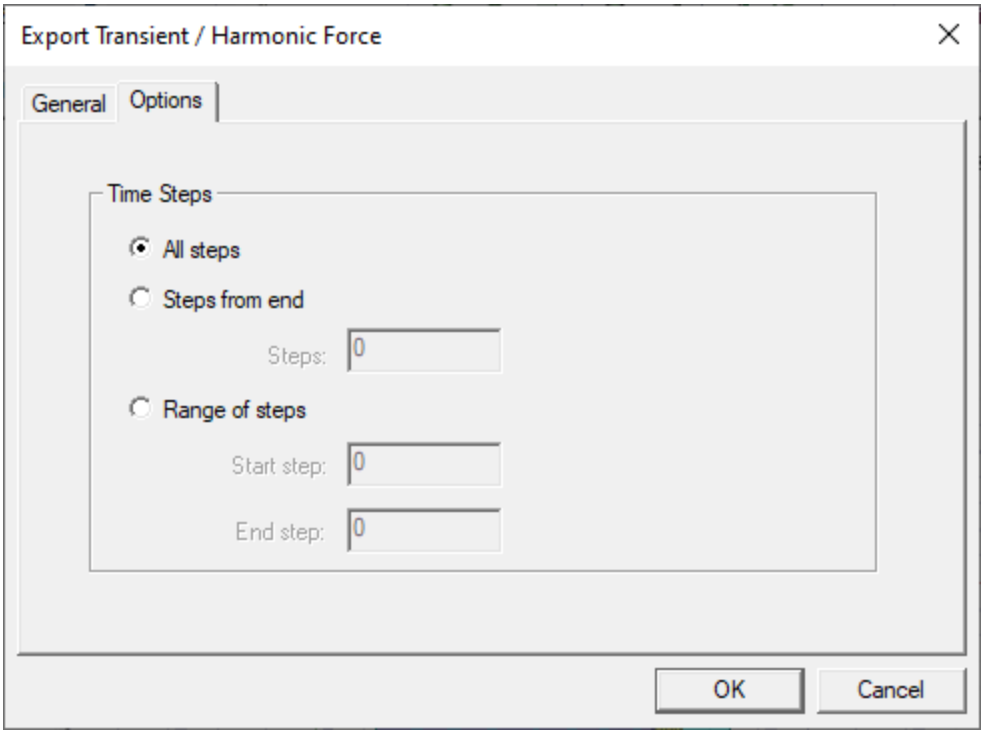


4. On the **Option** tab, settings allow you to control collection of data for export to a UNV file. You can choose to include **All steps**. The **Steps from end** setting allows you to choose the number of **Steps** from the end for which data is collected.

   > **Note:** If rotation motion is defined in the project, the input number of steps will be ignored. Instead the last cycle will be the time window.

   Alternatively, you can select a **Range of steps**, for which you can set a data collection **Start step** and a data collection **End step**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Click **OK** to close the window. When you click **OK**, all the results (nominal or parametric solution) will be exported to the specified folder.

Maxwell One-Way Coupling with Ansys Motion

Maxwell exports transient force to couple with Ansys Motion for acoustic simulation. There are two options to link to Motion software in Maxwell:

- Object-based: provides object based transient force and torque components on selected objects.
- Element-based (surface): provides element-based transient surface node force and mesh.
- Element-based (volumetric): provides element-based transient volumetric node force and mesh.

All forces and mesh are output with full model.

To enable output of Maxwell transient force solutions for link with Ansys Motion:

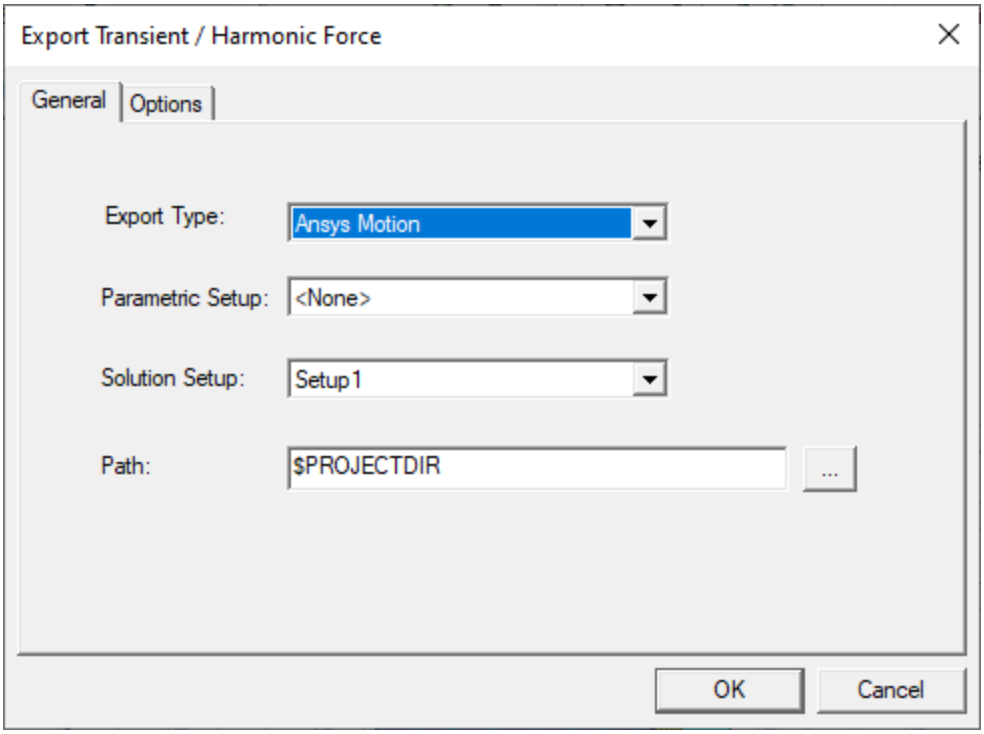
1. Follow the steps in **Export Transient/Harmonic Force from Maxwell** to enable transient force output before solving the Maxwell design.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. After solving, either select **Maxwell 2D** (or **Maxwell 3D**) > **Export Transient/Harmonic Force**, or right-click on the Design in the Project Manager tree and select **Export Transient/Harmonic Force** to open the **Export Transient/Harmonic Force Dialog** panel.

   > **Note:** Ansys Motion does not currently support harmonic force.

3. On the **General** tab:
   - Select **Ansys Motion** for the **Export Type**.
   - Specify a **Parametric Setup** if any (If **<None>** is selected, only the nominal results will be exported).
   - Select a **Solution Setup**.
   - Specify the path of the location of output file (the default is the project directory).

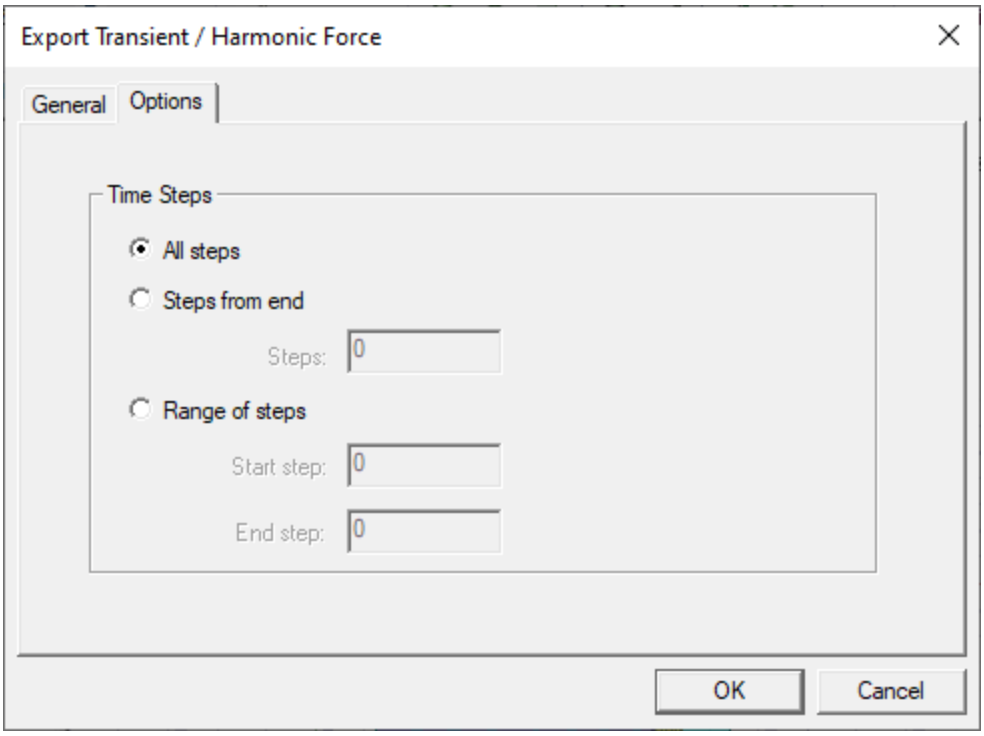


4. On the **Option** tab, settings allow you to control collection of data for export to UNV file (and .AMESH and .ANF files if element-based selected). You can choose to include **All steps**. The **Steps from end** setting allows you to choose the number of **Steps** from the end for which data is collected.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** If rotation motion is defined in the project, the input number of steps will be ignored. Instead the last cycle will be the time window.

Alternatively, you can select a **Range of steps**, for which you can set a data collection **Start step** and a data collection **End step**.



5. Click **OK** to close the window. When you click **OK**, all the results (nominal or parametric solution) will be exported to the specified folder.

Maxwell One-Way Coupling with Ansys Mechanical

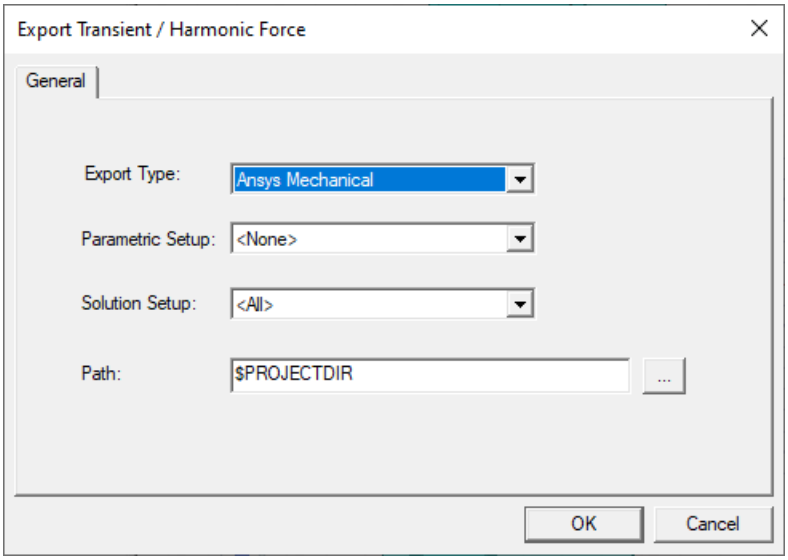
Maxwell 2D and 3D transient designs can export object-based harmonic force to couple with Ansys Mechanical for acoustic simulation.

To enable output of Maxwell harmonic force solutions for link with Ansys Mechanical:

1. Follow the steps in **Maxwell to Ansys Object-Based Harmonic Force Coupling** to enable object-based harmonic force output before solving the Maxwell design.
2. After solving, either select **Maxwell 2D** (or **Maxwell 3D**) > **Export Transient/Harmonic Force**, or right-click on the design in the Project Manager tree and select **Export Transient/Harmonic Force** to open the **Export Transient/Harmonic Force Dialog** panel.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. On the **General** tab:
   - Select **Ansys Mechanical** for the **Export Type**.
   - Specify a **Parametric Setup** if any (If **<None>** is selected, only the nominal results will be exported).
   - All solved solution setups will be exported, so the **Solution Setup** drop-down menu will always be **<All>**.
   - Specify the path of the location of output file (the default is the project directory).



4. Click **OK** to close the window. When you click **OK**, all the results (nominal or parametric solution) will be exported to the specified folder.

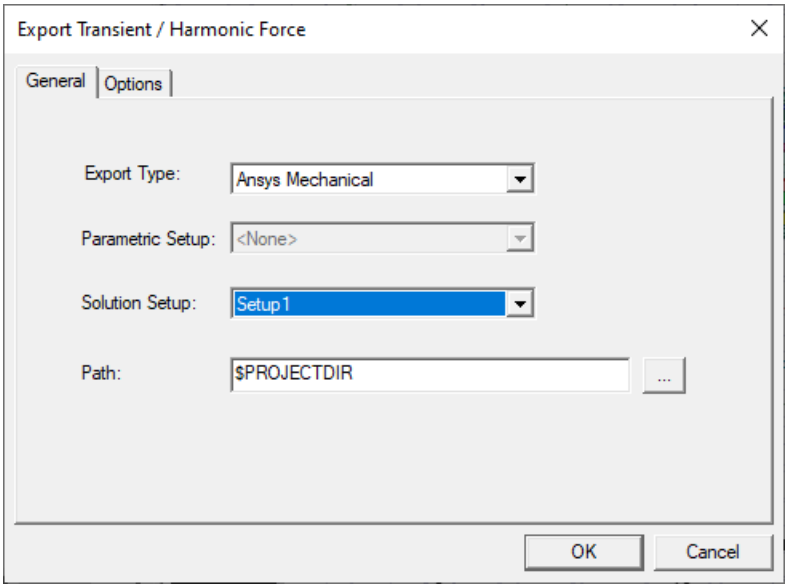
   You can then import the file into a WB harmonic component. For detailed information on setting up and using harmonic force coupling in Workbench, refer to the Workbench help topic: *Importing Data into a Harmonic Analysis*.

To enable output of Maxwell harmonic force solutions in a `.csv` file to couple with Ansys Mechanical:

1. Follow the steps in Maxwell to Ansys Element-Based Harmonic Force Coupling to enable element-based harmonic force output before solving the Maxwell design.
2. After solving, either select **Maxwell 3D** > **Export Transient/Harmonic Force**, or right-click on the Design in the project tree and select **Export Transient/Harmonic Force** to open the **Export Transient/Harmonic Force** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

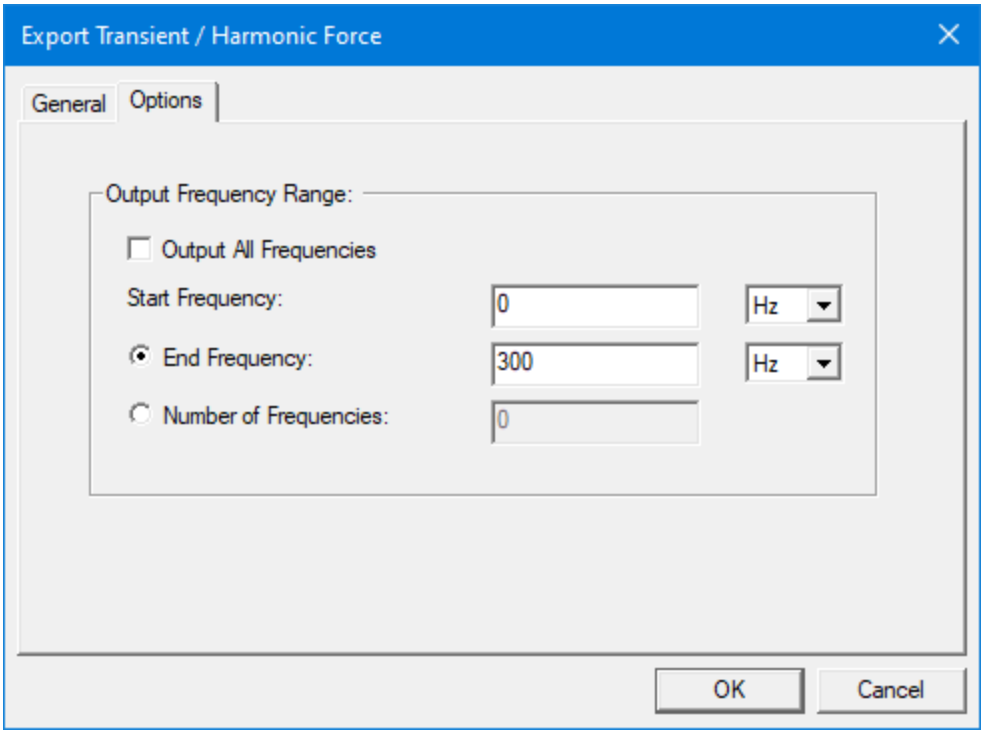
3. On the **General** tab:
    - Select **Ansys Mechanical** for the **Export Type**.
    - Specify a **Parametric Setup** as **<None>**.
    - Select a **Solution Setup** in the drop-down menu.
    - Specify the path of the location of output file (the default is the project directory).



On the **Options** tab, specify the **Output Frequency Range**. The file format in `.csv` file is (in line): Xc, Yc, Zc, Volume, Fx_re, Fx_im, Fy_re, Fy_im, Fz_re, Fz_im, Frequency1, Fx_re, Fxim, ……., Frequency2, ……

> **Note:**
>
> The frequency 0 (DC component) will be exported for either of the following conditions:
>
> - for **Output All Frequencies**: the first frequency will be 0.
> - for a specified range or number of frequencies: if the **Start Frequency** is zero.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



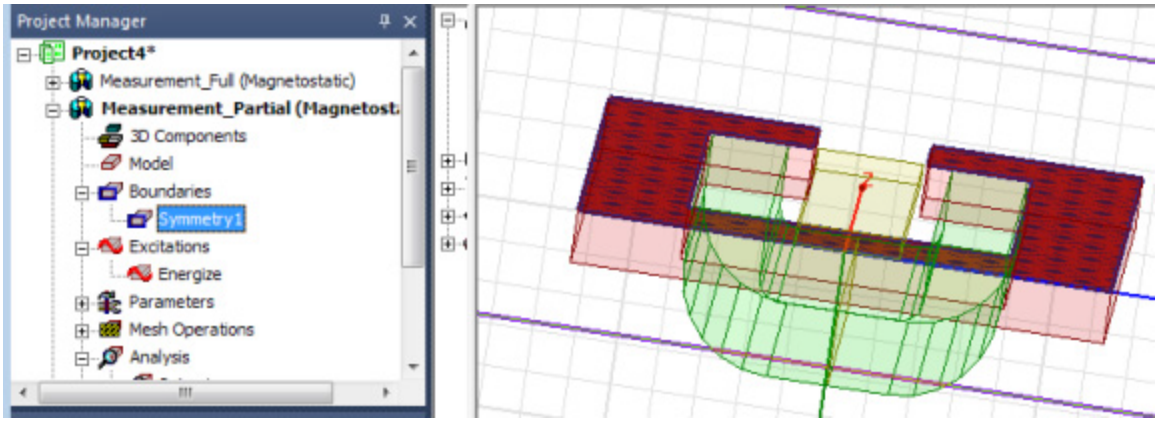
The **Number of Frequencies** defines the harmonic content where an upper frequency isn't necessarily the limit of interest. The Number of Frequencies is based on multiples of the fundamental frequency, which is defined by the duration of the transient dataset created by the user during pushback.

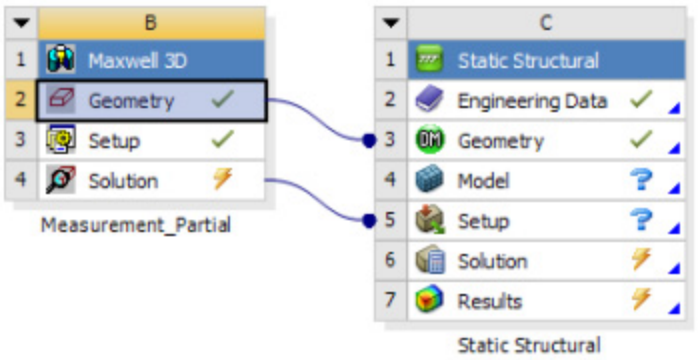
4. Click **OK** to close the window. When you click **OK**, the `.csv` file will be exported to the specified folder. You can then import the file into a WB harmonic component.

Maxwell Partial Model Coupling with Workbench

This feature allows Maxwell 2D and 3D partial model coupling with Workbench applications such as Mechanical and Thermal. The full model geometry and solution can be extracted through symmetry from the partial model. Symmetry boundary conditions are used to solve problems with symmetry and, thus, allow the users to take advantage of the significantly reduced problem size for a given accuracy.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



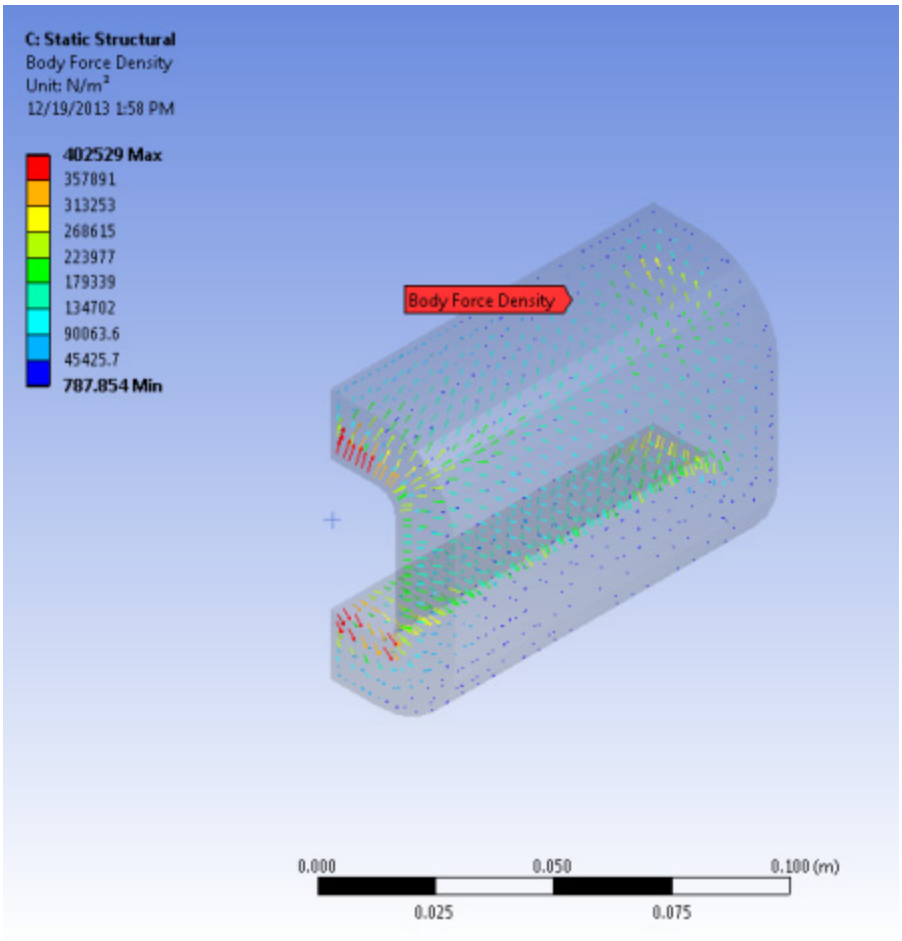


The following conditions must be observed for the Maxwell partial model:

- The model must be physically correct.
- Independent and dependent boundaries must be planar. If there is more than one pair of independent and dependent boundaries, all independents are coplanar and all dependents are coplanar.
- There are at most three mirroring symmetries, which are orthogonal to each other. If there is a pair of independent and dependent boundaries, then only one mirroring symmetry is allowed and it must be perpendicular to the axis of the independent and dependent boundaries.
- When the Maxwell side is a partial model and the Mechanical side is full model, full model loss / force will be mapped to the Mechanical side.
- When the Maxwell side is a partial model and the Mechanical side is also a partial model, only partial model loss / force will be mapped to the Mechanical side.
- When setting up loss transfers from Maxwell to WorkBench, if the user wants to map loss density on a face shared by two objects, this face only has to be used once in the setup.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

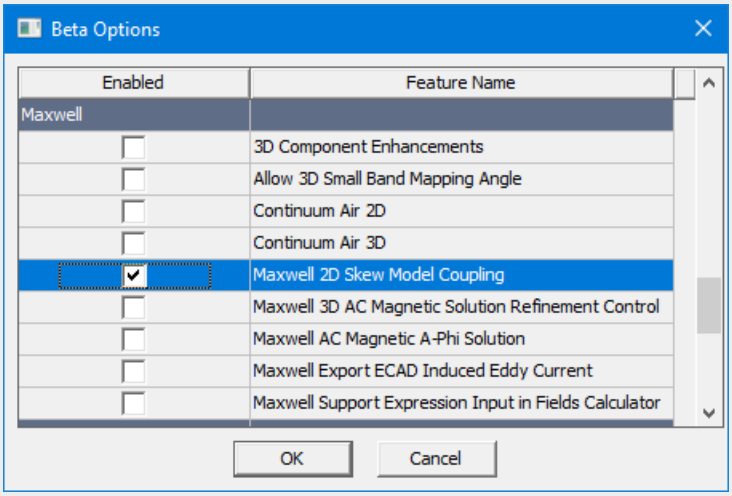
The user must ensure that the Maxwell partial model is physically correct, and that the full model in Workbench Mechanical, Thermal, *etc.*, matches the partial model. The loss/force data from the partial Maxwell model is then mapped to the full model in Workbench Mechanical for solving. The mapping is handled implicitly.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

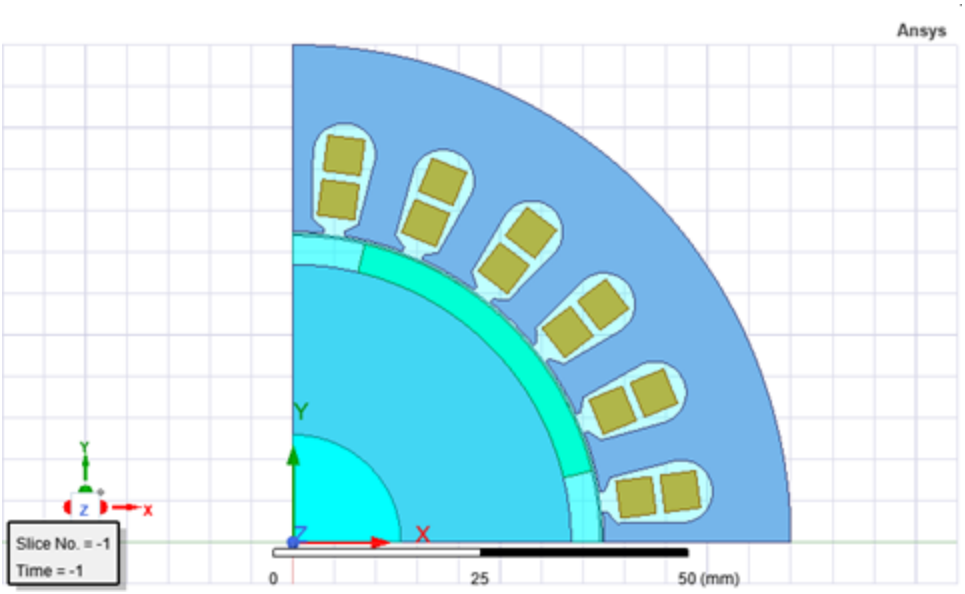
[Beta] Element-Based Multiphysics Coupling for 2D Multislice Skew Model

Note: You must enable this feature in General Options:Desktop Configuration > **Beta Options** before you can use it.



For Maxwell 2D transient skew models, this beta feature enables element-based multiphysics coupling to steady-state thermal and static/transient structural designs in Work bench. The target design could be 2D with matching geometry or 3D with equivalent geometry, *i.e.*, extrusion of 2D geometry with matching skew type.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Currently, RMxprt can only generate corresponding 3D models from electric machines with the continuous skew type, and corresponding 3D designs cannot be automatically generated from 2D skew model designs.

The Maxwell 2D skew model itself can be a partial model with an independent/dependent boundary, in which case, the target could be either a partial or a full model. If the target geometry is 2D, data from the default slice (middle) will be mapped. If the target geometry is 3D, it should be an extrusion of the 2D geometry with matching skew type. Any geometry in 3D that is beyond the 2D model depth will not be mapped.

Limitations

- 3D geometry creation from 2D Skew models is not supported. This needs to be done manually and then couple the two designs.
- Stress feedback on Structural is not supported. Regarding 3D to 3D coupling, Maxwell-Structural coupling provides forces from Maxwell and receives displacement and stress from Structural.
- 2D to 3D EM coupling is not supported.

Use the following work flow:

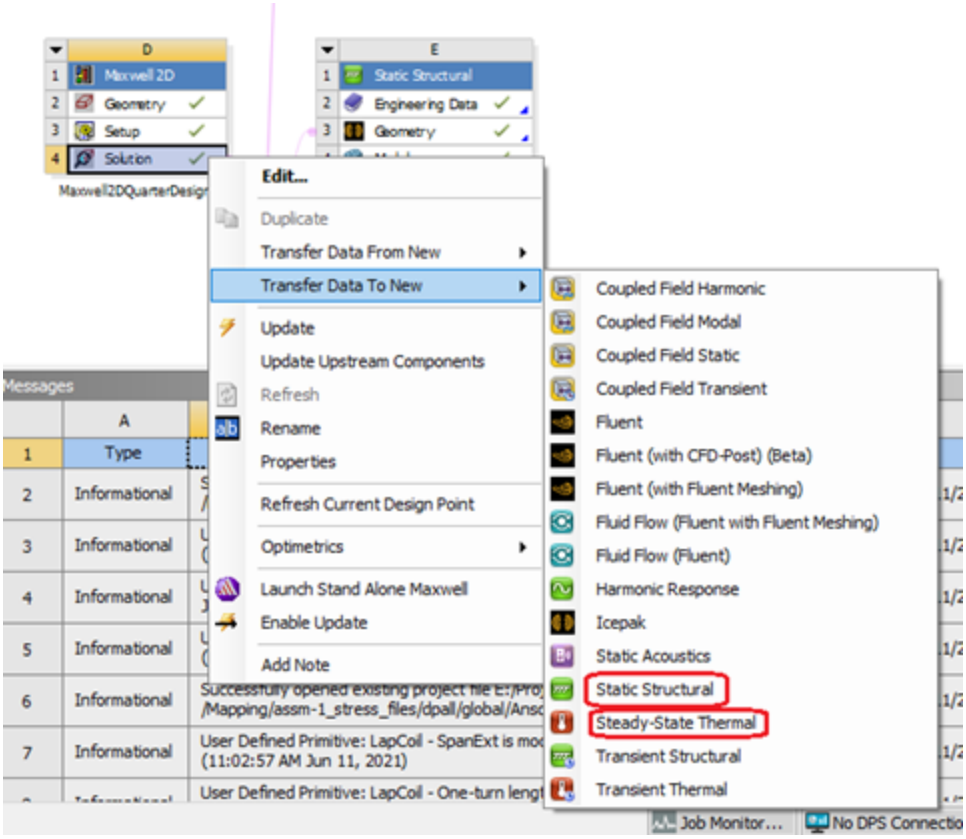
1. From RMxprt, create a 2D skew model design and corresponding 3D design.

   > **Note:** For detailed information on Skew Model settings, refer to the *Model Settings Tab* topic.

2. Solve the 2D design and save the project.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

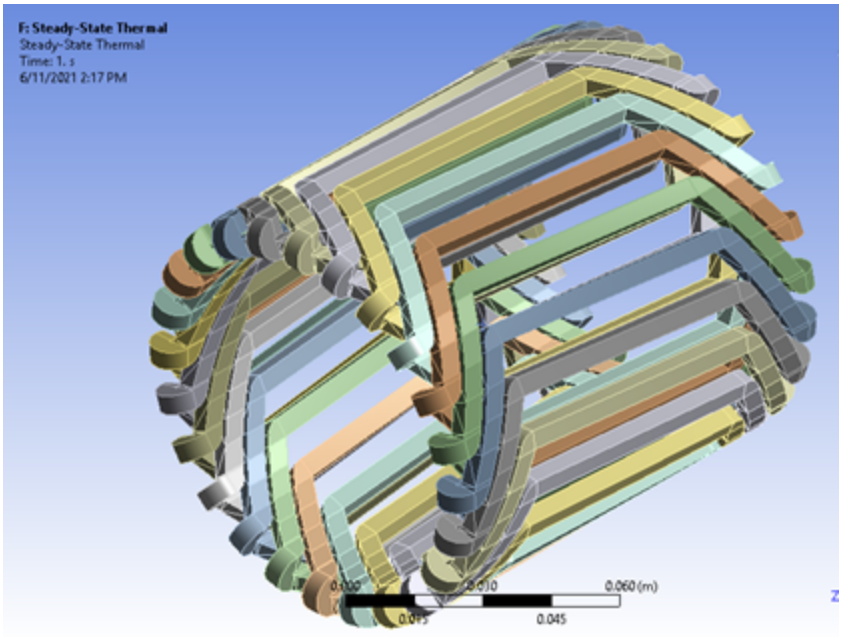
3. Drag the project into Workbench.
4. In the Maxwell 2D design, select **Solution** > **Transfer Data To New** > **Steady-State Thermal** or **Static Structural**.



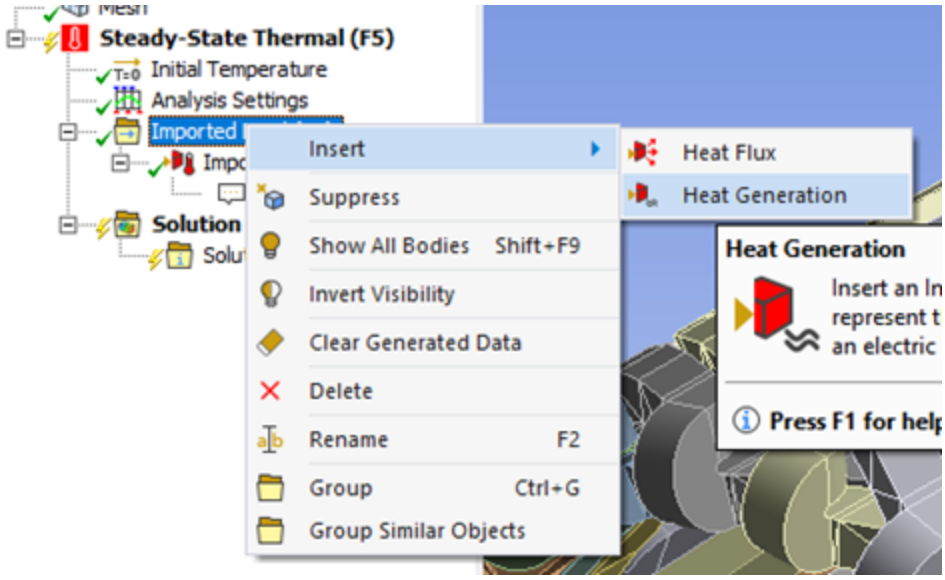
5. Drag the Geometry from the Maxwell 3D design to the Geometry of the newly created Mechanical design.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Refresh and/or Update the Geometry and Model in the Mechanical design.

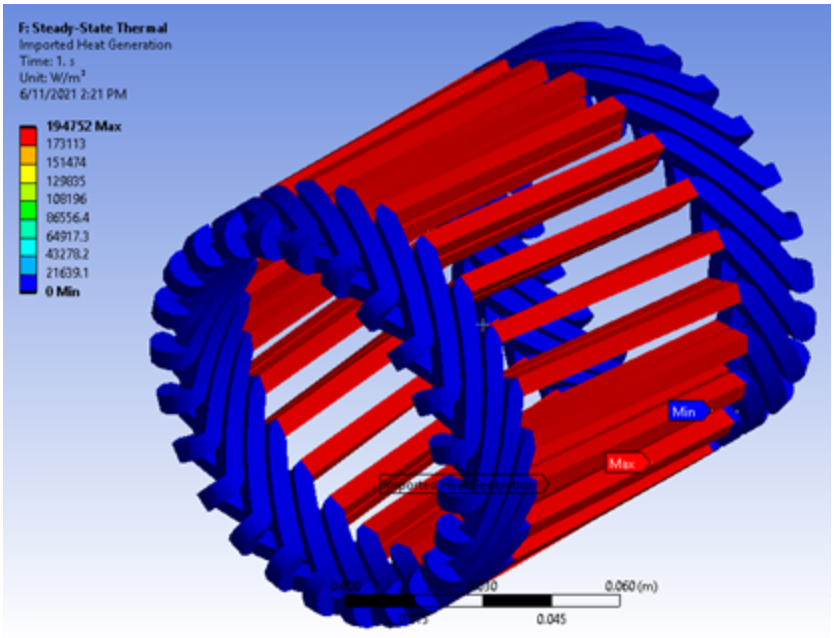


7. In the Maxwell 2D design, select **Solution** > **Update**.
8. In the Mechanical design, **Setup** > **Refresh**.
9. Double-click **Setup**.
10. From **Imported Load** > **Insert** > **Heat Generation**, select lossy objects and set end time.



11. Import the loads. Forces could be imported similarly.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Near Field Coupling between Maxwell and HFSS

The purpose of this coupling is to allow **Maxwell 3D** frequency domain solution magnetic fields to be used as a near field source in the coupled **HFSS** design at exactly matched frequency.

> **Note:** There is no **Far Field Wave** coupling datalink between **HFSS** and **Maxwell**.

The suggested sequence of steps to create the link between **Maxwell** and **HFSS** is as follows.

1. Create and solve the **Maxwell 3D** frequency domain application at the chosen frequency.
2. Create the **HFSS** model.
3. Add "mapping" object to **HFSS** model.
4. Select all faces of the mapping object and apply a radiation boundary:
   - check the **Advanced Options** check box.
   - select the **Enforced field** radio button.
5. Click **HFSS** > **Excitations** > **Assign** > **Incident Wave** > **Near Field Wave**.

   The **Incident Wave** wizard appears.
6. In the wizard, enter a **name** for the source or accept the default name and click **Next**.
7. In the **Incident Wave Source: Near Field Wave Options** dialog box, leave the translation and rotation data fields at their default zeros if the objects in the coupled designs have the desired location / orientation or change the values as needed.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** You can change the location of the source design origin and orientation (rotation) to perform different **HFSS** simulations without re-solving the **Maxwell** design.

8. Click the **Setup Link** button to specify the location of the source Maxwell design.
9. Select **OK** to exit the **Setup Link** dialog box.
10. Select **Finish** to finalize the Near Field link setup.

Magnetic Bias Source Coupling between Maxwell and HFSS

HFSS designs featuring ferrite materials can take advantage of non-uniform, static magnetic biasing fields to provide accurate design and simulation results. To achieve this goal it is possible to set up a coupling datalink between the HFSS target design and a Maxwell 3D magnetostatic source design.

The suggested sequence of steps to create the link between the HFSS and Maxwell 3D designs is as follows:

1. Create and solve the Maxwell 3D design which contains the ferrite object and the corresponding non-uniform biasing fields.
2. Create the HFSS design which can have entirely different geometry but which contains the ferrite object(s) present in the linked Maxwell 3D design.

   > **Note:** Make sure that in the coupled designs (HFSS and Maxwell) the ferrite objects have the same position and orientation in the global coordinate system.

3. In the HFSS design, assign appropriate ferrite material property to object.
4. In the HFSS design, set the magnetic bias source (only one allowed per HFSS design):
   - Select the object(s) to be assign the non-uniform magnetic bias.
   - Click **HFSS** > **Excitation** > **Assign** > **Magnetic Bias** in the menu.

   The **Magnetic Bias Source** dialog box appears:

   a. Enter a **Name** for the source.
   b. Select the **Non-Uniform** radio button.
   c. Click the **Setup Link** button to specify the applicable Maxwell 3D project, design and solution for the link.
   d. Select **OK** when done.
5. Select **Finish** to finalize the Magnetic Bias link setup.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tip: It is possible to have the datalink create the Maxwell project automatically if it doesn't exist. When selecting the "Setup Link" button above, entering nonexistent Maxwell project and design names will result in the creation of the needed Maxwell entities with the entire HFSS geometry included. All unneeded HFSS objects can be removed, and the rest of the Maxwell model can be built around the ferrite object(s).

Coupling Maxwell with Ansys Motion (Stand-Alone)

Direct cosimulation between Maxwell and Ansys Motion applications makes possible magnetic latch modeling, which is typically used for modeling magnetic latching structures such as magnetic chargers used by mobile devices. The Ansys Motion application is the primary (controlling) design, while the Maxwell application serves as the secondary design. During the cosimulation, Motion provides objects position data to the Maxwell application, then the Maxwell application feeds force/torque data for these objects back to Motion.

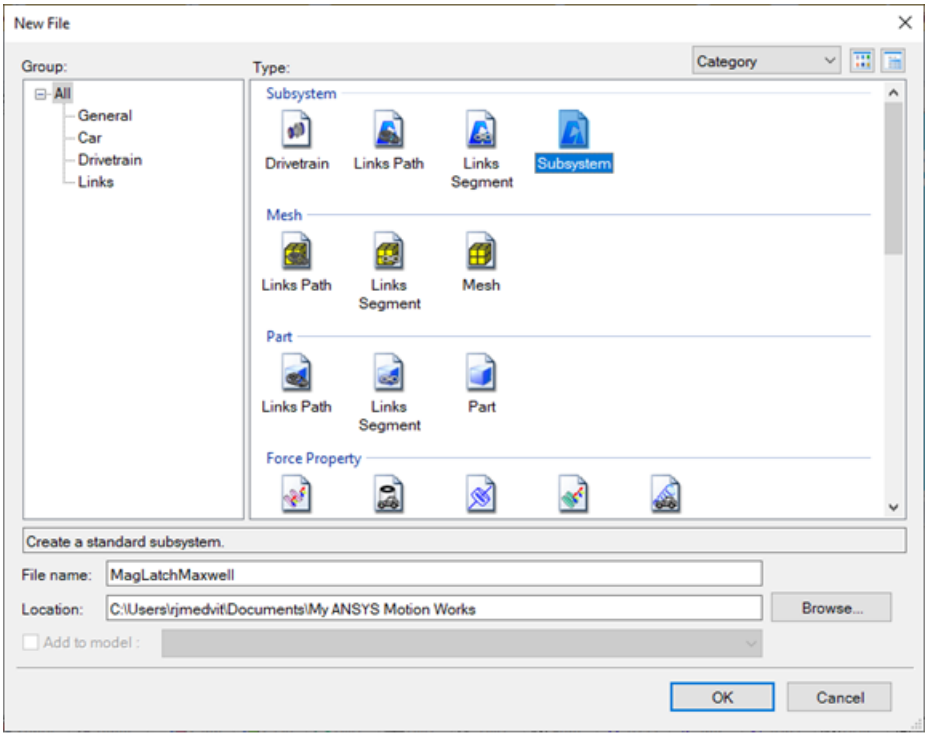
Note: For cosimulation between the Maxwell application and the Ansys Motion application integrated in the Workbench software, see *Coupling Maxwell with Ansys Motion in Workbench*.

The suggested workflow for magnetic latch modeling using cosimulation with the Ansys Motion stand-alone preprocessor and the Maxwell application is outlined in this topic.

Note: For details on using the Motion Preprocessor for the following steps, see the *Ansys Motion Preprocessor Manual* installed with the Ansys Motion application.

1. In **Ansys Motion Preprocessor**, create a new **Subsystem** file for which you specify a project path and project name.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



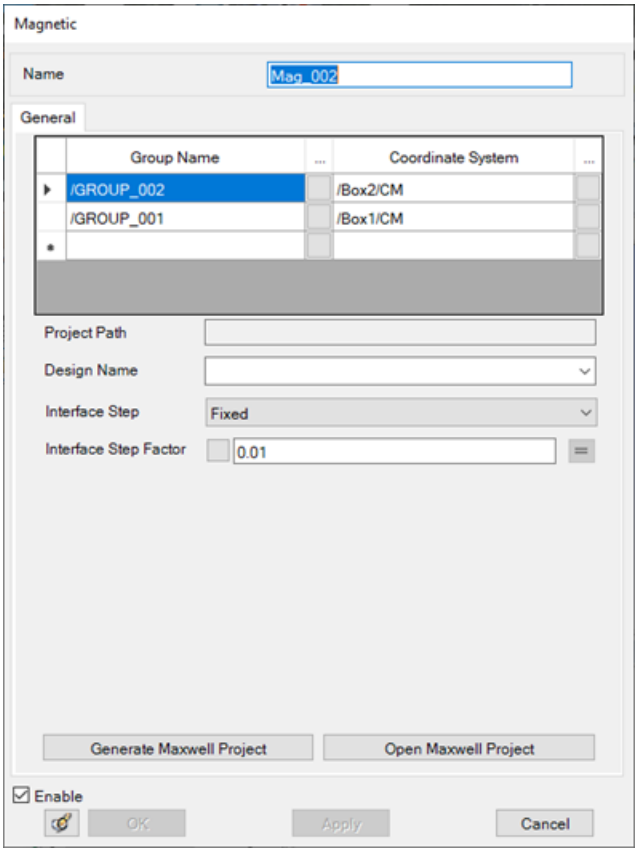
2. Import a CAD file of the model. You can modify object properties such as materials as needed.
3. After creating the model, **Group** objects of interest that have position changes and/or need force/torque calculations in the Maxwell design.

> **Note:** For performance considerations, it is recommended that you not perform force/torque calculations on all objects.

> **Note:** The Ansys Motion application interface is only available on Windows, but you can set up an Ansys-Maxwell cosimulation on Windows, and then run the simulation on Linux, see Running an Ansys Motion-Maxwell Cosimulation on Linux for details.

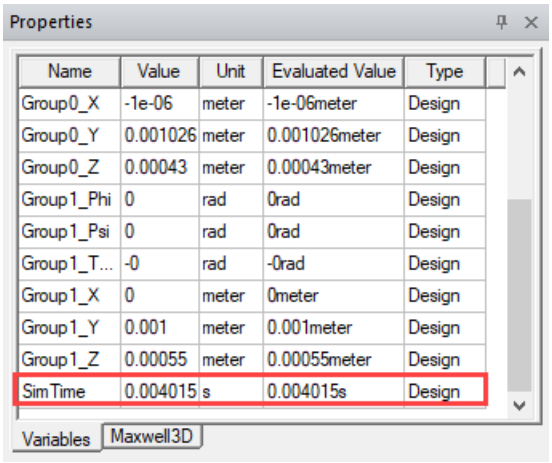
4. On the Ansys Motion Preprocessor **Force** tab, click the **Magnetic** button, pick the desired **Groups**, then click **OK**. In the **Magnetic** dialog box enter the desired **Interface Step Factor** (in seconds) or accept the default 0.01. This sets the Maxwell simulation rate. The **Coordinate System** assigned to each group defaults to center-of-mass.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Click **Generate Maxwell Project**. A Maxwell desktop will be launched automatically with the generated Maxwell magnetostatic design, which includes the following:
    - The model geometry (including necessary coordinate systems)
    - If a Material is not found in the Maxwell system or user libraries, a new material with the default properties of *vacuum* will be created.
    - Force/torque calculation setups
    - A [default] Solve setup
    - Data table for force/torque parameters
    - Field plot for magnetic objects
    - The Ansys Motion application passes the Motion solve timestep as a variable named **SimTime** to the Maxwell application. Excitations can be created depending on this variable.

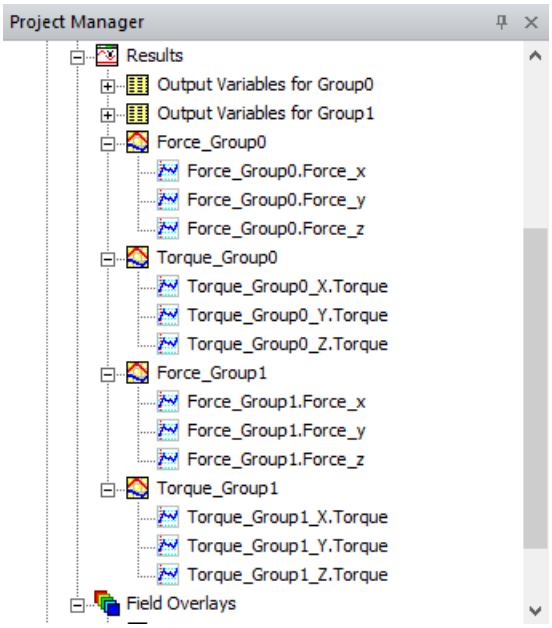
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- A parametric setup called **MotionHistory** will be automatically created and will be updated for each timestep, keeping a table of variations for all the simulation steps. This data can be used to create an animation of all the simulation steps.

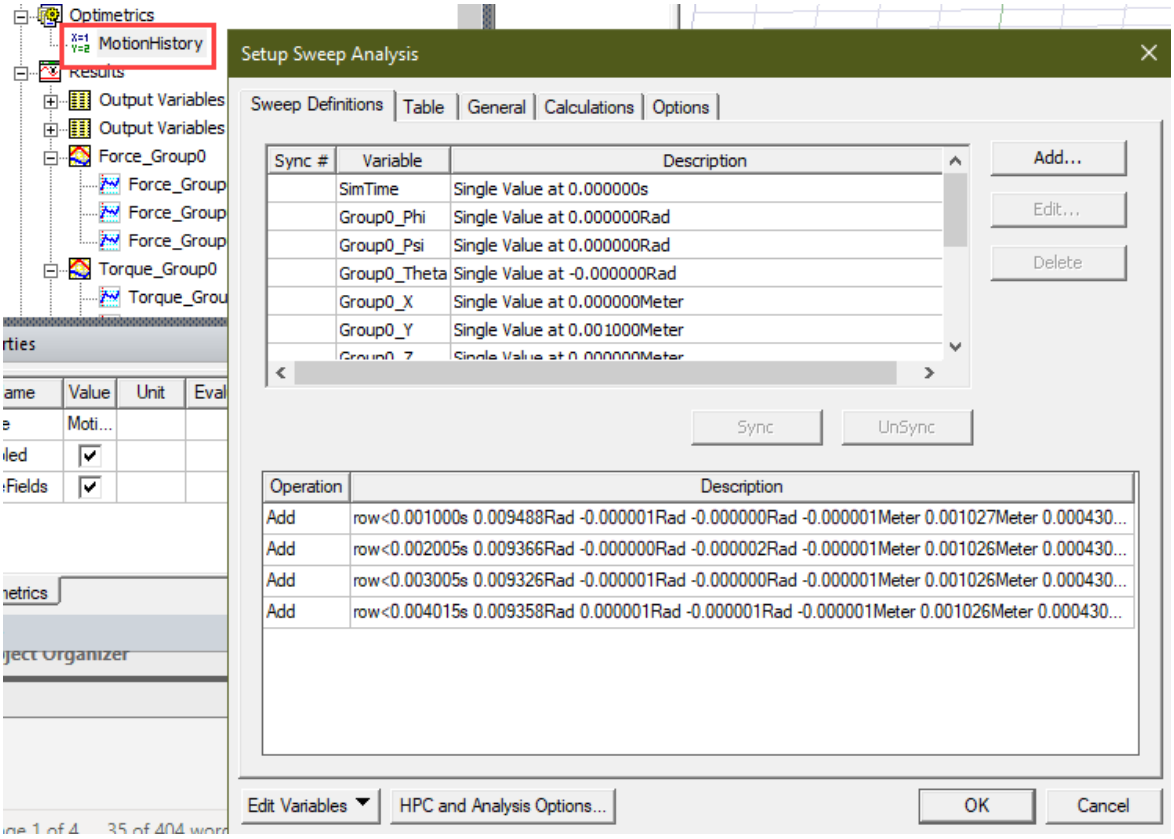
6. Optionally, you can customize various settings manually in the Maxwell application, such as
   - Material properties for objects, especially characteristics for permanent magnets
   - Excitations can be created dependent on the **SimTime** variable.
   - Coordinate system settings
   - Mesh setup
   - Solve setup
7. After the setups on both the Ansys Motion and Maxwell (if any) applications are completed, you can start the cosimulation solve in Motion by selecting **Simulation** > **Run** on the **Home** tab.
8. After cosimulation is completed, Maxwell Project Manager Results should show two reports for each group, one for force, one for torque. The reports plots are force/torque against **SimTime**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



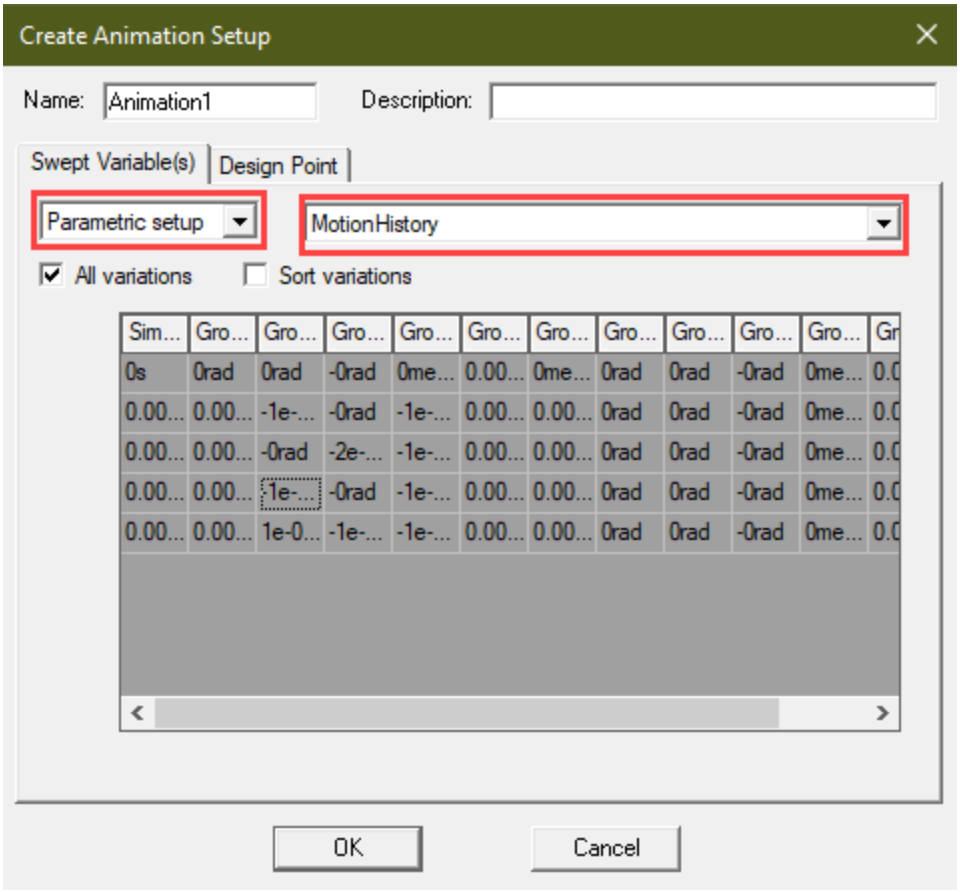
9. Optionally, you can create an animation of all the simulation steps. A parametric setup called **MotionHistory** is automatically created and updated for each timestep, keeping a table of variations for all the simulation steps.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



10. Optionally, to create an animation using the automatically generated **MotionHistory** parametric setup:
    a. Right-click on the desired field plot and click **Animate…**
    b. In the **Create Animation Setup** dialog, select **Parametric setup** in the first box, then select the parametric design called **MotionHistory** in the second box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



c. Click **OK** to create the animation.

Related Topics

Coupling Maxwell with Ansys Motion in Workbench

Running an Ansys Motion-Maxwell Cosimulation on Linux

Running an Ansys Motion-Maxwell Cosimulation on Linux

For users who would like to use the compute power of a Linux machine, an Ansys Motion-Maxwell cosimulation can be set up on a Windows machine, and then executed on a Linux machine. This topic details this procedure.

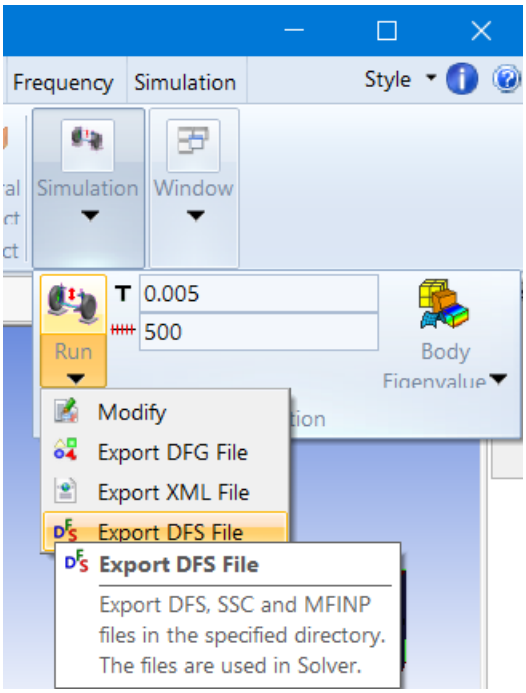
1. Install the pyaedt package in the CPython of the Ansys installation: In a Linux terminal, enter

   ```
   <path to ansys_inc>/commonfiles/CPython/3_
   10/linx64/Release/python/runpython -m pip install pyaedt
   ```

2. On a Windows machine, open the stand-alone version of Ansys Motion Preprocessor.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Set up the Motion-Maxwell cosimulation; see Coupling Maxwell with Ansys Motion (Stand-Alone) for more details.
   The Ansys Motion subsystem file (.dfsub file) and the Maxwell project generated by Motion (.aedt file) must be in the same folder.
4. Export a DFS file for use on Linux: from the Motion **Home** tab, select **Simulation** > **Run** > **Export DFS file**. It should be exported to the same folder as the Motion and Maxwell files.



5. Export the DFG file to the same folder as the Motion and Maxwell files: from the Motion **Home** tab, select **Simulation** > **Run** > **Export DFG file**. This file is necessary to postprocess the solved model on Windows.
6. Copy the entire folder from the Windows machine to a Linux machine.
7. On the Linux Machine, cd to the copied folder, and use the rundfs.sh script in the Motion installation folder to run the cosimulation:

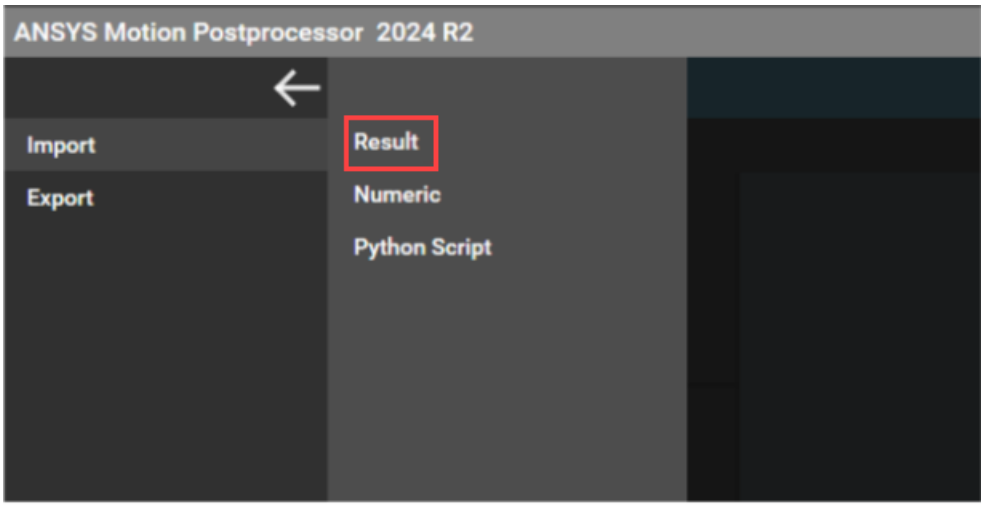
   ```
   <path to ansys_inc>/v242/Motion/solver/rundfs.sh -i Name of dsub file
   ```

   There will be a message when the simulation runs to completion:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
* END TIME = Wed Feb 28 01:25:28 2024

< END OF ANSYS LINUX SOLVER >

End : FeeLibrary
*   ANSYS LINUX SOLVER IS SUCCESSFULLY FINISHED
* ENDING ANSYS LINUX SOLVER ( VERSION: 24.20 )

{tyu@seoc7motiondv02}{/home/tyu/Case1}$
```

8. Copy the entire folder back to the Windows machine for postprocessing.
9. Open the Ansys Motion Postprocessor to read the solved files in the solved folder:



The animation will be displayed after opening the .dfr file.

Coupling Maxwell with Ansys Motion in Workbench

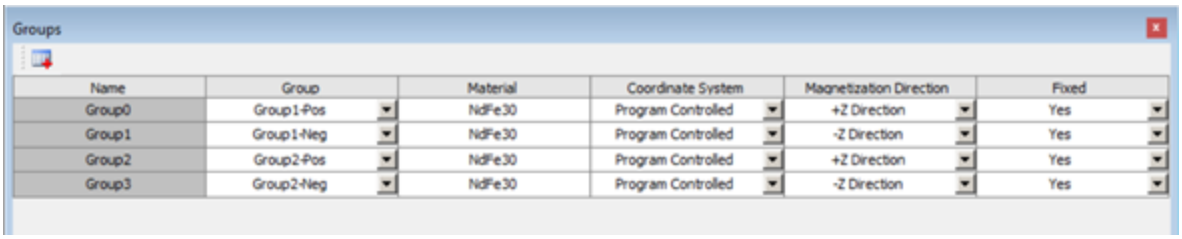
Direct cosimulation between the Maxwell application and the Workbench Ansys Motion application makes possible magnetic latch modeling, which is typically used for modeling magnetic latching structures such as magnetic chargers used by mobile devices. The Ansys Motion application is the primary (controlling) design, while the Maxwell application serves as the secondary design. During the cosimulation, the Motion application provides objects position data to the Maxwell application, which then feeds force/torque data for these objects back to the Motion application.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: The Maxwell project is not a part of the Workbench project. Instead, the Motion project in the Workbench software keeps a path pointing to the Maxwell project, which it generates and launches to calculate magnetic force/torque.

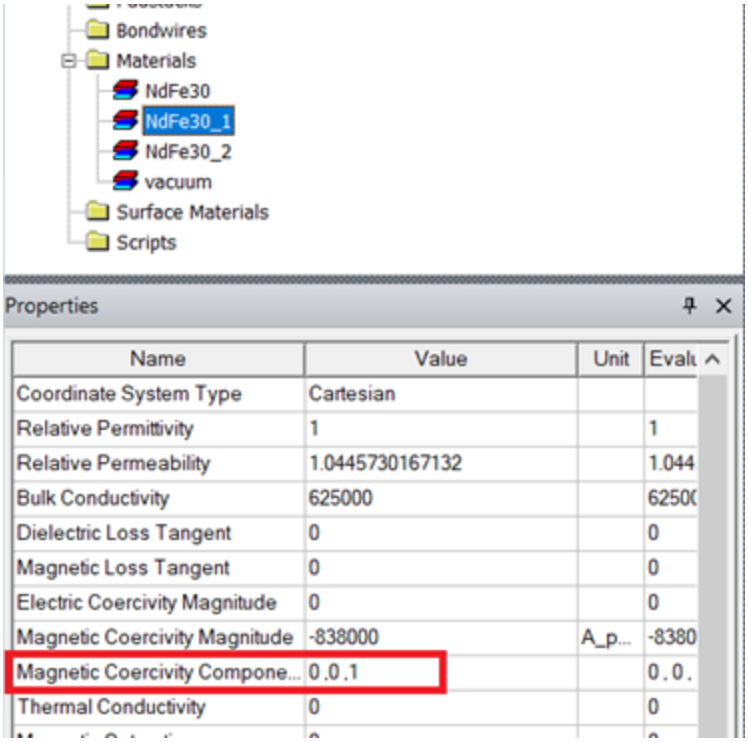
Note: For cosimulation between the Maxwell application and the stand-alone Ansys Motion Preprocessor, see to Coupling Maxwell with Ansys Motion (Stand-Alone).

The Workbench Ansys Motion application can define the material name, coordinate system, and magnetization direction of a group.



The material name and the magnetization direction are passed to the Maxwell application, which creates the materials with the correct Magnetic Coercivity Component, then assigns them to the corresponding objects. If the material has a different direction than the system default value, a new clone of the material will be created with a new name adding a number at the end.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

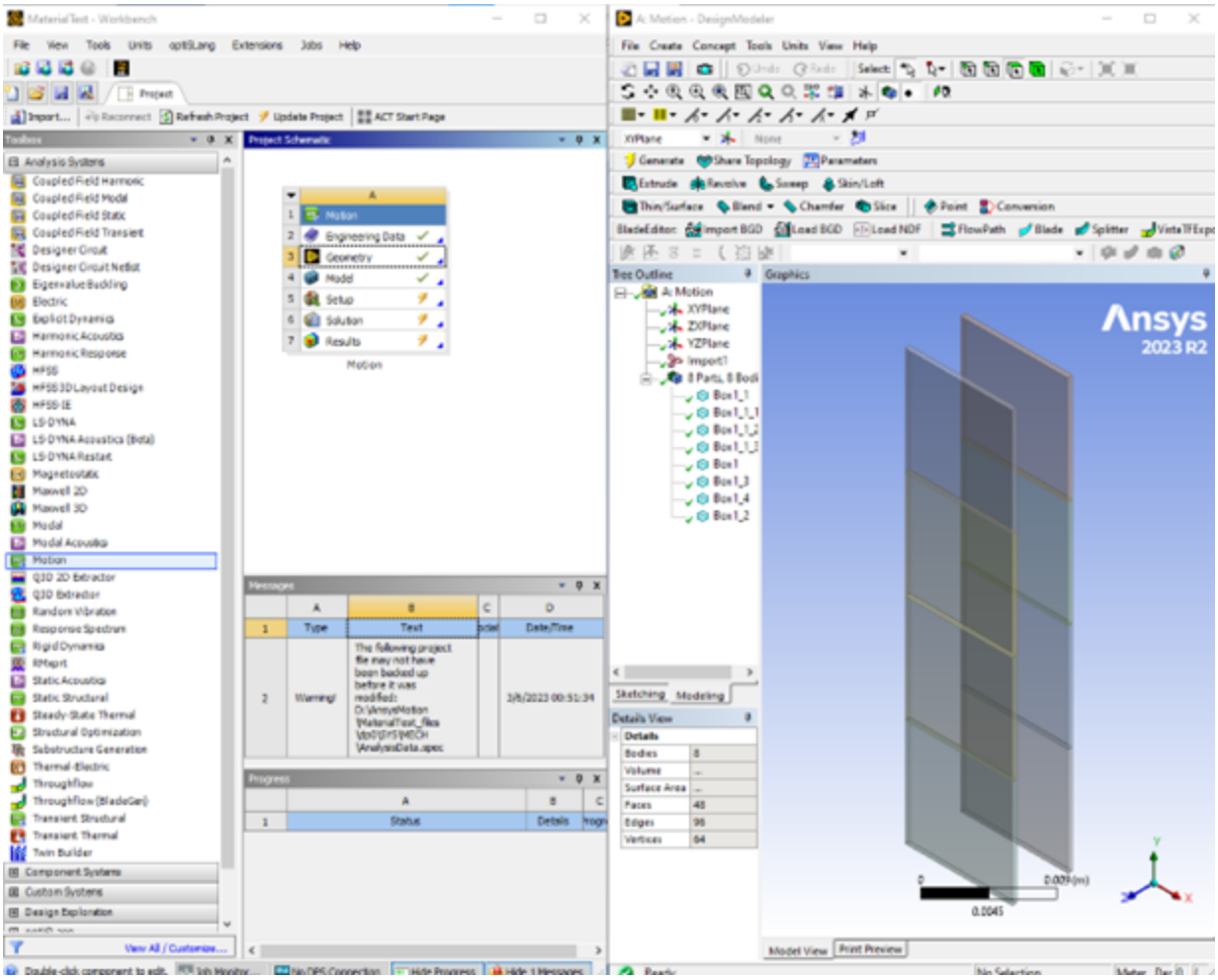


The suggested workflow for magnetic latch modeling using cosimulation between the Ansys Motion application and the Maxwell application is as follows:

> **Note:** For details on using the Motion application for the following steps, refer to the *Ansys Motion in Workbench User Guide* installed with the Ansys Motion software.

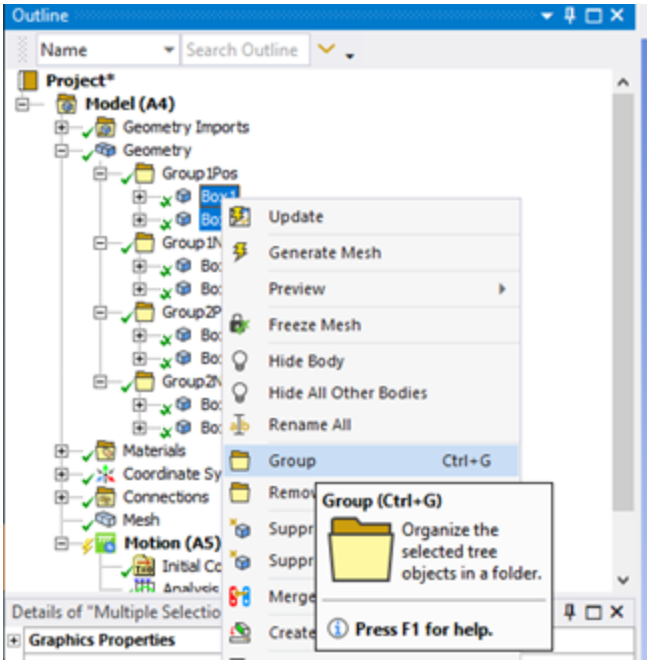
1. Create a Motion project in the Workbench application; then draw or import geometry in the design modeler.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

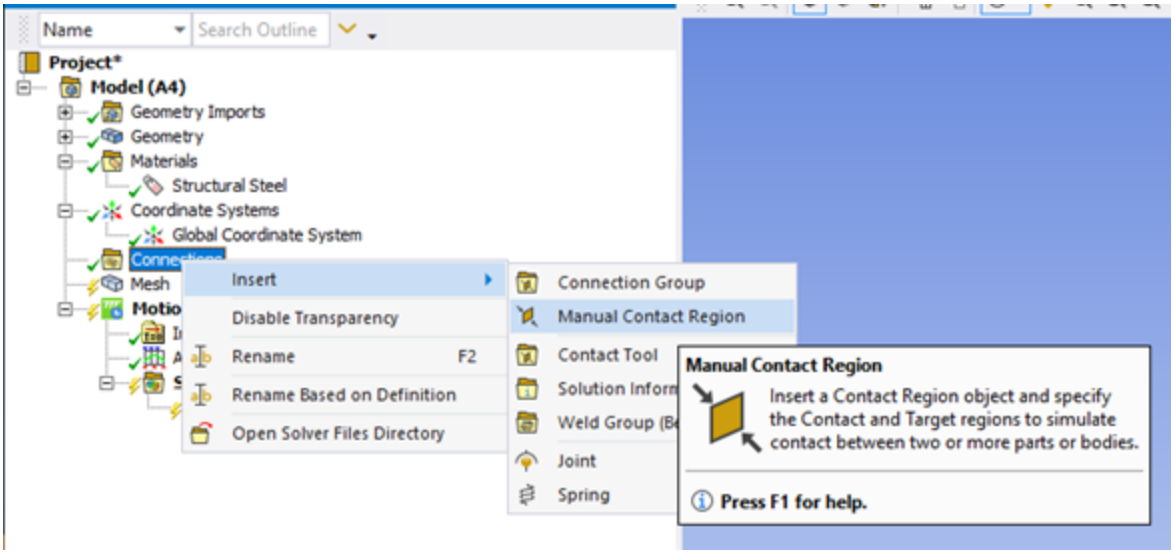


2. In the Motion application, create groups of objects that are fixed together and have the same material.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

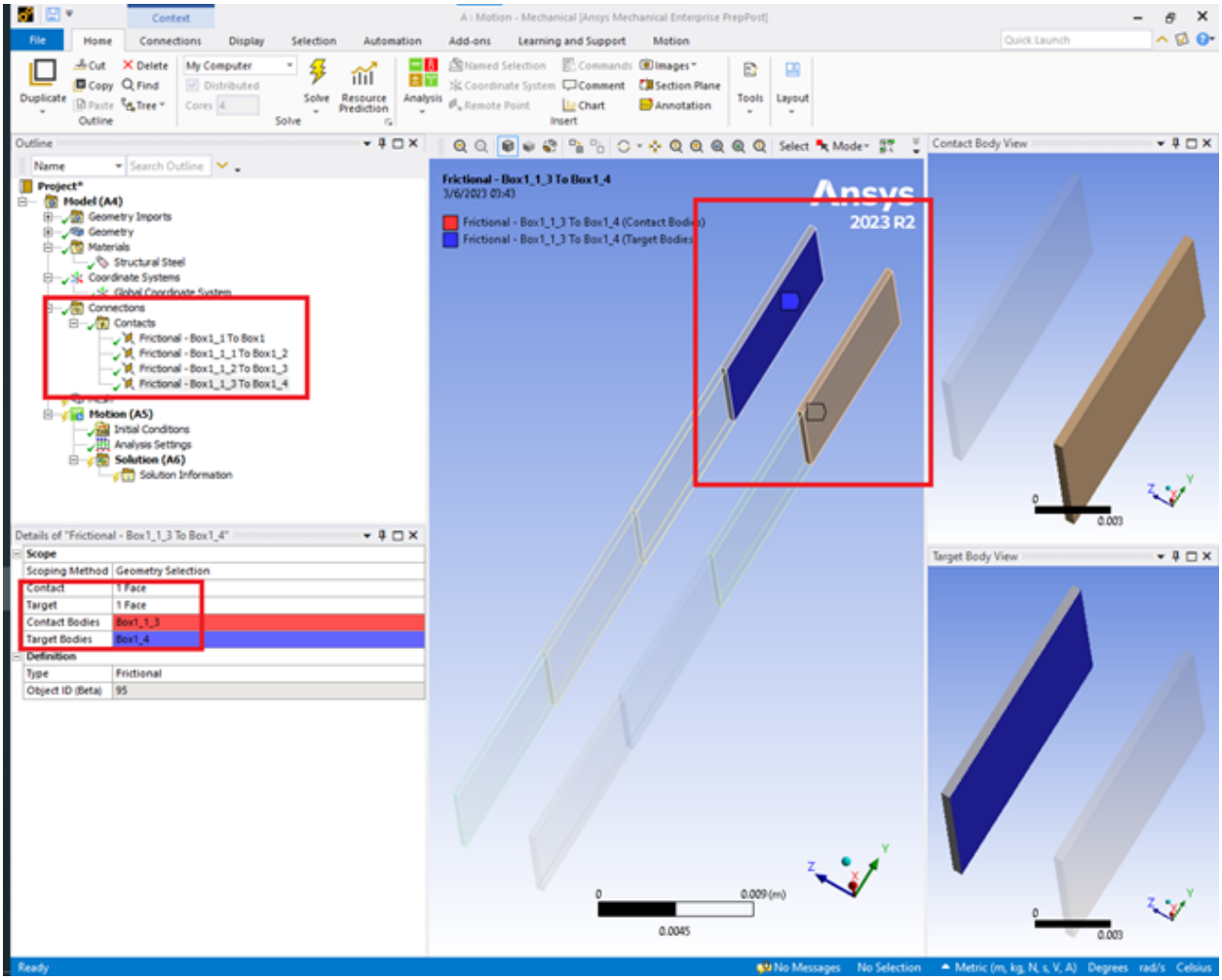


3. Delete the automatically created contacts, and create contact connections between the faces that will be touching each other.



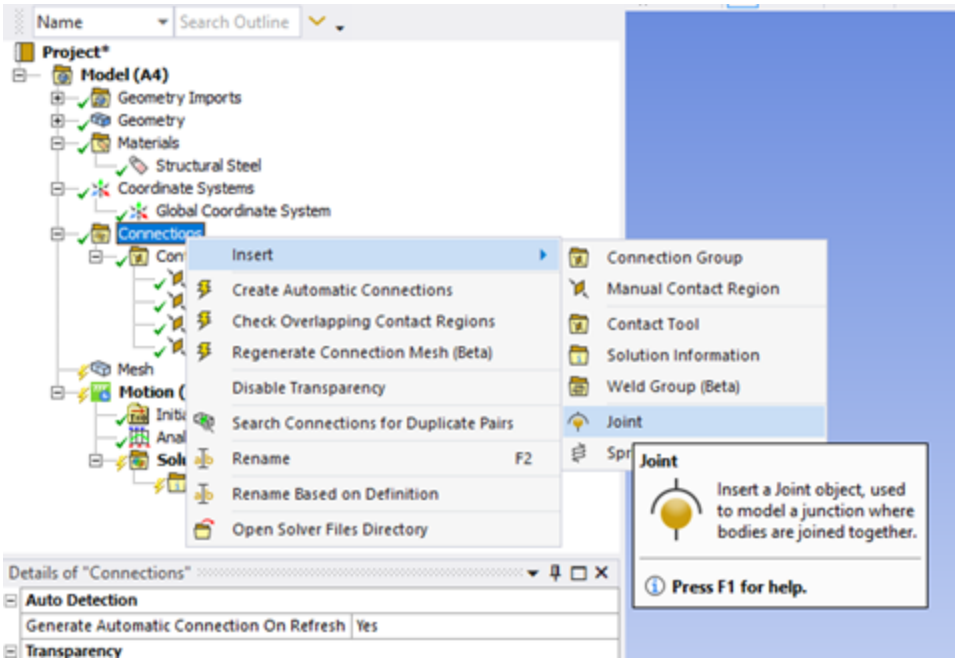
This is an example of creating four contacts between four pairs of faces:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



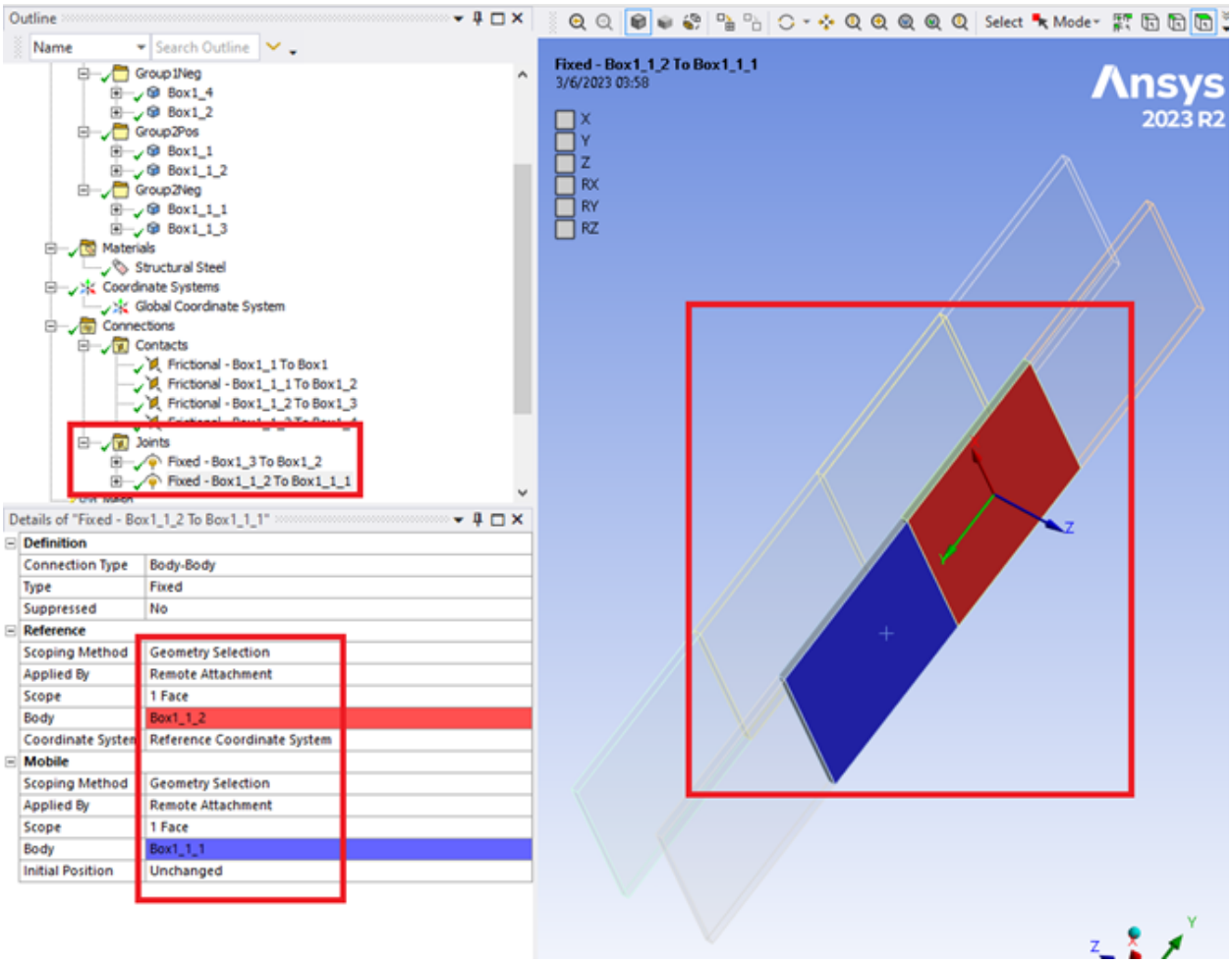
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Create Fixed Joints on the groups that move together.



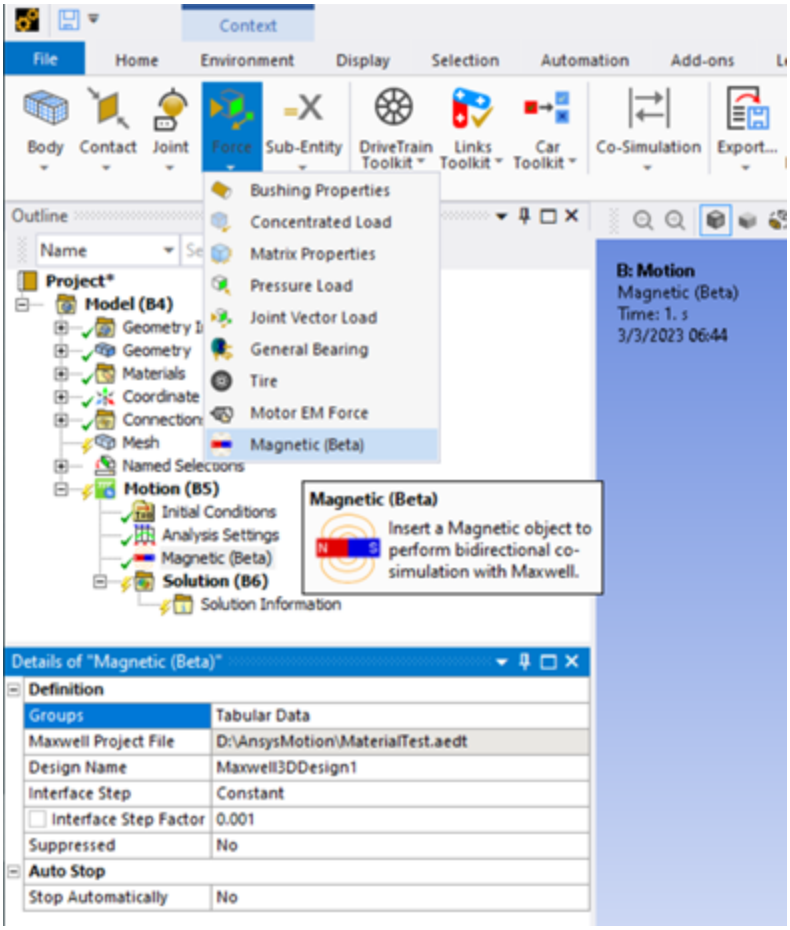
The objects within a group are fixed together by default, so only one Fixed Joint is required to connect two groups:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

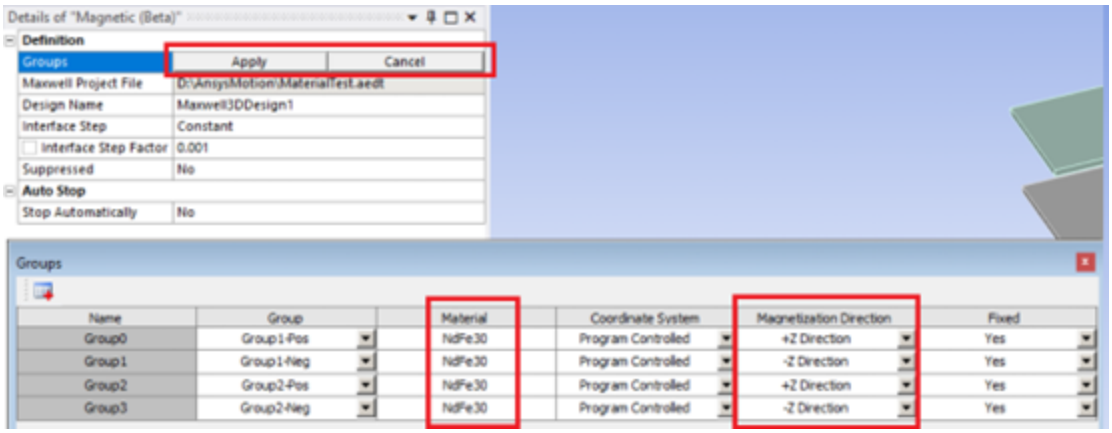


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Create a Magnetic Force in the Motion application.

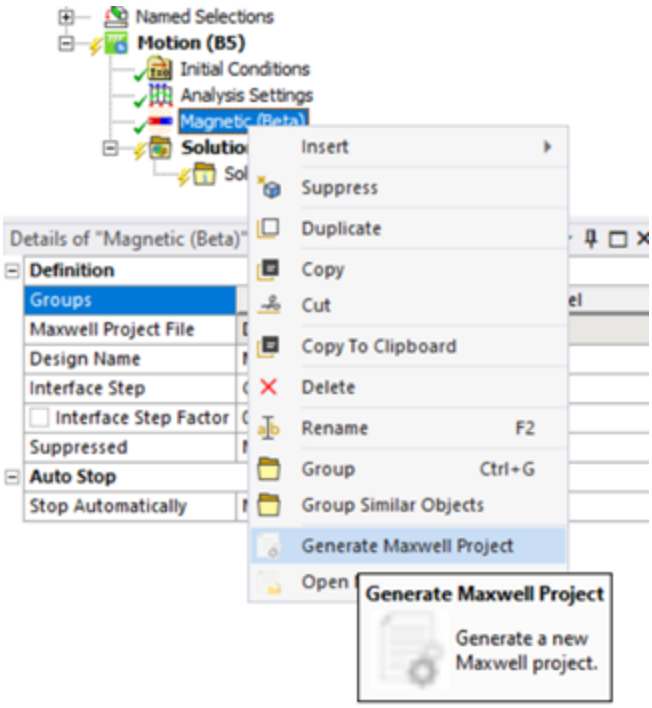


6. Assign the material name and magnetization direction to the groups.



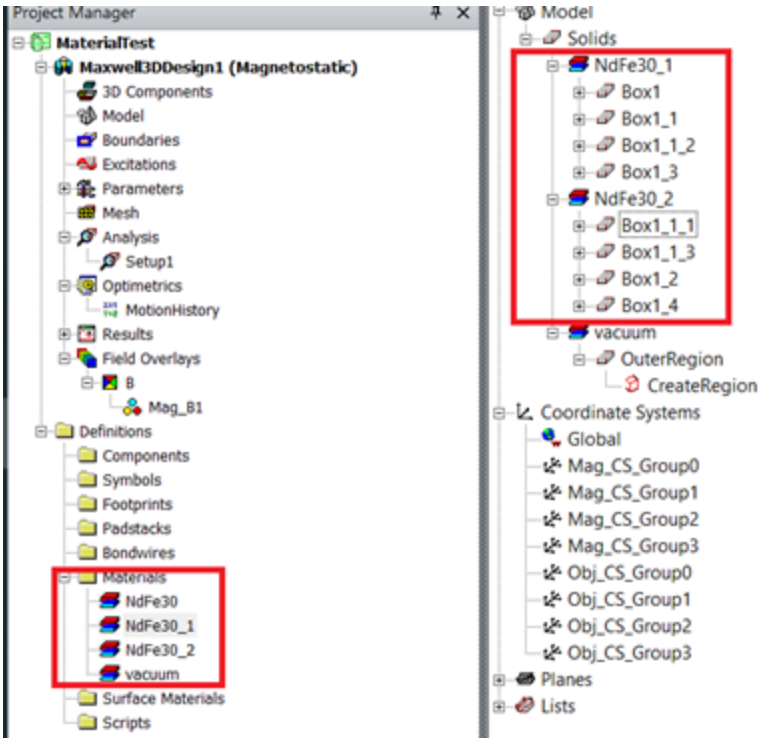
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Generate the Maxwell project.



The objects in the groups should be assigned to the corresponding materials.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Tip:** With the stand-alone Ansys Motion application, you can set up a Maxwell cosimulation on a Windows machine, and then run the simulation on a Linux machine. For more information on this capability, see Running an Ansys Motion-Maxwell Cosimulation on Linux.

Related Topics

Coupling Maxwell with Ansys Motion (Stand-Alone)

Including Transient Data in Maxwell-Motion Workbench Coupling

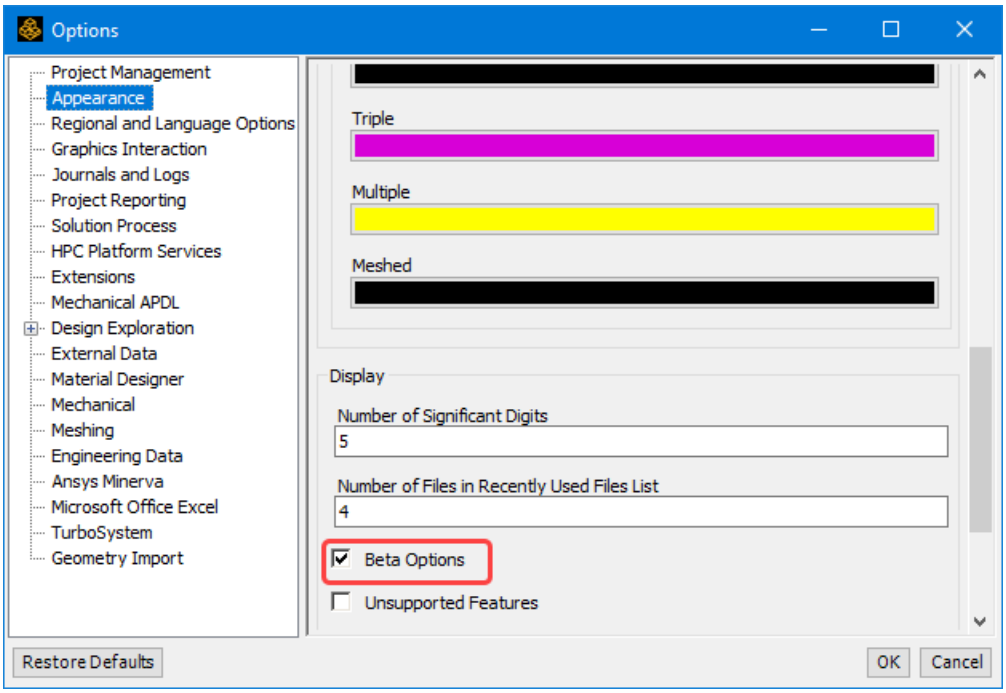
For Maxwell Transient 2D and 3D designs, users can export transient solution data from the Maxwell application to the Motion design via a Workbench link.

> **Note:** This feature is only available on Windows operating systems.

To import transient force data from Maxwell, make sure to activate the Beta flag in Workbench:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

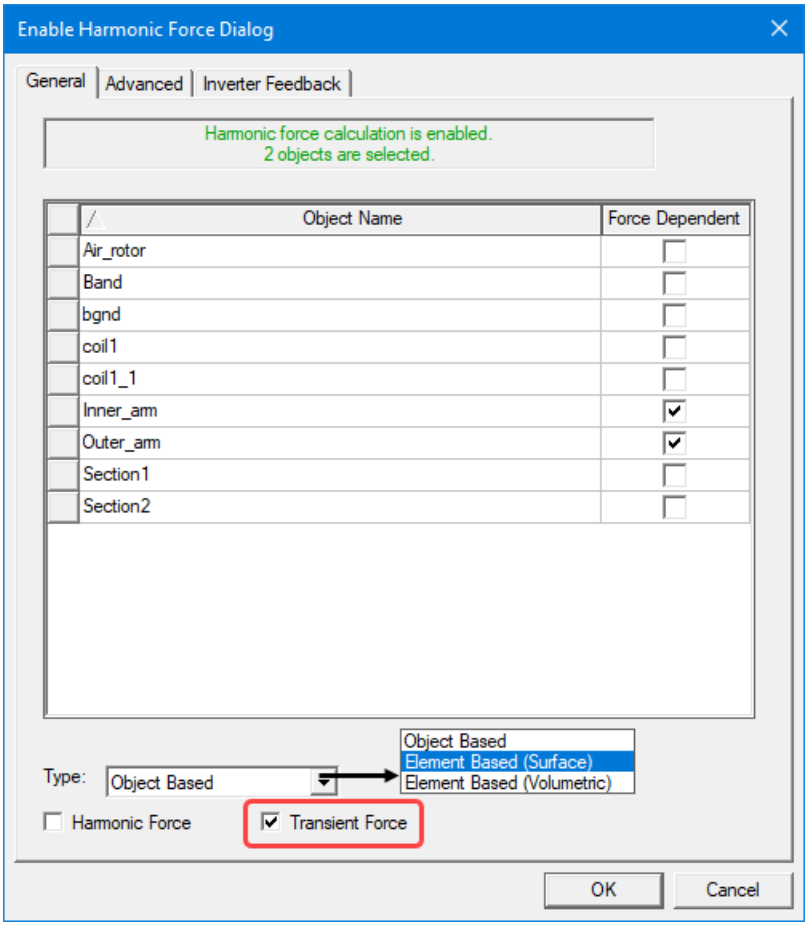
1. From the Workbench menu bar, select **Tools** > **Options** > **Appearance**.
2. Select the **Beta Options** check box.



From within the Maxwell application, enable the harmonic force calculation and select the parametric study (if any):

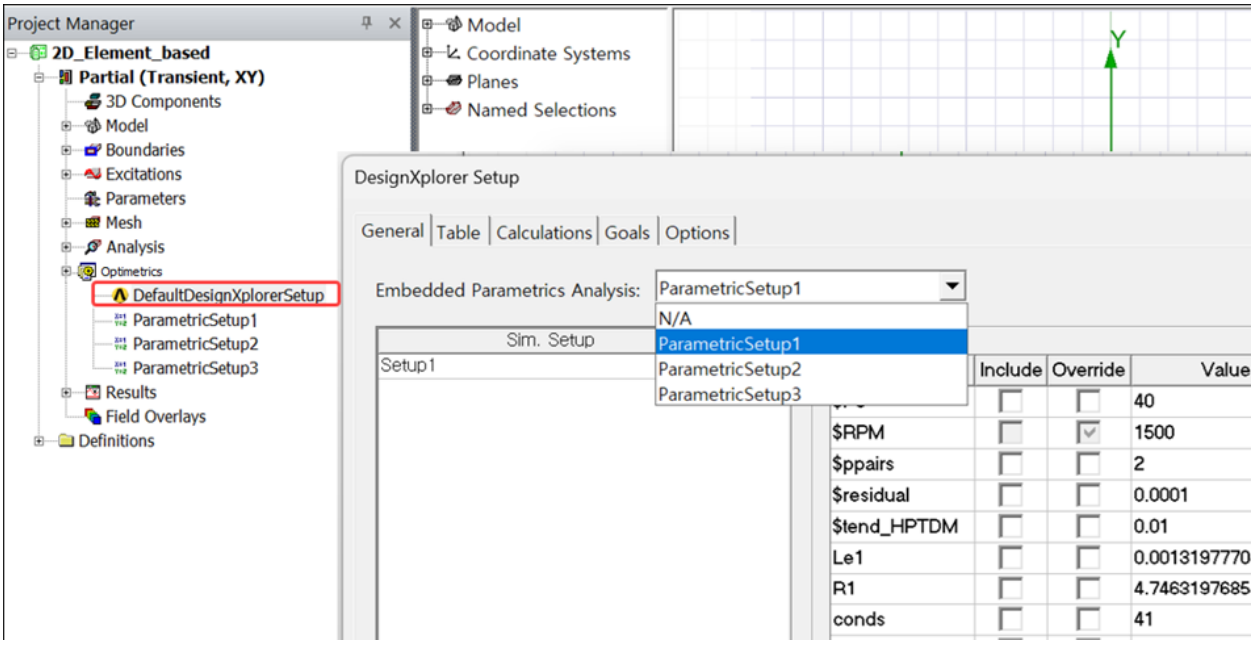
1. In the Project Manager tree, right click the design, and select **Enable Harmonic Force Calculation**.
2. In the **Enable Harmonic Force Dialog** window that opens, select **Transient Force** and the desired force data type for Maxwell-Motion WB Coupling. Force calculation can be **Object Based**(2D and 3D), **Element Based (Surface)** (2D) and **Element Based (Volumetric)** (2D).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

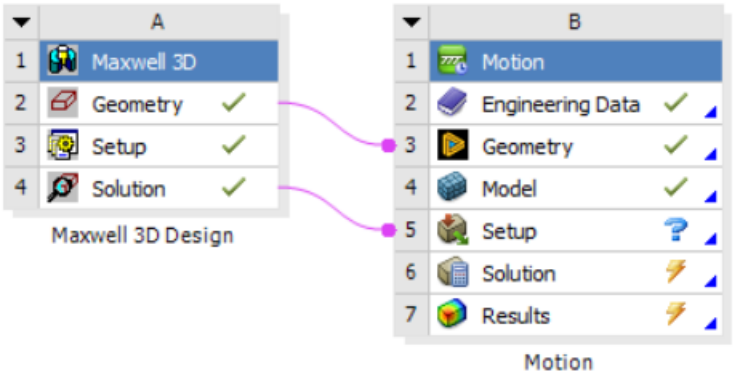


3. If there are Parametric Setups in the Maxwell design, **double click** the **DefaultDesignXplorerSetup** entry (added automatically when the design is set up in the Workbench application) under the **Optimetrics** branch of the Project Manager tree, and select the parametric setup to be used in the Workbench application.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

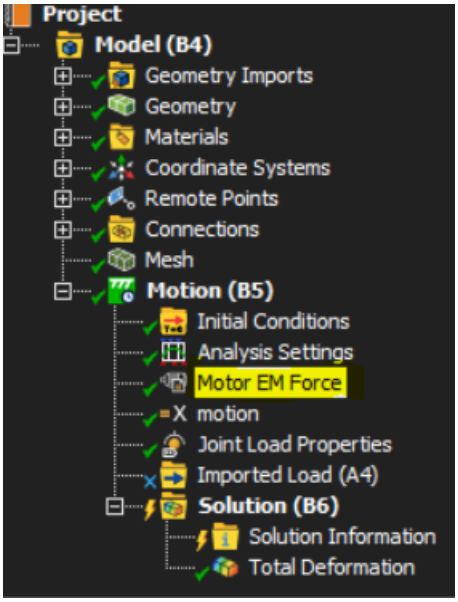


The Maxwell and Motion Workbench configurations are shown below:

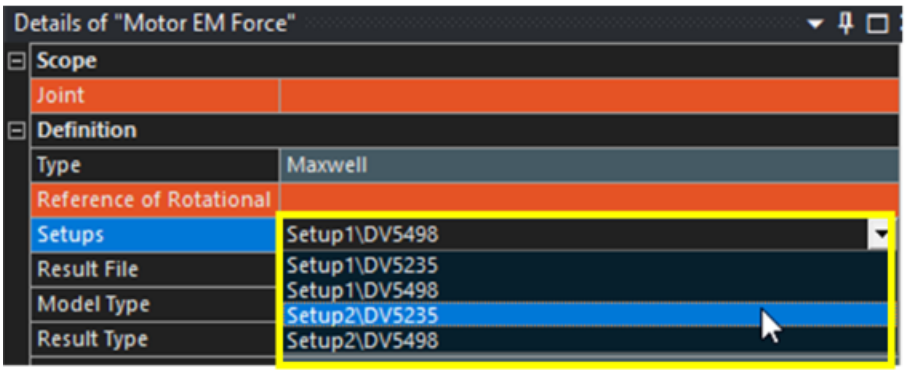


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Motor EM Force is created automatically through a Workbench link:



You can select the target setup and variable version for Motor EM Force if there are multiple nominal setups and variables:

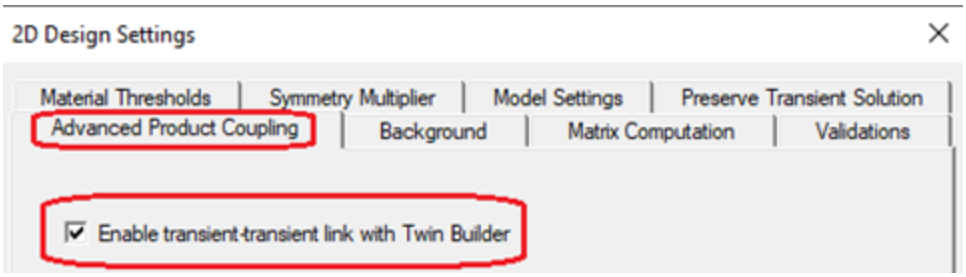


Coupling Maxwell to a Twin Builder Component

Maxwell designs may be dynamically coupled to Twin Builder components through the Twin Builder user interface. The Twin Builder interface allows Maxwell 2D, 3D, and RMxprt designs to be linked as dynamic components. Source windings set to **External** are available as conservative pins in Twin Builder.

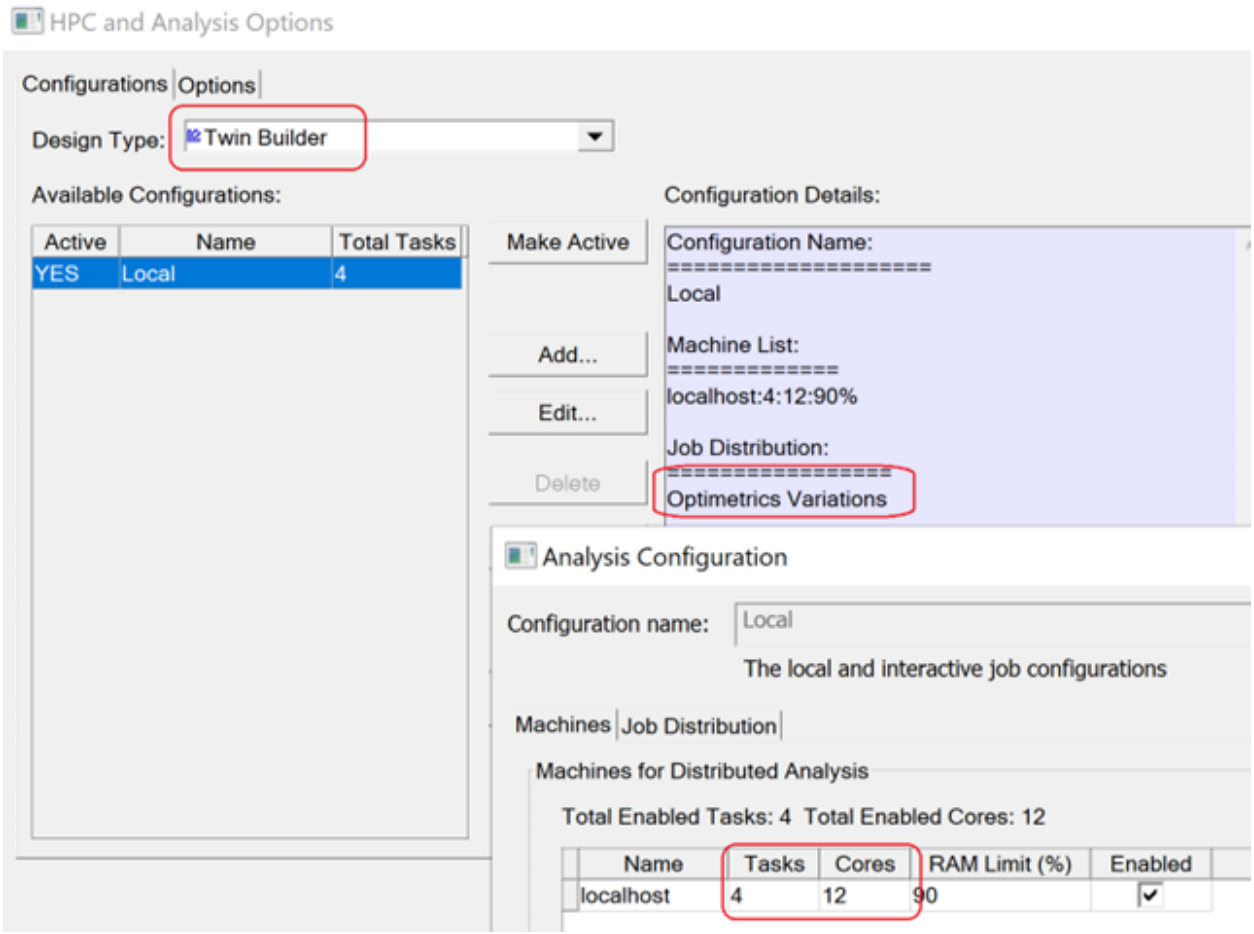
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Transient coupling links must be enabled in Maxwell on the **Design Settings** > **Advanced Product Coupling** tab for Transient Cosimulation with Twin Builder.



For parametric HPC solve of Maxwell 2D and 3D transient coupling designs, HPC setup must be done on the Twin Builder side. Maxwell will use the same HPC setup from Twin Builder; and any HPC settings on the Maxwell side will be ignored. The Twin Builder HPC setup includes user inputs for Number of tasks and Number of cores. Specify the task number to be greater than 1; and the core number to be equal or greater than the task number. Only one level HPC (Optimetric Variations) is supported. OpenMP is supported. For OpenMP, the number of cores divided by number of tasks should be greater than 1 (multiple cores per task).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:** A variable, **TBOptiVariation**, will be added in the Maxwell design for such cosimulation. A warning message will display when this variable is added to the Maxwell design for the first time. Any changes in the Twin Builder design will reset the value of the variable to 0 and Maxwell solutions will be invalidated.

The ECEIM_Model component must be used in a Maxwell AC Magnetic design to enable Twin Builder to couple to the design.

Dynamically linked components provide the ability to have Twin Builder vary a variable in a Maxwell design and have Maxwell solve the design at the new design point and provide the solution data back to Twin Builder for use in the circuit simulation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

For more information on coupling Maxwell designs to Twin Builder components, refer to the *Maxwell Coupling Components* and *RMxprt Dynamic Coupling Component* topics in the **Twin Builder Help**.

For more information on setting up HPC in Twin Builder with a coupled Maxwell design, refer to the *Maxwell Coupling Components* and *Setting HPC and Analysis Options* topics in the **Twin Builder Help**.

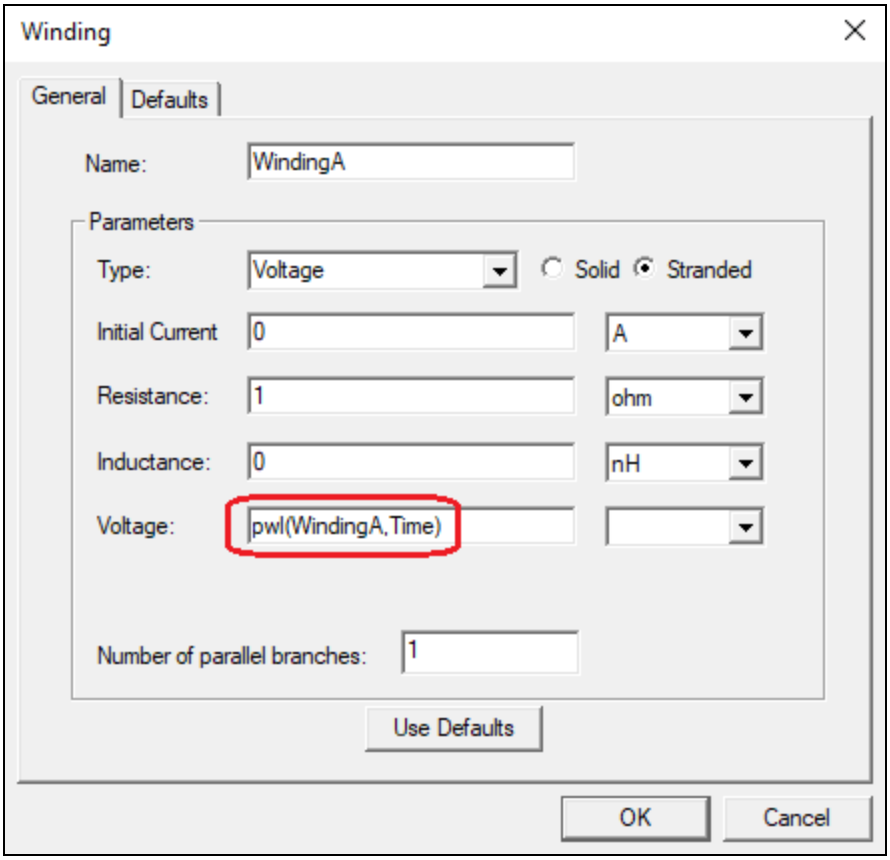
Coupling Maxwell to a Twin Builder Excitation Component

You can couple a Maxwell 2D or 3D transient, or a 3D A-Phi transient design using the Twin Builder **Excitations Component**. The Excitations component user interface allows you to assign output quantities in the Twin Builder design to windings in the coupled Maxwell design. Based on Maxwell excitations (voltage/current), Twin Builder will perform simulation, then generate detailed PWL expressions and dataset data. These expressions and data can then be pushed back to Maxwell as excitations for further analysis.

Refer to the *Maxwell Excitation Component* section in the **Twin Builder** help for the procedures on coupling a Twin Builder **Excitations Component** to a Maxwell 2D or 3D transient, or a 3D A-Phi transient design; and sending **Push Excitations** data back to the coupled Maxwell design.

After the excitations push-back, you can see the push-back excitation expression in Maxwell. For example, for winding voltage:

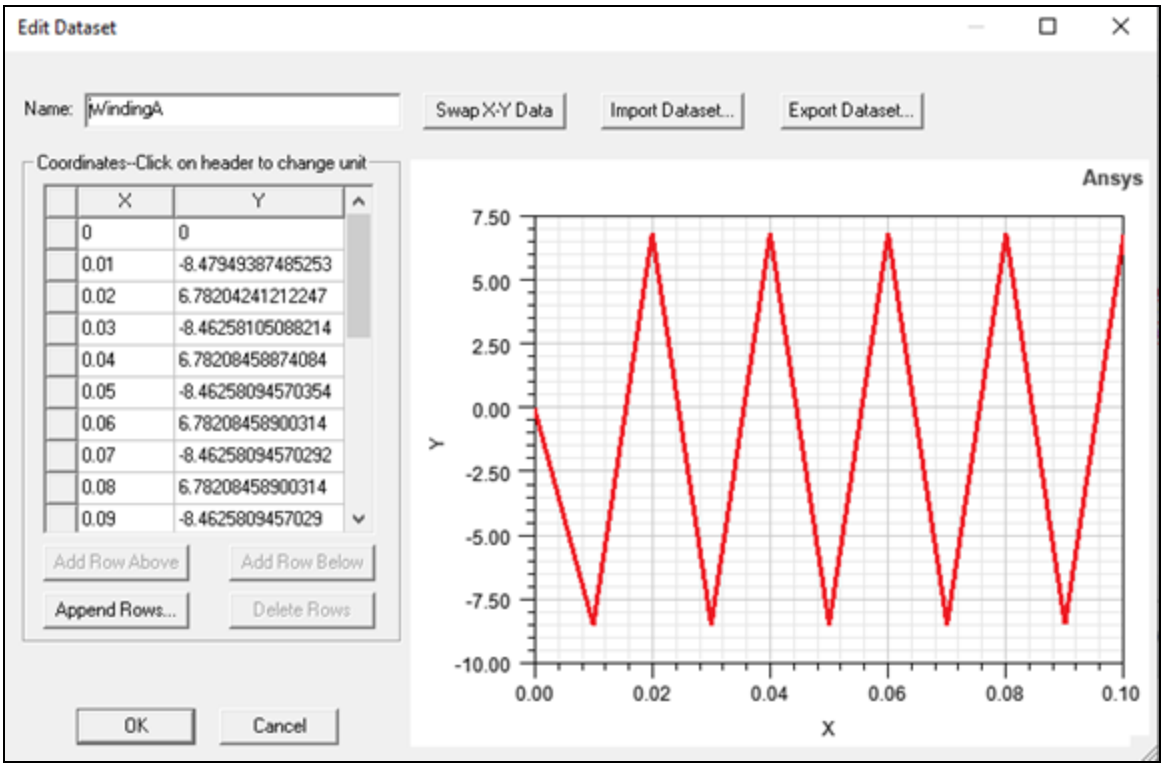
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The dataset for the PWL function can be viewed by selecting **Maxwell** > **Design Datasets**. In the Dataset window, you can see the data set details.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To view the data set in a table, click **Edit**.



Coupling Magnetostatic and AC Magnetic Designs with Ansys Fluent

Coupling is supported between Ansys Fluent and Maxwell 3D magnetostatic and AC magnetic designs. For example, a transient Fluent simulation can use a Maxwell solution to simulate the magnetic effect of the electric arc of contacts. This coupling supports designs that have a datalink defined, such as mesh, demagnetization, and so on. A design that is enabled for thermal and stress feedback is not supported.

In Ansys Fluent, during a transient simulation that requires exchanging data with Maxwell, a user-defined function (UDF) invokes the Maxwell solver. Ansys Fluent and Maxwell exchange inputs and solution data in a coupling directory that you specify. Maxwell calculates the B field based on the mesh-based conductivity (and meshed-based permeability in AC magnetic designs); and current excitation provided by Ansys Fluent. Ansys Fluent calculates flow field, temperature, and feedback electrical conductivity (and permeability in AC magnetic designs) to Maxwell. The Maxwell software does not provide direct access to the data in this coupling directory.

Follow these steps to set up the coupling:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Set up the magnetostatic or AC magnetic project in Maxwell 3D.
2. In the Project Manager tree, right-click **Setup** under **Analysis** and select **Configure Fluent Conductivity Coupling**.
3. In the **Configure Fluent Conductivity Coupling** window, browse to select the coupling directory or type a path, and click **OK**. You can enter a local directory, such as where Maxwell desktop is running, or you can enter a mapped network location.
    - If you enter an existing directory that contains files, a warning notifies you that all files in that directory will be permanently deleted. Click **Yes** to use that directory and delete all files contained in it. Click **No** if you do not want to select that directory. When you click **No**, the dialog box stays open for you to enter a different directory.
    - If the directory you entered does not exist, Maxwell creates the directory. If the directory is not successfully created, the dialog remains open.

    Note:
    - If there are errors during the process or if you terminate the process, the coupling directory is not configured.
    - If there is version incompatibility between Ansys Fluent and Maxwell, the process is stopped and an error message is displayed.
    - If you Clean Stop the process and there is enough solution data to configure the directory, the coupling directory will still be configured.
4. Set up the Ansys Fluent project as described in the *Ansys Fluent User's Guide*.
5. Build and load the UDF that was developed for your application. For more information about UDFs, see the *Ansys Fluent UDF Manual*.
6. In the **input.txt file** that is specific to the UDF, provide the information that is required by the UDF. For more information about the required information and format, see Sample Input.txt File Format.

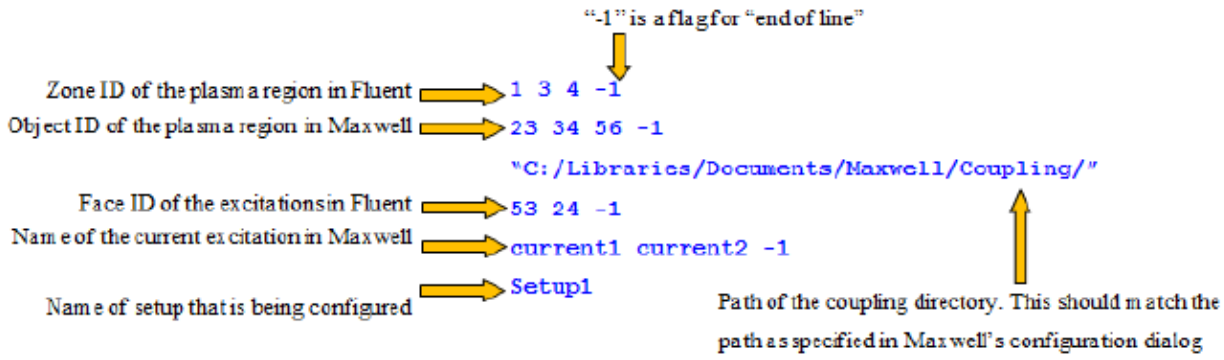
The UDF is invoked during the transient simulation. When there is a need to exchange data, the UDF invokes the Maxwell solver.

Sample Input.txt File Format

The **input.txt file** is used by the UDF during the coupling between Ansys Fluent and Maxwell.

The following example shows the format and provides information about the parts of the file.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Determining the Zone ID in Ansys Fluent

You can determine the Zone ID in Ansys Fluent as follows:

1. In the navigation pane under **Solution Setup**, select **Cell Zone Conditions**.
2. Select a zone on the task page.

The ID of the zone you selected is displayed in the ID text box.

Determining the Face ID in Ansys Fluent

You can determine the Face ID in Ansys Fluent as follows:

1. In the navigation pane under **Solution Setup**, select **Boundary Conditions**.
2. Select a boundary on the task page.

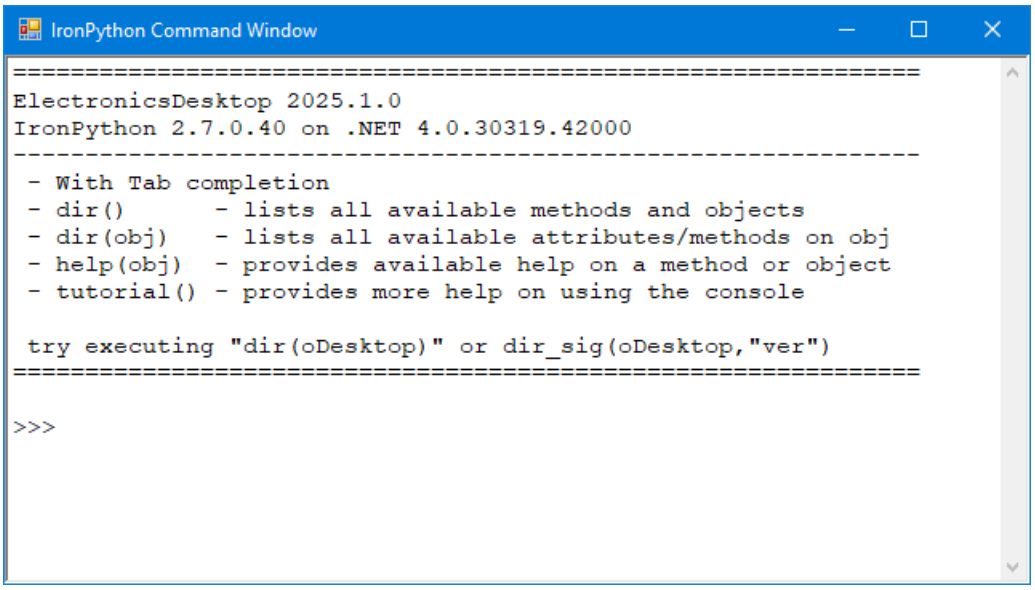
The ID of the face assignment is displayed in the ID text box.

Determining the Object ID in Maxwell

You can determine the Object ID in Maxwell as follows:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Tools** > **Open Command Window**.
2. In the **IronPython Command Window**, query the object ID by its name.



For example:

You can use `oEditor.GetNumObjects()` to find the total number of objects; and then use `oEditor.GetNumObjects(`*i*`)` to return the name of *i*th object.

Magnetostriction in Maxwell Magnetostatic to Workbench Mechanical Coupling

Maxwell supports magnetostriction and inverse magnetostriction forces in Workbench Mechanical coupling for Maxwell 2D/3D magnetostatic designs. Both one-way and two-way coupling are supported.

One-Way Coupling

The Maxwell solver calculates force density, which includes magnetostriction force data. The force density data including magnetostriction force data is output to the Workbench Mechanical system.

Two-Way Coupling

- Maxwell provides force density data to the mechanical system as it does for one-way coupling.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The Workbench Mechanical system solves and sends stress/strain data (plus displacement data) to Maxwell, which will be used in the Maxwell solution for the next iteration.

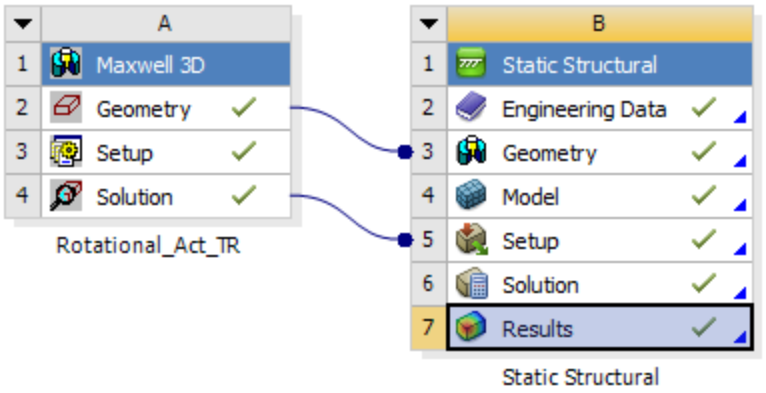
Magnetostriction Stress Coupling Between Maxwell 2D/3D Transient and Workbench Mechanical

Maxwell supports magnetostriction forces in two-way coupling between Workbench Mechanical and Maxwell 2D/3D Transient designs.

- The Maxwell transient solver provides force density data to the Mechanical system.
- The Workbench Mechanical system solves and sends stress/strain data (plus displacement data) to Maxwell, which will be used in the Maxwell solution for the next iteration.

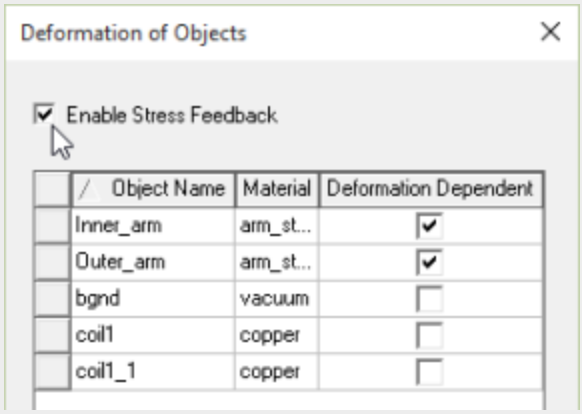
To analyze the effect of stress due to magnetostriction forces on objects in a design using two-way coupling between a Maxwell 2D/3D transient design and Workbench Mechanical:

1. Set up the Maxwell 3D/2D transient Mechanical structure coupling.

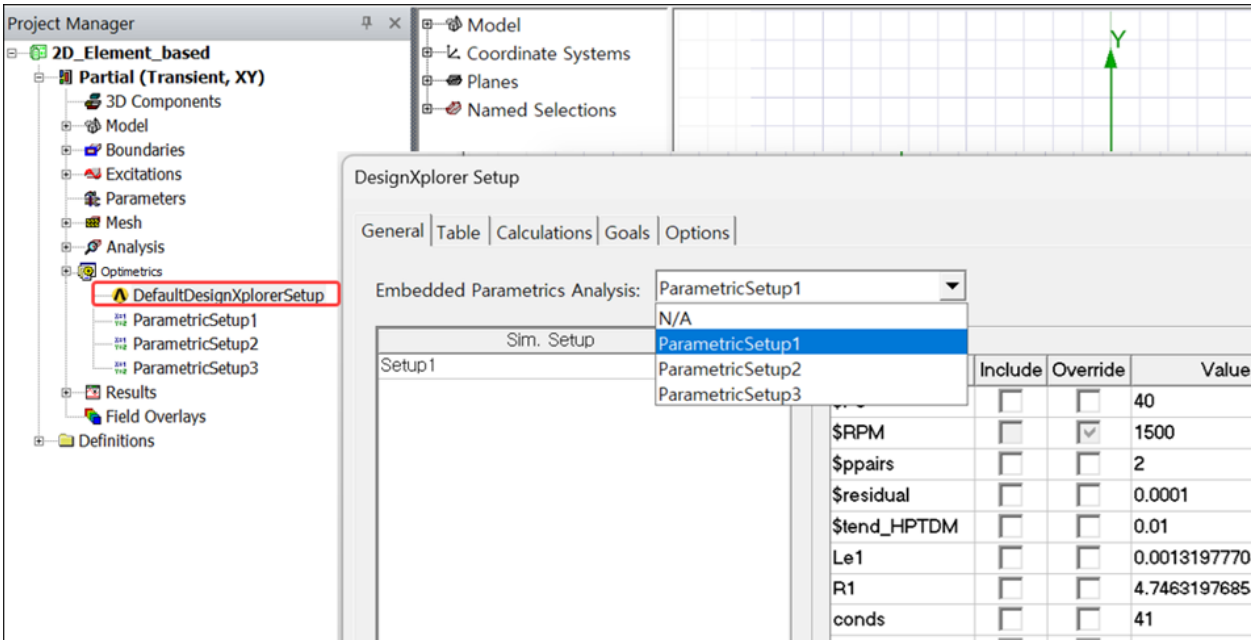


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: You must manually Enable Stress Feedback and select the desired stress feedback objects in the Maxwell design.

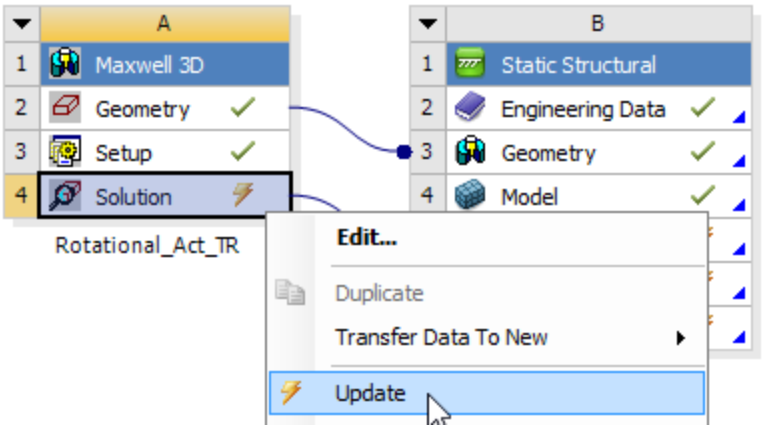


2. If there are Parametric Setups in the Maxwell design, **double click** the **DefaultDesignXplorerSetup** entry (added automatically when the design is set up in the Workbench application) under the **Optimetrics** branch of the Project Manager tree, and select the parametric setup to be used in the Workbench application.



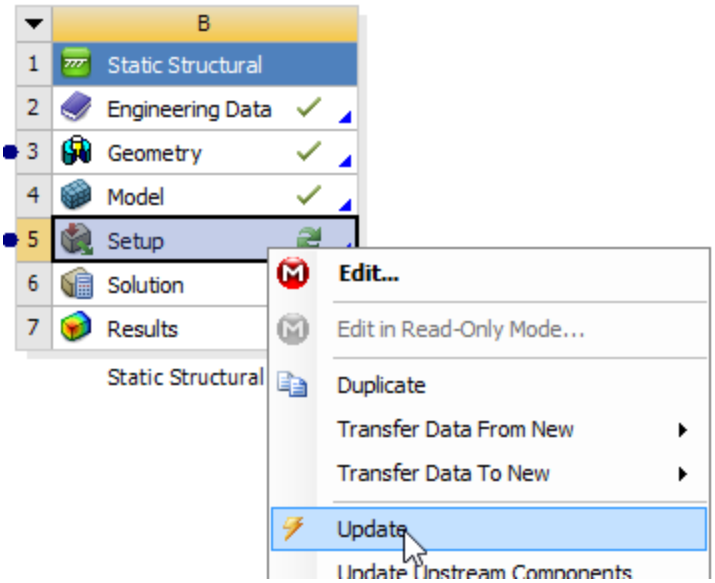
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. **Update** the Maxwell **Solution** cell to initiate a Maxwell solve.



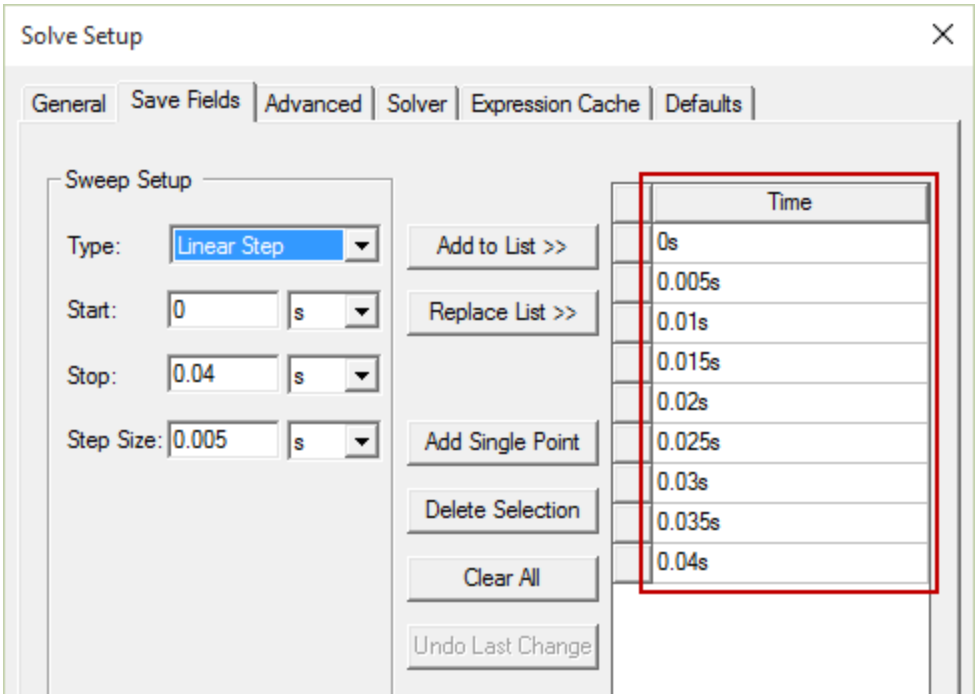
The Maxwell transient solver generates force density data, which can be used by Workbench Mechanical.

4. **Update** the Workbench Mechanical **Setup** cell to import the force density data generated by Maxwell.



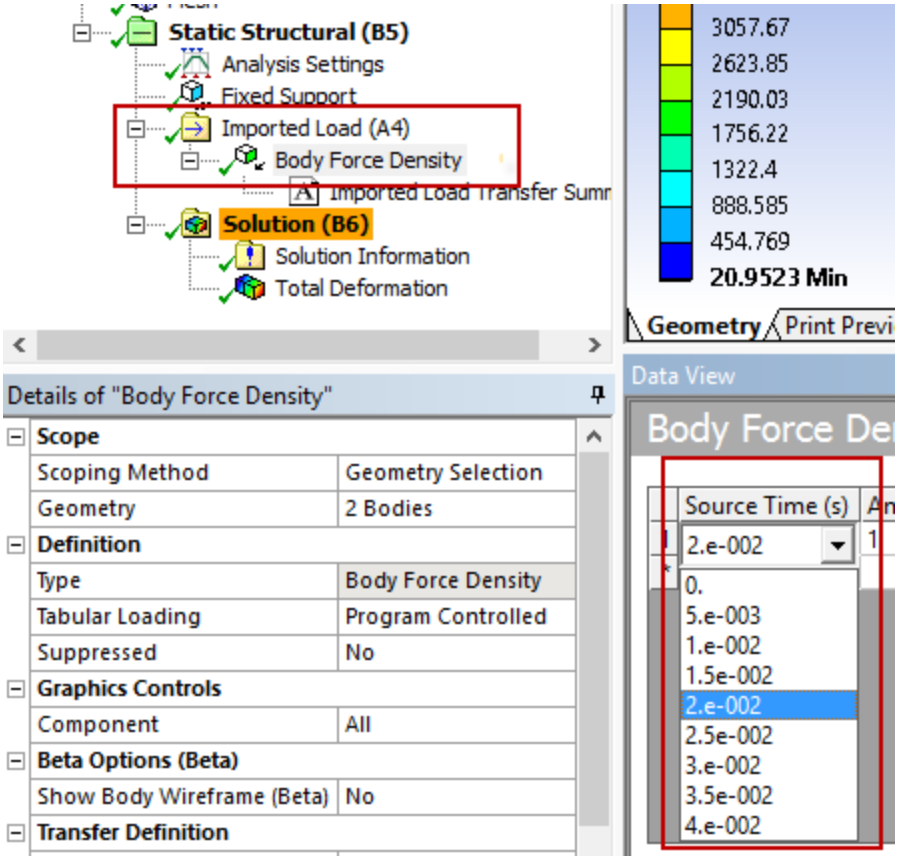
Maxwell maps the Maxwell mesh to the Workbench Mechanical mesh; and the calculated force density values for the time points you set up on the Maxwell transient **Solve Setup** window's **Save Fields** tab are loaded into Workbench Mechanical.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



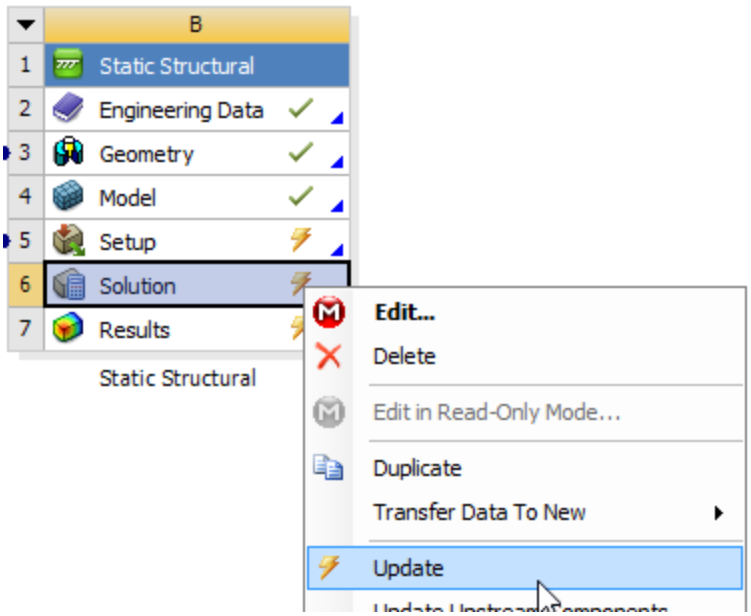
5. In Workbench Mechanical, select the desired source time point you want to use for the force density calculations.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. **Update** the **Solution** cell of the Mechanical panel.

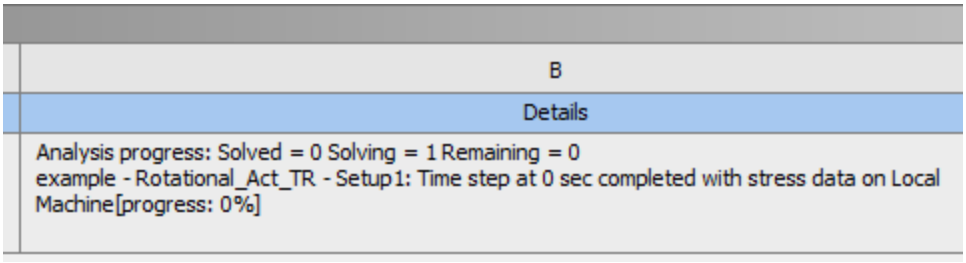
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Workbench Mechanical solves the structural project using the force density at the selected source time, then outputs displacement and stress/strain data to be used as feedback by the Maxwell transient solver.

7. To re-solve using the feedback data, set the **Solution** cell of the Maxwell panel to **Enable Update**, then perform an **Update** for the Solution cell. (You can also use the feedback iterator to automatically update the Maxwell Solution cell.)

   The Maxwell 3D/2D transient solver uses the feedback displacement and stress/strain data to re-solve from time 0.

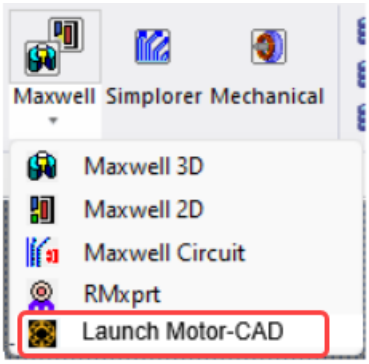


Launching Motor-CAD from Ansys Electronics Desktop

The Ansys Motor-CAD application can be launched from the Ansys Electronics Desktop interface. Access to the Motor-CAD application is available from the **Desktop** tab's Maxwell

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

icon:



This menu will launch the Motor-CAD application that is registered via the Automation Registration interface. To view and/or edit the Motor-CAD version launched, please refer to the section "Automation and Scripting » Automation Registration" in the Motor-CAD manual.

Coupling between Maxwell and Motor-CAD is set up within the Motor-CAD Lab module; see the Motor-CAD documentation for more details.

System Coupling Analyses Using Maxwell

Ansys System Coupling™ facilitates and manages the execution of coupled simulations between multiple physics solvers, coupling active cosimulation participants using either System Coupling's graphical user interface (GUI) or its command-line interface (CLI).

The Maxwell application can engage as a coupling participant in thermal cosimulations with Ansys Fluent® and Ansys Mechanical™ applications and in electrical arc modeling simulation with Ansys Fluent, Ansys Motion™, and Ansys Twin Builder® applications. System Coupling provides enhanced control over coupled simulation processes, including automatic starts and restarts for participants, the ability to manipulate the System Coupling data model, an interactive solution workflow, and multiple output formats for reviewing and postprocessing results.

In System Coupling analyses, each participant completes its physics and coupling setups in its own user interface generating a Participant Setup (.scp) file along with its solver file(s). The coupled analysis itself includes loading participants, setting values for coupling-related settings, specifying coupled analysis settings, and starting the run. Once the solution has started, System Coupling directs the execution of the coupled analysis – including general coupling management, participant communications, and transfers of data between participants.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: For detailed information on System Coupling in general, refer to the System Coupling User's Guide. For details on working in System Coupling's GUI or CLI, refer to the Using System Coupling's User Interfaces section.

Note: For examples of Maxwell participating in cosimulations, review the coil-and-core induction heating, bar-and-coil induction heating, bus bar, and electric motor examples in System Coupling Tutorials. Also see Electrical Arc Modeling Using System Coupling.

Supported Capabilities and Limitations

The following capabilities are supported for System Coupling analyses with Maxwell:

- Multi-region data transfers for Maxwell participants.
- Maxwell 3D AC Magnetic, Maxwell 3D Transient, and Maxwell 2D Transient Cartesian, XY geometry mode solutions.
- Update frequency controls for a Maxwell participant running a steady solution.
- Automatic and manual distribution parameters for parallel execution
- Dynamic remeshing and remapping during the execution of Maxwell 3D AC Magnetic simulations with time-dependent motion or excitation defined. For details, see Mapping Process in the System Coupling User's Guide.
- Variable displacement for simulations with time-dependent motion defined.
- Thermal data transfers on bodies. EM loss can be provided to other coupling participants, and temperature can be received from them.
- Motion data transfers on coupled bodies.
- Parallel/distributed processing.
- Electrical arc modeling:
    - System Coupling between the Maxwell, Fluent, and Twin Builder applications can be used to model an electric arc in a static design.
    - System Coupling between the Maxwell, Fluent, and Motion applications can be used to model an electric arc in a device with motion.

The following are known limitations when using System Coupling with Maxwell:

- Only Maxwell 3D AC Magnetic, Maxwell 3D Transient, and Maxwell 2D Transient, Cartesian, XY geometry mode solution types are supported.
- For electrical arc modeling, only Maxwell 3D magnetostatic designs are supported.
- Maxwell 3D AC Magnetic solution type is limited to a single frequency.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Variable velocity is not supported for designs with time-dependent motion and/or excitations.
- Update frequency controls are not supported when Maxwell runs a solution with time dependencies.

Using HPC Distribution Settings for Coupled Analyses

System Coupling supports the use of Maxwell's automatic and manual HPC distribution parameters for distributed coupled analyses. Each of these **Execution Control** settings corresponds to one of Maxwell's distributed analysis parameters.

- **Auto Distribution Settings**: Whether job distribution is performed automatically.

When auto-distribution is disabled, the following manual settings are available:

- **Include HPC Distribution Types**: List of the HPC distribution types to be included for distributed runs.
- **Number of Cores Per Task**: Number of HPC cores to be used per task.
- **Batch Options**: Batch option arguments to be applied to the run.

For more information, see *Using AEDT HPC Distribution Options* in the *System Coupling User's Guide*.

Variables Available for System Coupling

The following variables will be available on convective boundaries:

Name / Internal Name	Transfer Direction	Data Type	Physical Type
Loss / Loss	Output	Scalar	Heat Rate
Temperature / Temperature	Input	Scalar	Temperature

The following variables will be available for magnetostatic simulations:

Name / Internal Name	Transfer Direction	Data Type	Physical Type
Arc Voltage	Output	Scalar	Voltage
Arc Current	Input	Scalar	Current
Loss / Loss	Output	Scalar	Heat Rate
Lorentz Force	Output	Vector	Force
Electrical Conductivity	Input	Scalar	Electrical conductivity
Temperature	Input	Scalar	Temperature

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

System Coupling Related Settings

System Settings

To enable coupling with Maxwell, the environment variable **ANSYSEM_ROOT252** must be set to the location of your Ansys Electromagnetics installation.

> **Note:** The environment variable name is ANSYSEM_ROOT<*major*><*minor*>, where <*major*> is the two digit major version number (the last two digits of the year); and <*minor*> is the one digit minor version number. For example, for the Ansys Electronics 2025 R1 release, the environment variable name is ANSYSEM_ROOT252. For the Student Version of the software, the variable name is ANSYSEM_ROOTSV<*major*><*minor*>.

- For Windows, this environment variable is automatically set during the installation of the Ansys Electromagnetics Suite (for example, *C:\Program Files\ANSYS Inc\v252\AnsysEM*).
- For Linux, you must set this environment variable manually (for example, */opt/ansys_inc/v252/AnsysEM*).

> **Note:** The environment variable must be set on all hosts if the analysis is distributed to multiple hosts. For Windows, this environment variable must generally be set as a System environment variable (not a user environment variable) for distributed analysis so that it is set for the Ansoft RSM Service. For Linux, the user must ensure that the environment variable is set for the AnsoftRSM Service daemon processes.

You must also integrate Maxwell with Ansys: from the Start menu, select **Ansys EM Suite 2025 R2** > **Modify Integration with Ansys 2025 R2**.

Maxwell Settings

Solution Type:

Must be set to one of the following:

- **AC Magnetic** for Maxwell 3D designs only.
- **Transient** for either Maxwell 3D or Maxwell 2D designs. For Maxwell 2D designs, you must also specify the **Cartesian, XY** geometry mode when selecting this solution type.
- **Magnetostatic** for Maxwell 3D designs only.

Materials:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Bodies to receive temperature data must have temperature-dependent properties. Note that when the same materials are used in multiple participants, common properties (e.g., conductivity) must be consistently defined.

Motion:

Motion must be appropriately and consistently defined for all bodies involved in the analysis.

Stationary bodies must not move from the reference position or participate in multiple reference frames during the cosimulation.

Motion should be defined with a (rigid body) displacement.

For more information, see Additional Participant Setup Considerations in the *System Coupling User's Guide*.

Creating a System Coupling Setup

The **System Coupling Setup** enables the exchange of data between participants by creating an interface between regions on the participant models.

To create a coupling setup, select (**Optimetrics** > **Add** > **System Coupling Setup**), and do one of the following:

- For AC Magnetic designs, select the desired solution **Setup**, **Frequency**, and input/output quantities. For the **Temperature** quantity, click the **Object temperature** button to open the **Temperature of Objects** dialog in which you can choose to **Include Temperature Dependence**, and to set the temperatures for model objects composed of materials having temperature-dependent properties.
- For AC Magnetic designs with time-dependent motion, create a dataset to define the position of the moving body, and design variables for the coupling time as provided by System Coupling and the displacement vector of the body.
- For 3D Magnetostatic designs:
    - For Maxwell\Twin Builder coupling, input a current excitation; the voltage value output to Twin Builder will be calculated from loss/current.
    - For Maxwell\Fluent system coupling, input temperature, system coupling time, and electrical conductivity.
    - For an example of system coupling involving the Maxwell, Twin Builder, and Fluent applications, see Modeling an Electric Arc in a Run Rails Design.
    - For an example of system coupling involving the Maxwell, Motion, and Fluent applications, see Modeling an Electric Arc in a Device with Motion.
- For Transient designs, select the desired solution **Setup**, **Start time**, **End time**, and input/output quantities. For the **Temperature** quantity, click the **Object temperature** button to open the **Temperature of Objects** dialog in which you can choose to **Include**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Temperature Dependence, and to set the temperatures for model objects composed of materials having temperature-dependent properties.

- For Transient designs with time-dependent motion/excitation:
  - Define motion on coupled bodies.

    > **Note:** A profile or expression can be used to define rigid body motion using total displacement relative to the original object location.

  - Create design variables for both the coupling time (as provided by System Coupling) and the displacement vector of the body.
  - For Transient designs, Ansys Electromagnetics time is associated with System Coupling time. Maxwell uses this time to calculate excitation and position, but itself always begins its calculations at time zero.
  - Optionally, the following inputs can also be defined as a function of System Coupling time: **Maxwell Timestep Size**, **Loss Averaging Start Time**, **Loss Averaging Stop Time/Duration**.

Generating a System Coupling Participant Setup File

System Coupling uses each participant's Participant Setup (.scp) file and the corresponding Python (.py) configuration file to load the participant and its information into the data model.

To use System Coupling, you must generate a System Coupling participant file (**.scp**). Once the Maxwell setup is complete, generate the file using either of the following options:

- In the **System Coupling Setup** window, the system coupling configuration files are generated when you click the **OK** button.
- In the **Project Manager** under **Optimetrics**, right-click your System Coupling Setup and select **Generate Configuration Files**.

In both cases, the configuration files needed by System Coupling (an **.scp** file and a **.py** file) are generated at the same time and in the same location as the Maxwell project file.

Licensing Considerations when using System Coupling

A distinct license is required for each coupling participant product, but no additional licenses are required for the System Coupling infrastructure itself.

Note, however, that Ansys EnSight is required for the postprocessing of System Coupling's interface results. If you are using a Multiphysics Licensing Bundle that does not include an EnSight license (e.g., the Ansys Mechanical Maxwell bundle), you will need a separate EnSight license to visualize cosimulation results.

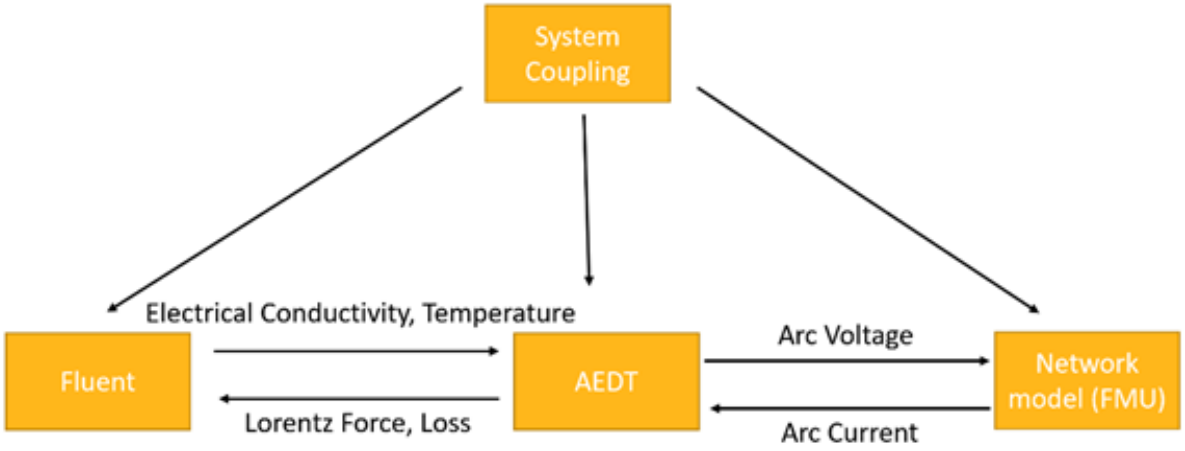
For more information on applicable licenses, see **Product Licensing Considerations** in the *System Coupling User's Guide*.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modeling an Electric Arc in a Static Device

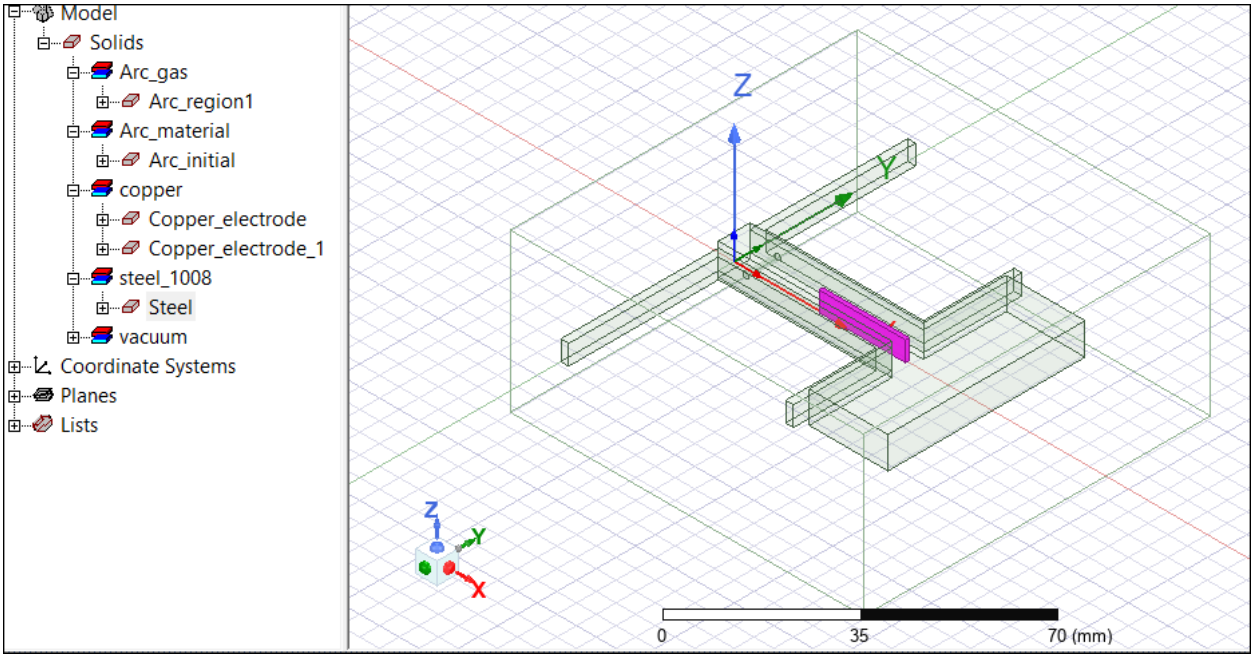
To model an electric arc in a static device, System Coupling can be used with the Maxwell, Fluent, and Twin Builder applications. All participants in the System Coupling are coupled and dependent on each other. The workflow for this type of modeling has the following order: **Twin Builder** > **Maxwell** > **Fluent**. Multiple iterations (data exchanges) can occur between each two participants.

As seen in the flow chart, Maxwell provides voltage to an external circuit in Twin Builder (through a FMU model), and then Twin Builder feeds a current to Maxwell through a circuit model. Then Maxwell generates volume loss, surface loss (including nonlinear resistance thin layer losses on Electric Arc Anode and Cathode surfaces) and Lorentz forces for Fluent. Fluent can consume heat rate (surface loss from Maxwell) on solid-fluid interfaces, and it feeds electrical conductivity, temperature, and simulation time back to Maxwell.



The model used for this example is an electric arc between two run rails. The Maxwell model is shown below:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

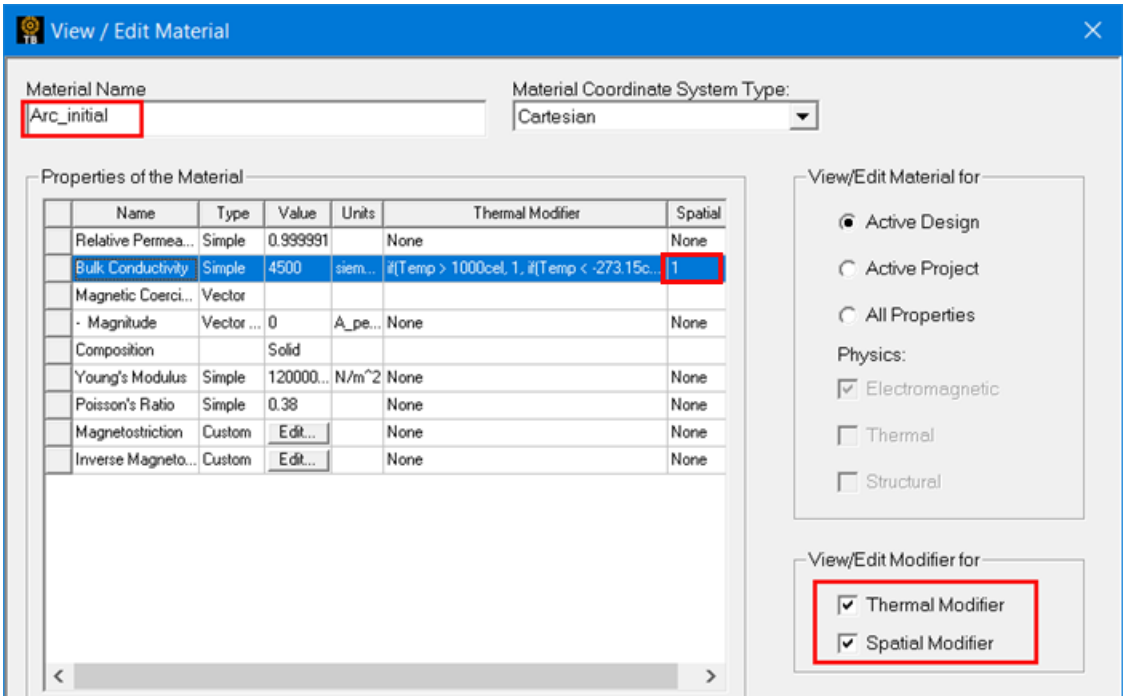


This example assumes that the user is familiar with the Twin Builder, Maxwell, and Fluent tools, so it will only detail the requirements for the System Coupling setup. For details on the individual tools, see the corresponding reference and tutorial topics.

Maxwell Setup

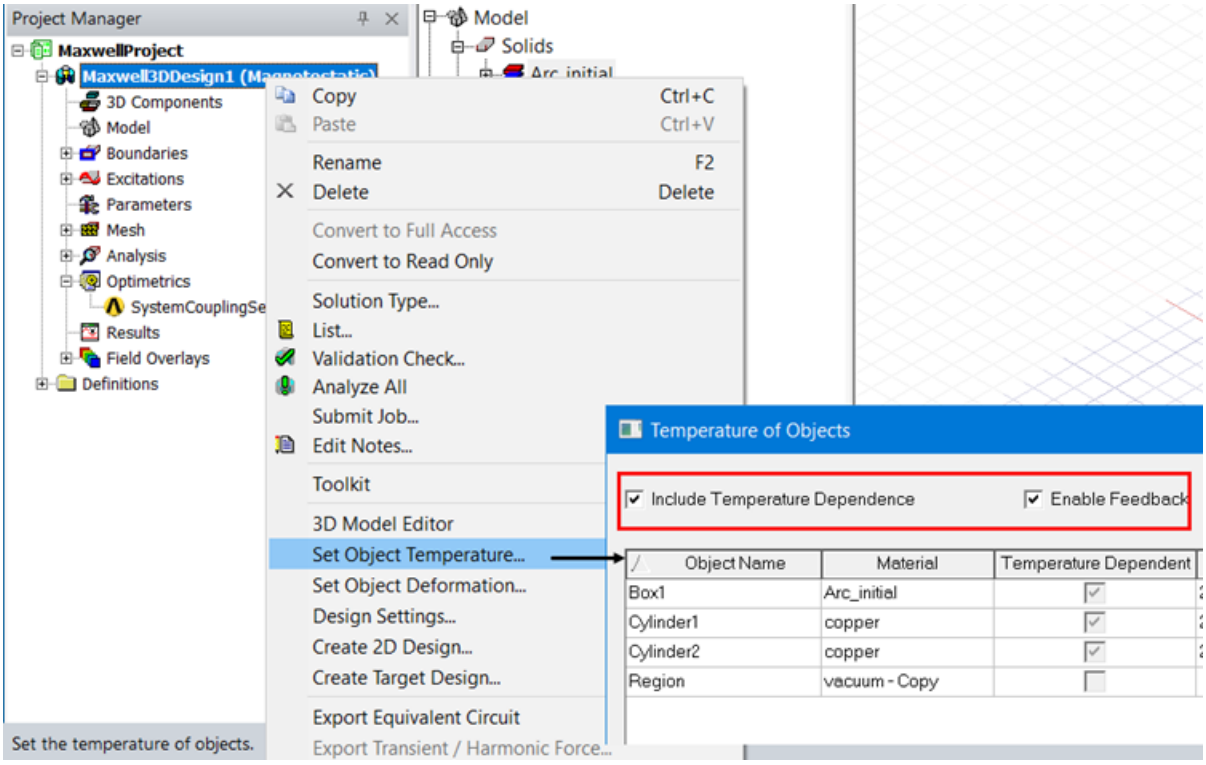
For the Maxwell model, the arc material should have the **Thermal Modifier** and **Spatial Modifier** options checked for Bulk Conductivity, and the **Spatial Modifier** should be set to **1**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

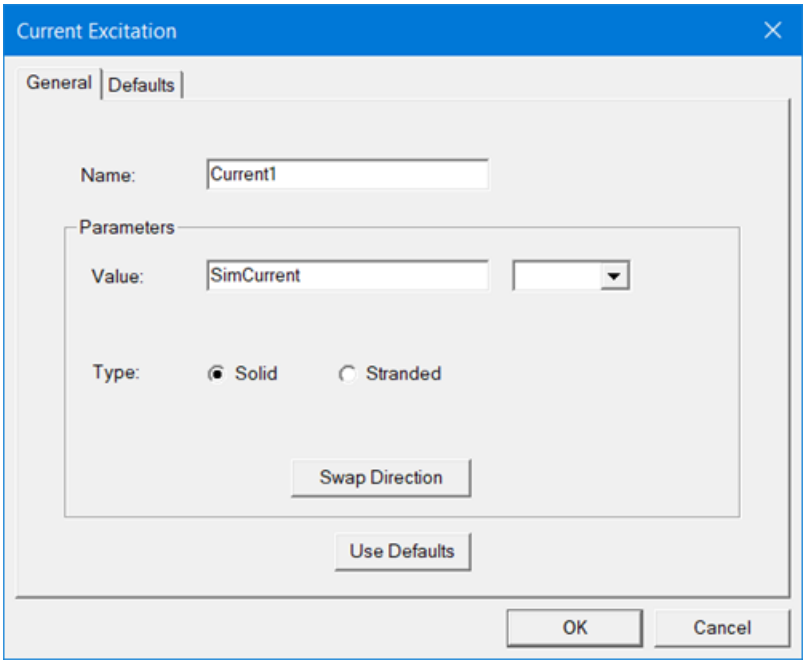


For the model Temperature of Objects, check **Include Temperature Dependence** and **Enable Feedback**:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



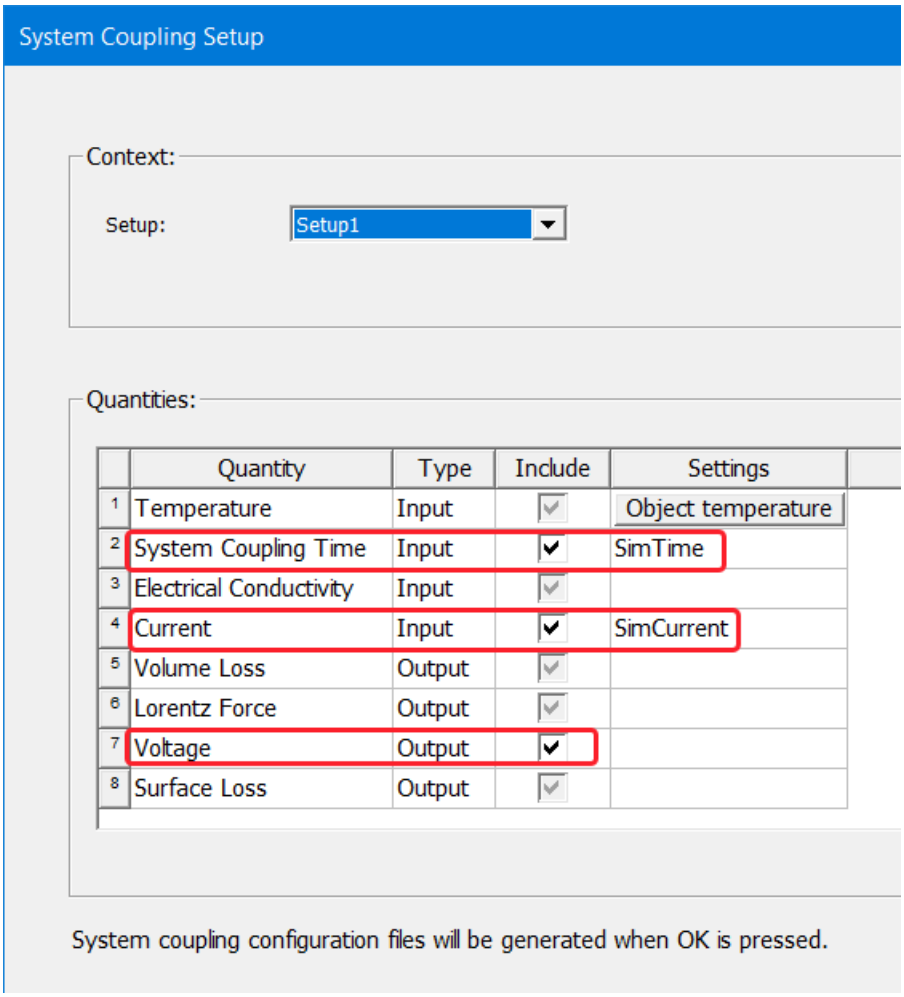
Set up the desired magnetostatic simulation, including boundaries, current excitations, and simulation time:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

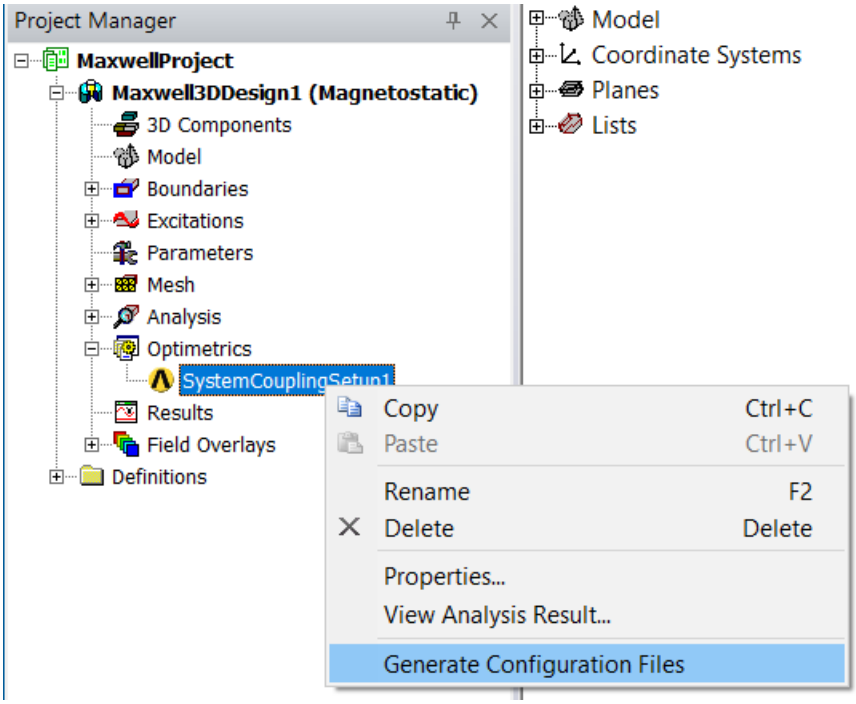
Add a System Coupling Setup to the Optimetrics folder, and set the following:

- Check **System Coupling Time**, and set the **Settings** variable to **SimTime**.
- Check **Current**, and set the **Settings** variable to **SimCurrent**.
- Check **Voltage**.



The configuration files needed for System Coupling are automatically generated when you click **OK** in the System Coupling Setup dialog box, but if you want to regenerate the configuration files without reopening the dialog box, right click on the System Coupling Setup in the Project Manager, and select **Generate Configuration Files**:

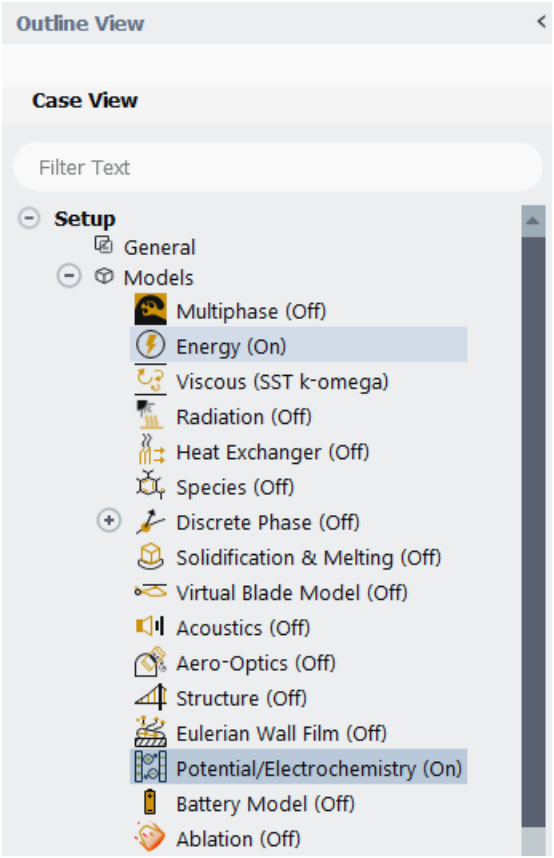
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Fluent Setup

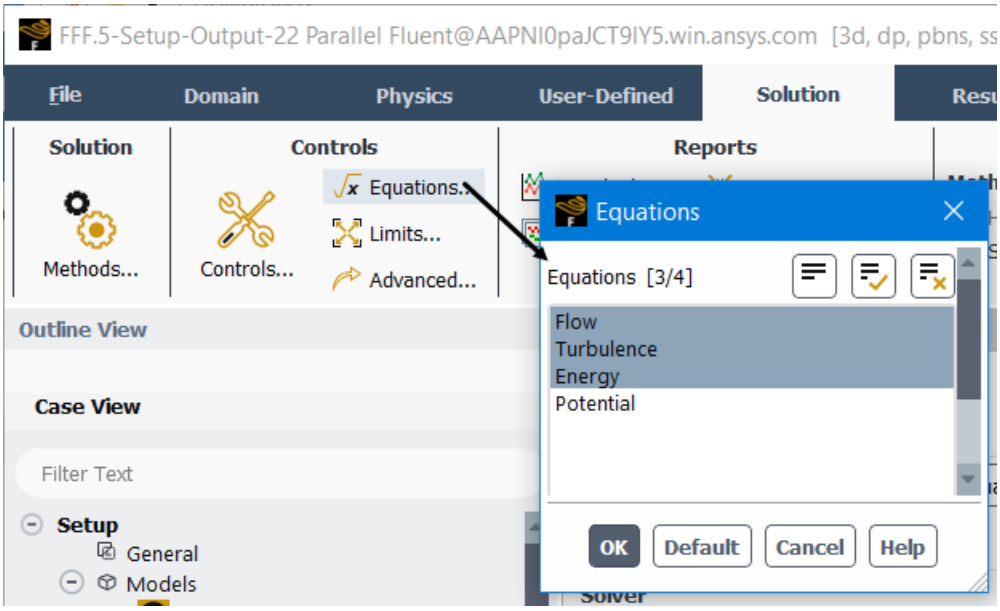
For the Fluent model, the materials should have the same properties as the Maxwell model. The Energy and the Potential/Electrochemistry models should be turned on. Potential is needed because it is used to calculate the Electrical Conductivity required by Maxwell.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

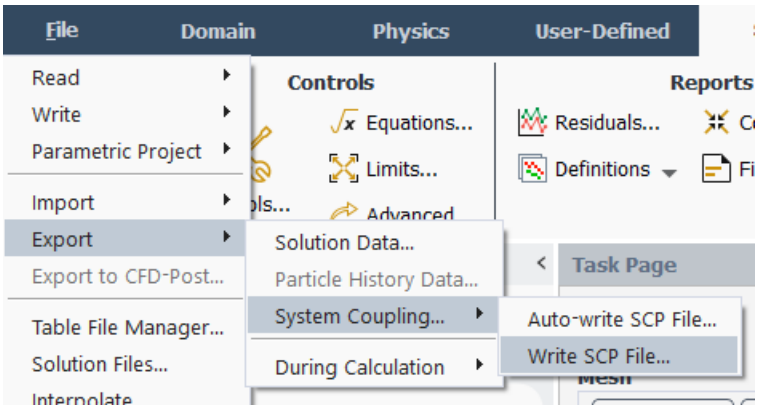


From the **Solution** panel, the **Potential** equation must be deactivated because electromagnetics will be simulated in Maxwell:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After setting any required properties and boundary conditions for your model, export the Fluent setup to a System Coupling file:



System Coupling Setup

Load the System Coupling participants one by one following the order Twin Builder FMU Model

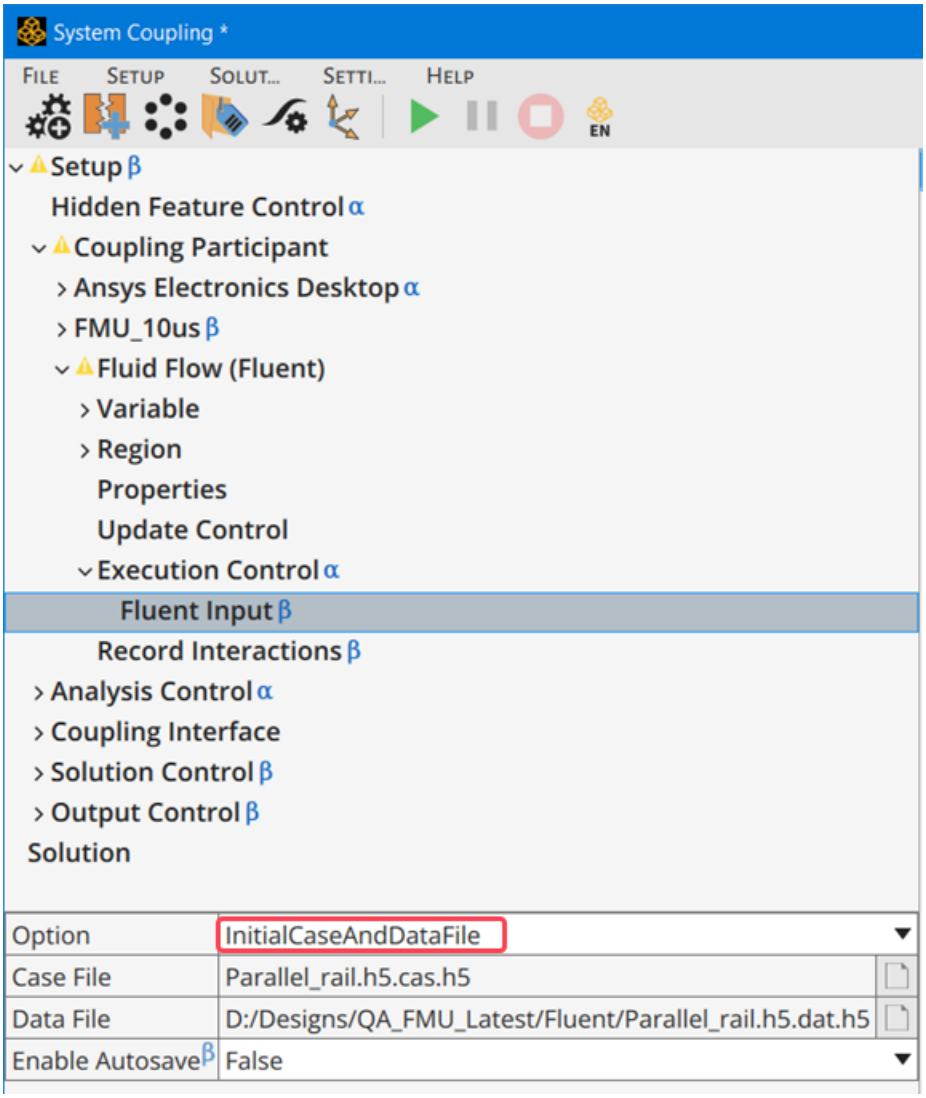
> Ansys Electronics Desktop Maxwell > Fluid Flow: click **Add Participant** icon , and in the dialog box that opens, find the input files for each participants and add one by one.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The input file for TwinBuilder is a .fmu file.
- The input file for AEDT is a \\MaxwellProject_Maxwell3DDesign_SystemCouplingSetup1.scp file.
- The input file for Fluid Flow is a \\Fluent\fluent.scp file.

Note that the order in which they were added reflects the order in which they have to be solved.

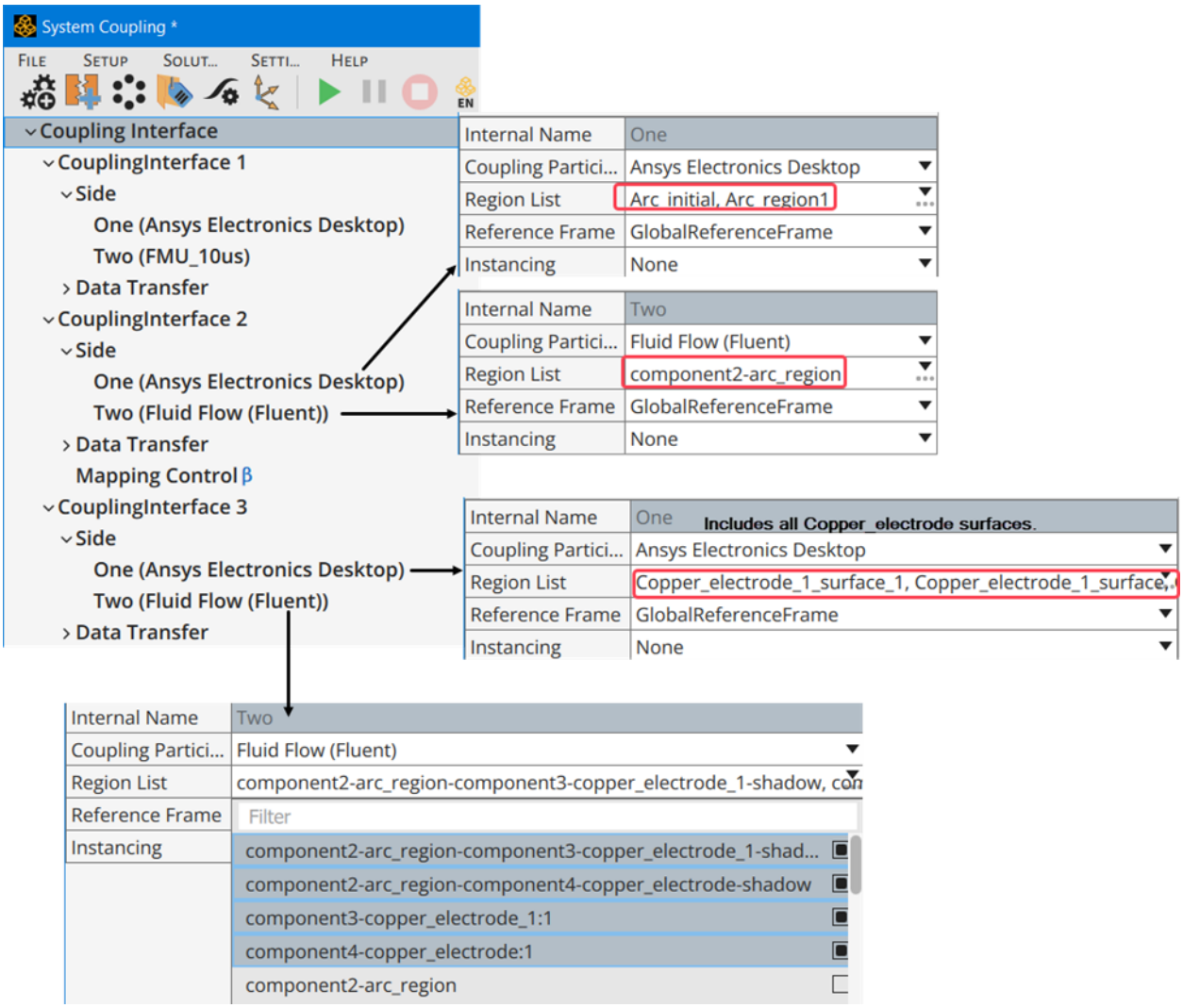
For the **Fluid Flow\Fluent Input**, add the Fluent case and data files in from the \\Fluent directory:



Add three Coupling Interfaces: One between the FMU model and Maxwell and two between Maxwell and Fluent. For Maxwell\Fluent interfaces, select a region or regions for each side where the data will be exchanged: for CouplingInterface2/Side 2, the region specified for Fluent

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

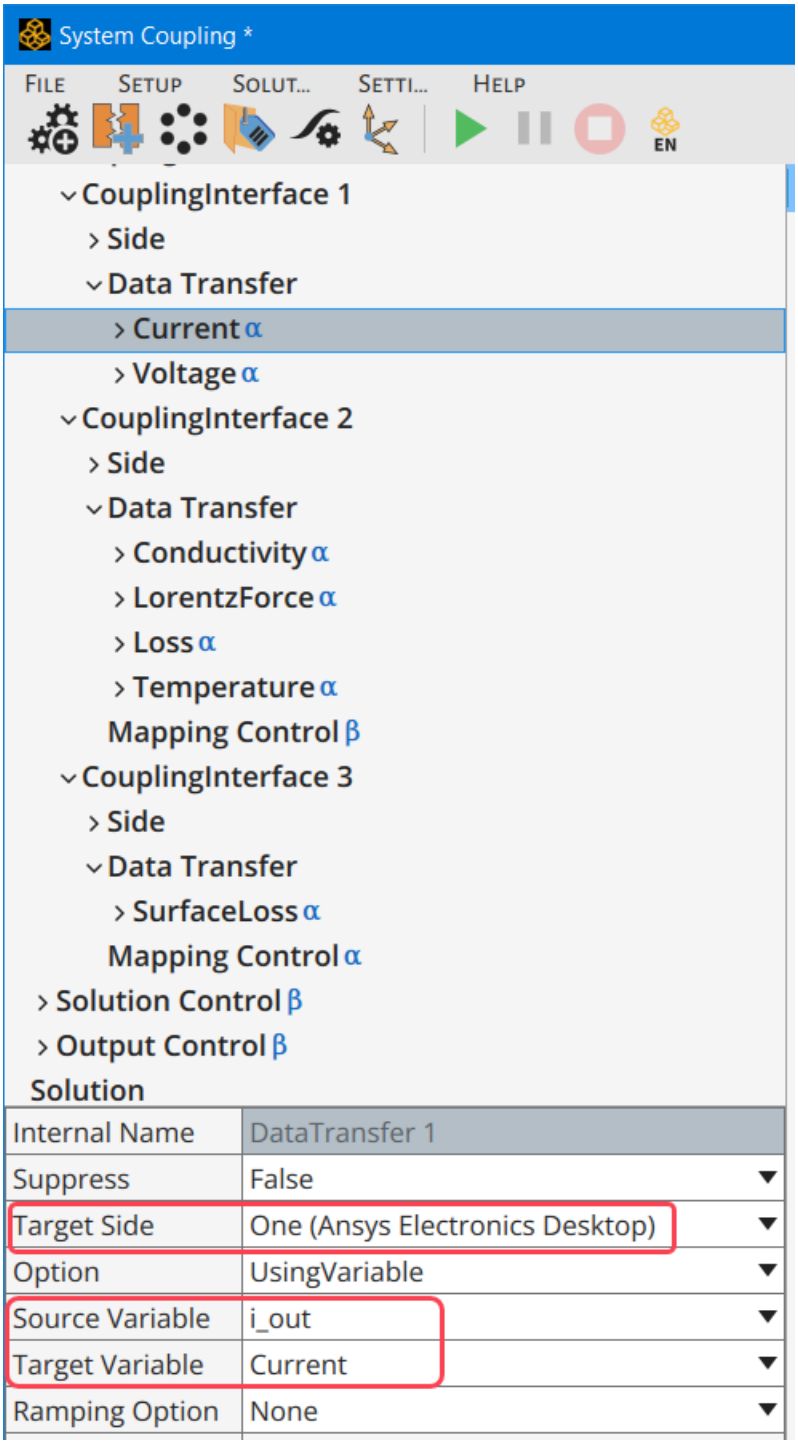
is a fluid; for CouplingInterface3/Side 2, the regions specified for Fluent are the solid-fluid interfaces.



For each coupling interface, specify the data that will be transferred. The example setup includes the following:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- CouplingInterface1\DataTransfer 1: i_out is transferred to Side 1(Maxwell) as Current.

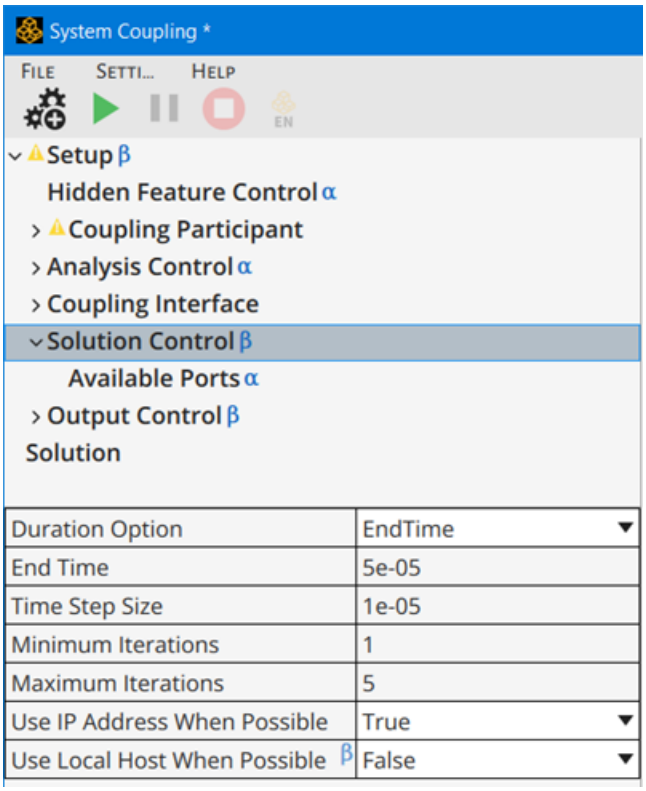


- CouplingInterface1\DataTransfer 2: Voltage is transferred to Side 2 (Twin Builder model) as voltage in.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- CouplingInterface2\DataTransfer 1: Electrical Conductivity is transferred to Side 1 (Maxwell) as conductivity.
- CouplingInterface2\DataTransfer 2: LorentzForce is transferred to Side 2 (Fluent) as Lorentz_Force
- CouplingInterface2\DataTransfer 3: Loss is transferred to Side 2 (Fluent) as heatrate.
- CouplingInterface2\DataTransfer 4: temperature is transferred to Side 1 (Maxwell) as Temperature.
- CouplingInterface3\DataTransfer 1: SurfaceLoss is transferred to Side 2 (Fluent) as heatrate.

When the System Coupling is executed, it will run according to the end time, timestep and iterations specified in the Solution Control setup:



When the system coupling is executed, Maxwell will output the arc voltage (calculated through loss/current) to the external circuit arc voltage interaction with network, and Maxwell will input the arc current from the external circuit (Twin Builder FMU). Maxwell loss and surface loss calculations will be output to Fluent, which will input conductivity and temperature into Maxwell.

The simulation can be monitored from the **Command Console** tab, and plotted results will be available from the **Chart** tab when the simulation is finished.

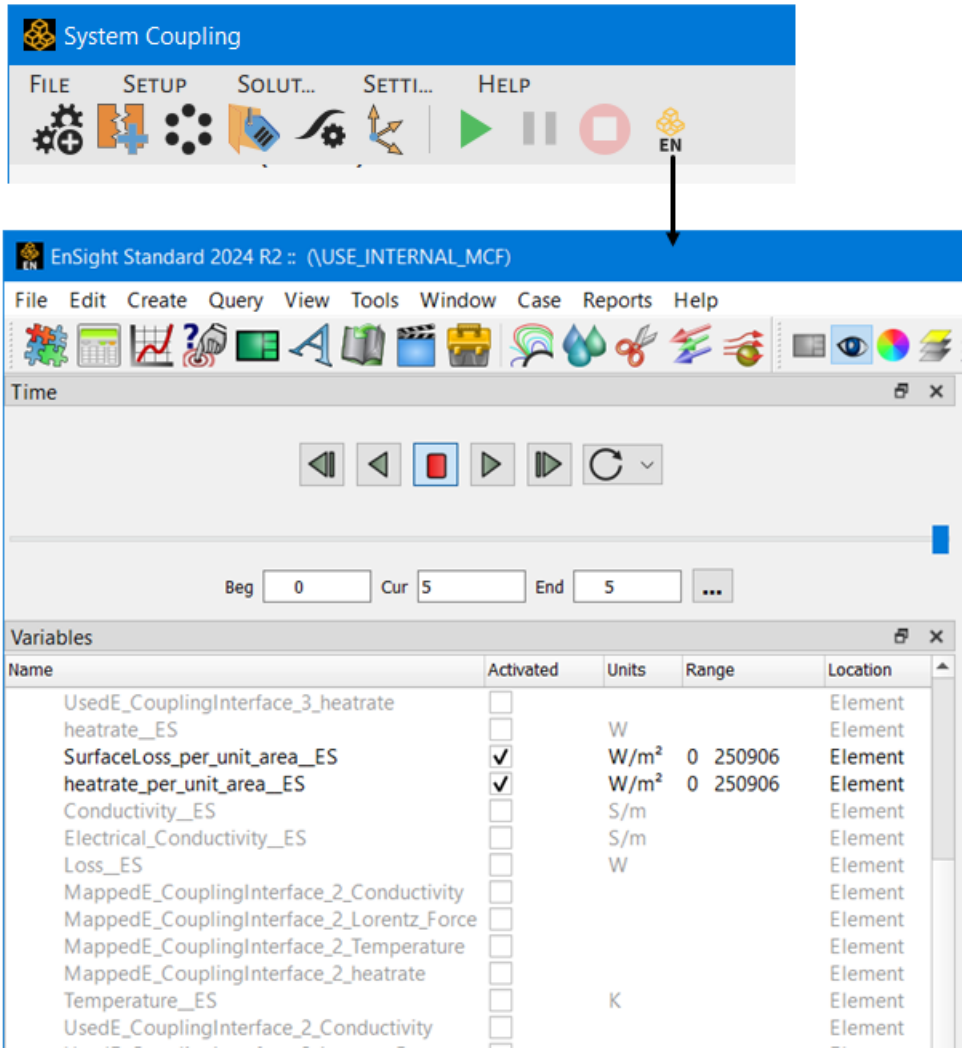
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MESSAGES | COMMAND CONSOLE | CHART | SC TRANSCRIPT | AN

Participant solution status		
Ansys Electronics Desktop	Complete	
Fluid Flow (Fluent)	Converged	
COUPLING ITERATION = 5		
Ansys Electronics Desktop		
Interface: CouplingInterface 1		
Current	Convergence not evaluated	
Value	1.17E+01	1.17E+01
Interface: CouplingInterface 2		
Conductivity	Converged	
RMS Change	1.00E-14	1.00E-14
Weighted Average	2.00E+02	2.00E+02
Temperature	Converged	
RMS Change	4.31E-06	4.92E-06
Weighted Average	3.10E+02	3.10E+02
Fluid Flow (Fluent)		
Interface: CouplingInterface 2		
LorentzForce	Converged	
RMS Change	2.01E-14	4.14E-14
Sum x	3.14E-05	3.14E-05
Sum y	-2.91E-09	-2.91E-09
Sum z	-4.74E-09	-4.74E-09
Loss	Converged	
RMS Change	2.04E-14	3.32E-14
Sum	1.52E+01	1.52E+01
Interface: CouplingInterface 3		
SurfaceLoss	Converged	
RMS Change	1.00E-14	1.00E-14
Sum	6.69E+01	6.69E+01
Participant solution status		
Ansys Electronics Desktop	Complete	
Fluid Flow (Fluent)	Converged	

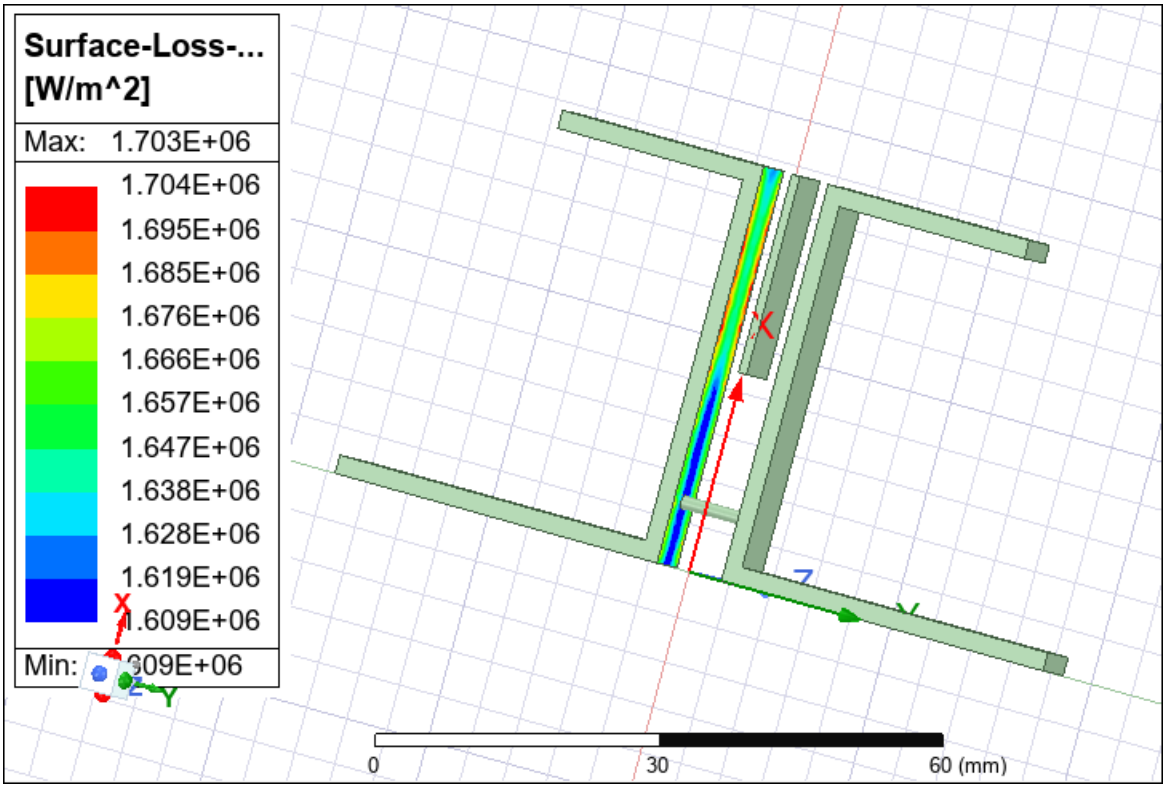
To see how well data is transferred from one region to another, we can use Ensight to compare Maxwell surface loss to Fluent heat rate:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To visualize surface loss in Maxwell, right-click on the **surface** in the Modeler window, and select **Fields** > **Other** > **Surface Loss Density**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note: For detailed information on System Coupling in general, refer to the System Coupling User's Guide. For details on working in System Coupling's GUI or CLI, see the Using System Coupling's User Interfaces section.

Note: For more examples of Maxwell participating in cosimulations, review the coil-and-core induction heating, bar-and-coil induction heating, bus bar, and electric motor examples in System Coupling Tutorials.

Modeling an Electric Arc in a Device with Motion

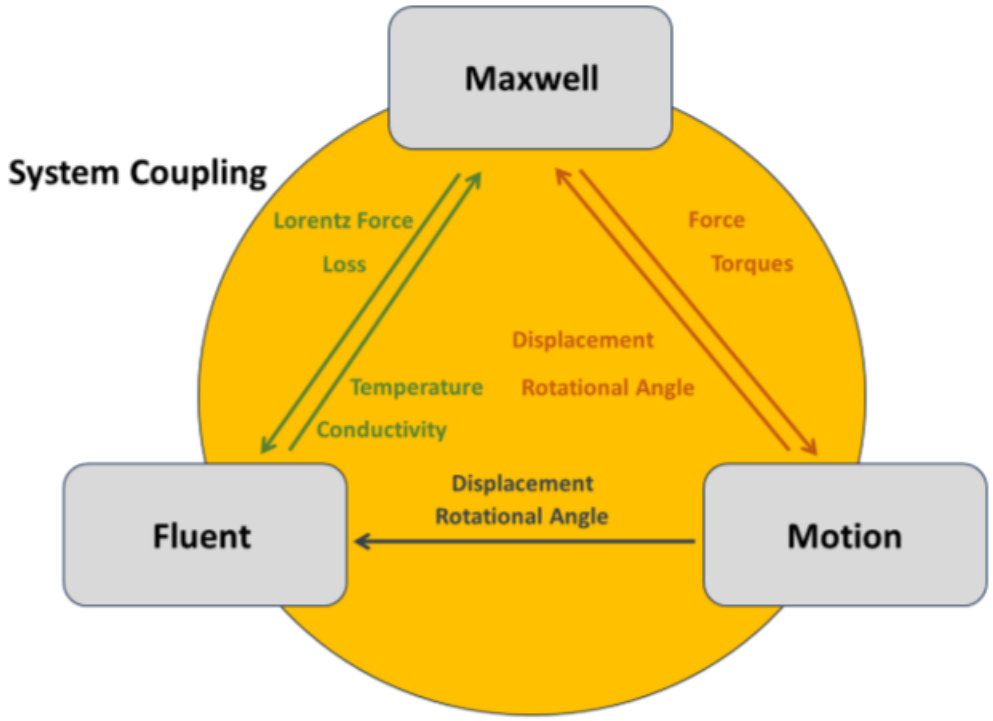
To model an electric arc in a device with motion, System Coupling can be used with the Maxwell, Fluent, and Motion applications. All participants in the System Coupling are coupled and dependent on each other. The workflow for this type of modeling has the following order: **Maxwell** > **Fluent** > **Motion**. Multiple iterations (data exchanges) can occur between each two participants.

As seen in the flow chart, the Maxwell application generates volume loss, surface loss (including nonlinear resistance thin layer losses on Electric Arc Anode and Cathode surfaces) and Lorentz forces for the Fluent application. In turn, the Fluent application can consume heat rate (surface

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

loss from the Maxwell application) on solid-fluid interfaces, and it feeds electrical conductivity, temperature, and simulation time back to the Maxwell application.

The Maxwell application inputs forces and torques to the Motion application, which then feeds back displacement and rotation angle to the Maxwell and Fluent applications to be used in the calculations of those applications.



This topic will address the general requirements for simulating an electric arc with displacement and/or rotational motion. It assumes that the user is familiar with the Maxwell, Fluent, and Motion tools, so it will only detail the requirements for the System Coupling setup. For details on the individual tools, see the corresponding reference and tutorial topics.

> **Warning:** When running a System Coupling analysis, make sure that the input from the Motion, Maxwell and Fluent applications are up-to-date and are of the same release version as the System Coupling application.

Motion Setup

For the electric arc, there are two motion applications:

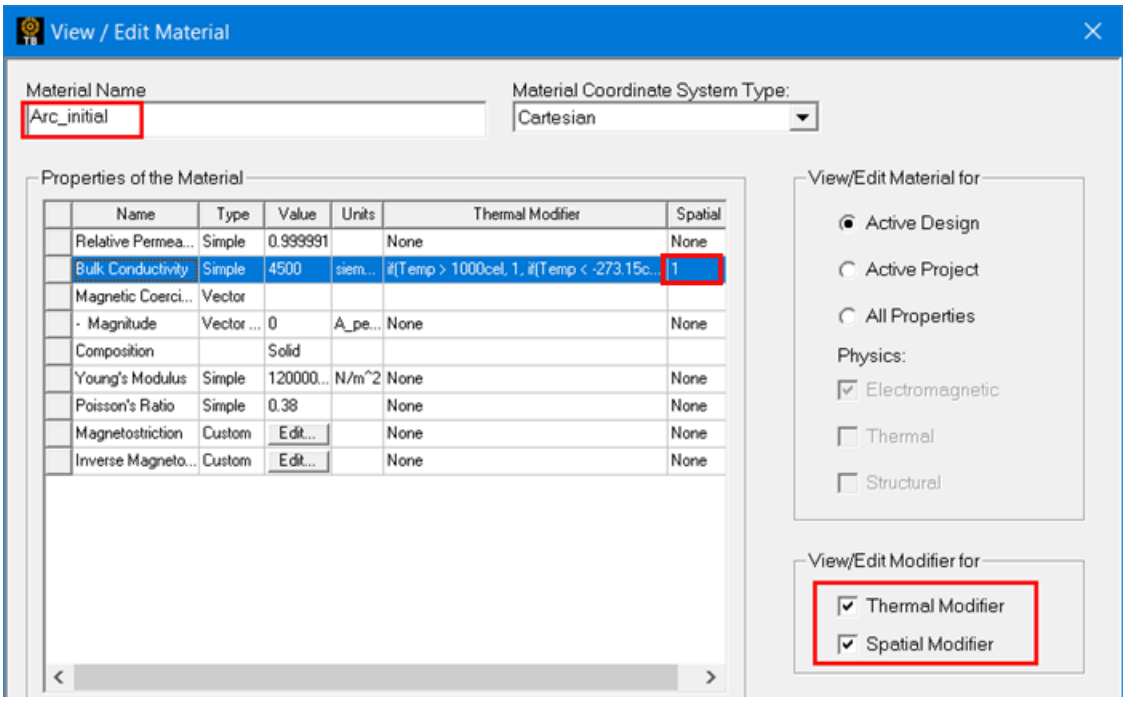
- Trigger + Switch Arm (translation)
- Trigger + Switch Arm (rotation)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The trigger and switch arm objects are designated in the Motion application. The co-simulation variables that will be used in the Maxwell System Coupling window are also defined in the Motion application. For example, in the Motion application, the user defines a motion variable and applies this variable to an object to determine the motion of that object. In the Maxwell System Coupling Setup dialog box, the user will apply that motion variable to a trigger or switch arm quantity.

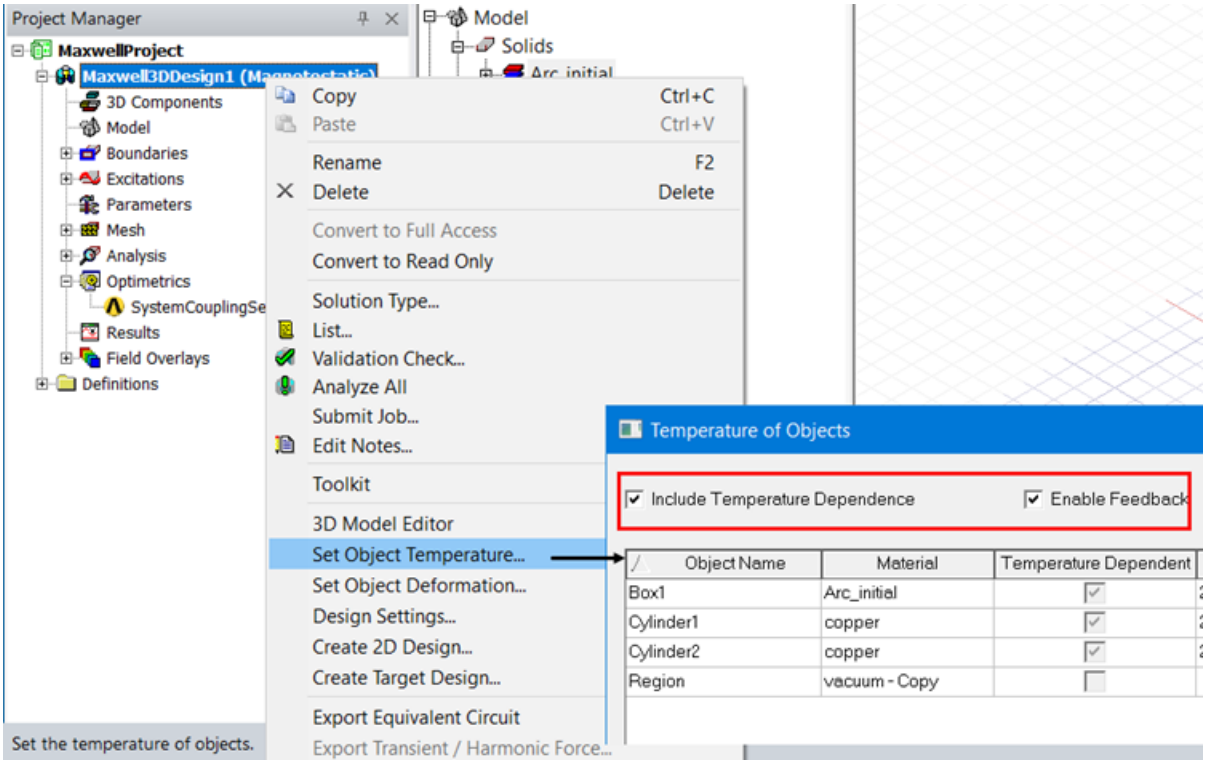
Maxwell Setup

For the Maxwell model, the arc material should have the **Thermal Modifier** and **Spatial Modifier** options checked for Bulk Conductivity, and the **Spatial Modifier** should be set to **1**.



For the model Temperature of Objects, check **Include Temperature Dependence** and **Enable Feedback**:

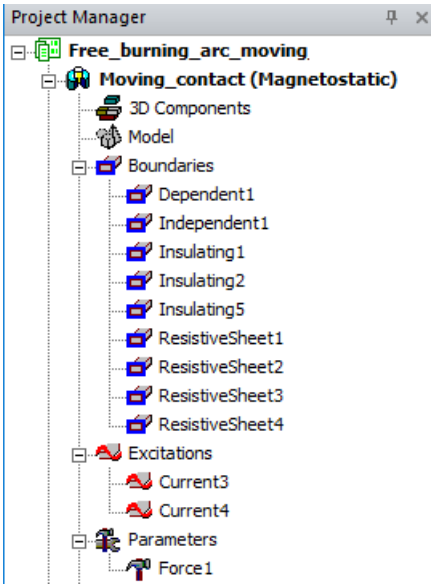
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note: A Resistive Sheet boundary can be used to simulate nonlinear loss and nonlinear surface loss; see Assigning a Resistive Sheet Boundary for the 3D Magnetostatic Solver for more details.

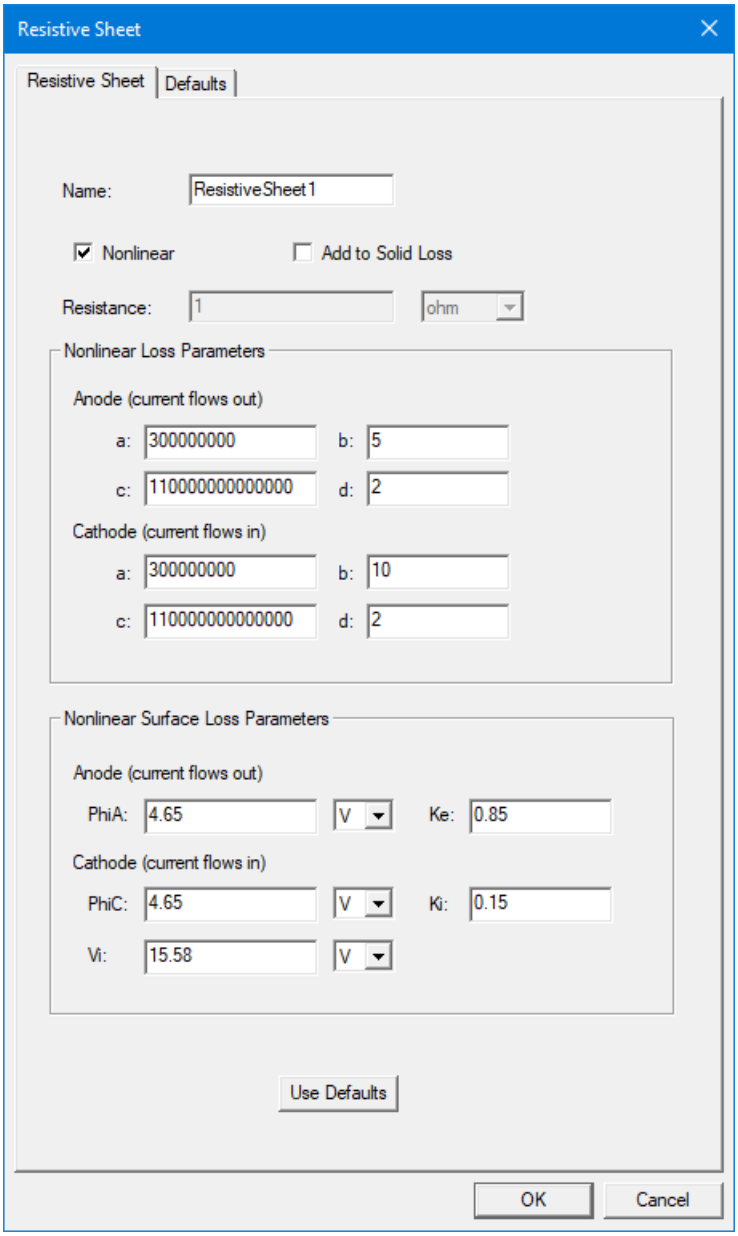
Set up the desired magnetostatic simulation, including boundaries, current excitations, and simulation time:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



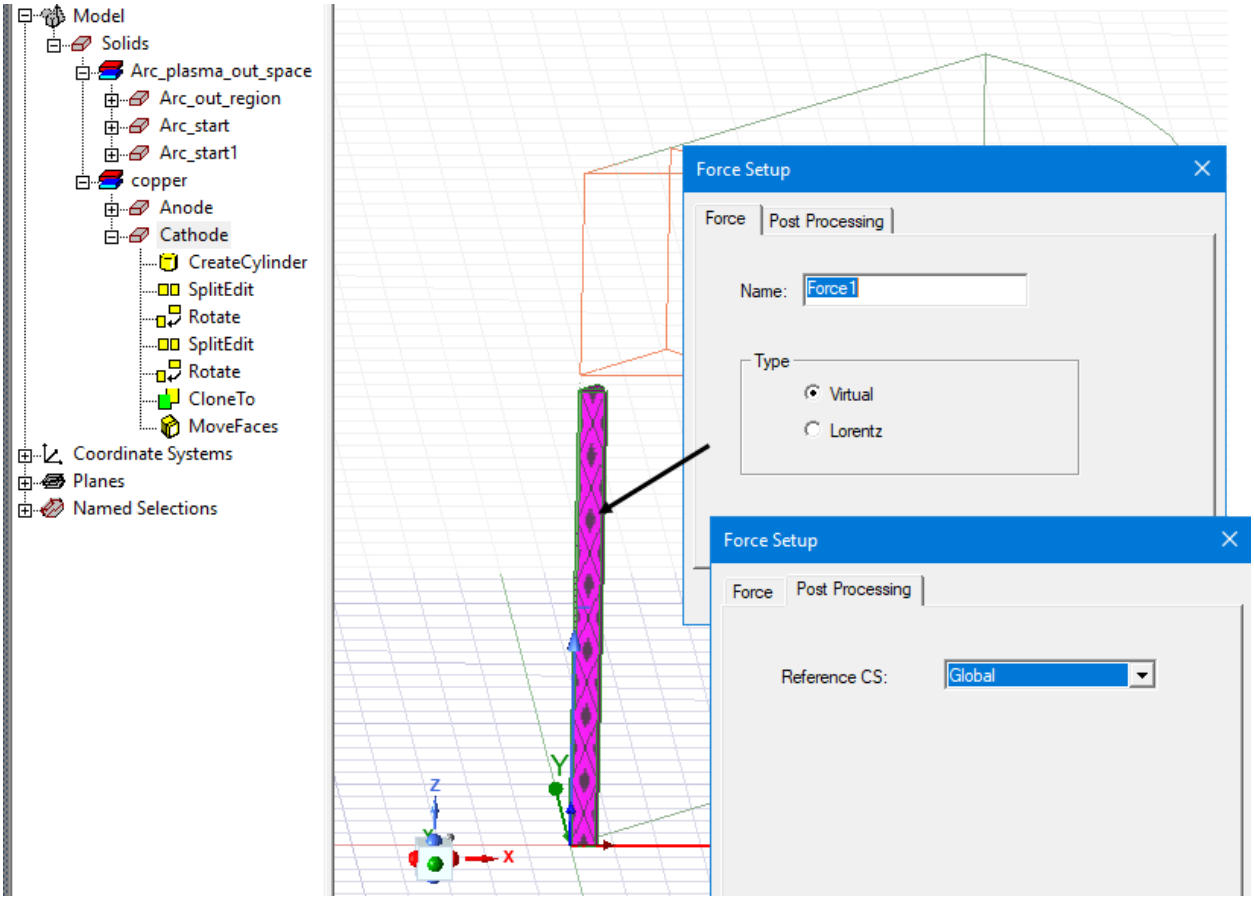
The nonlinear Resistive Sheet boundary condition is used on Anode/Cathode surfaces to simulate nonlinear loss and nonlinear surface loss:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For force or torque, define force/torque parameters on the object of interest. For example, in a free-burning arc design, we apply a force to the Cathode object:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



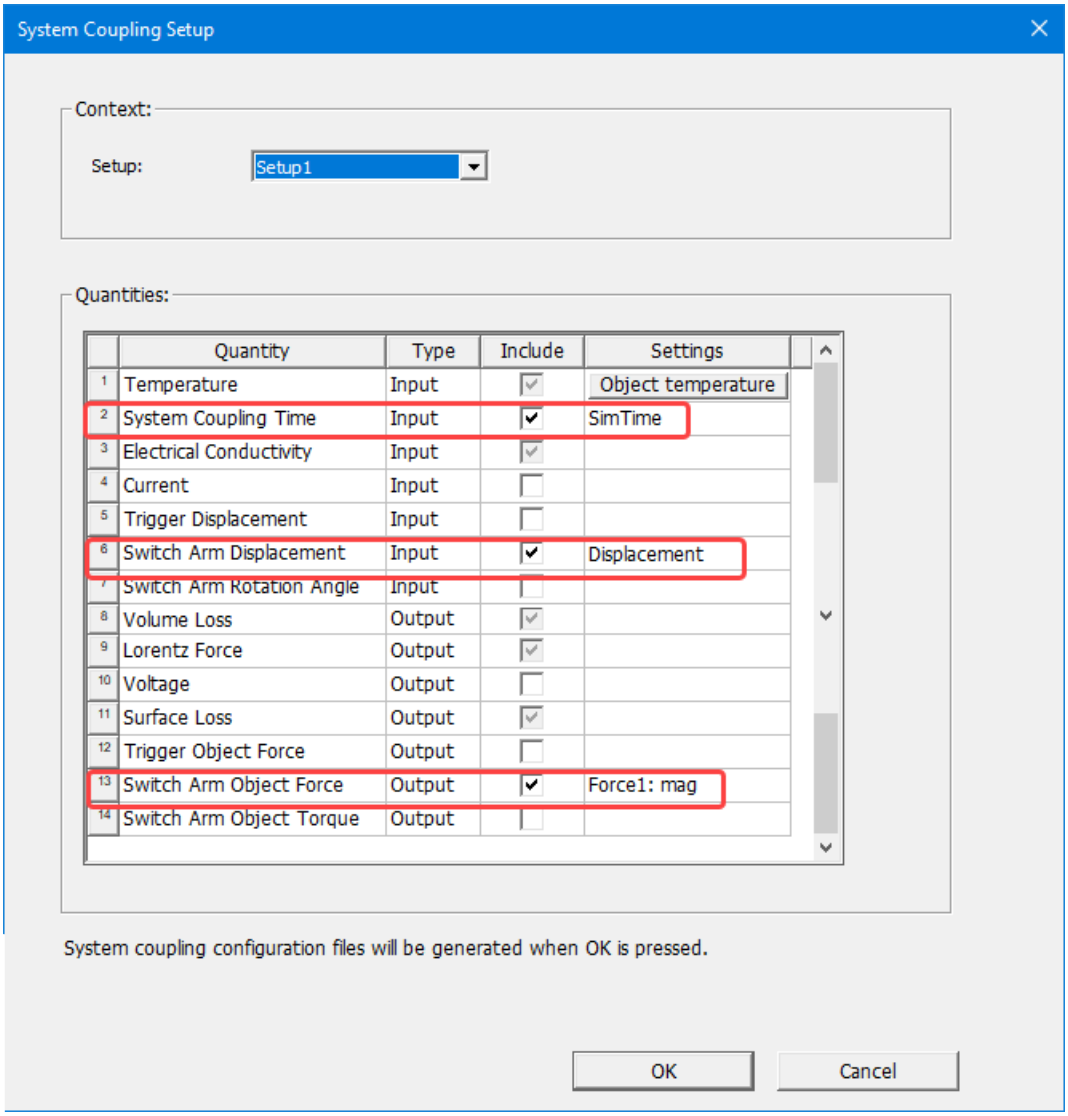
In Force Setup dialog box's **Post Processing** tab, select the correct reference coordinate system for the force of the selected object. Note that the Maxwell application exports Object Force in the global coordinate system.

Add a System Coupling Setup to the Optimetrics folder, and set the following:

- Check **System Coupling Time**, and set the **Settings** variable to **SimTime**.
- For translational motion, check a switch arm displacement input and a switch arm object force output.
- For rotational motion, check a switch arm rotation angle input and an switch arm object torque output.
- For both translational and rotational motion, trigger displacement input and trigger force output can also be included.

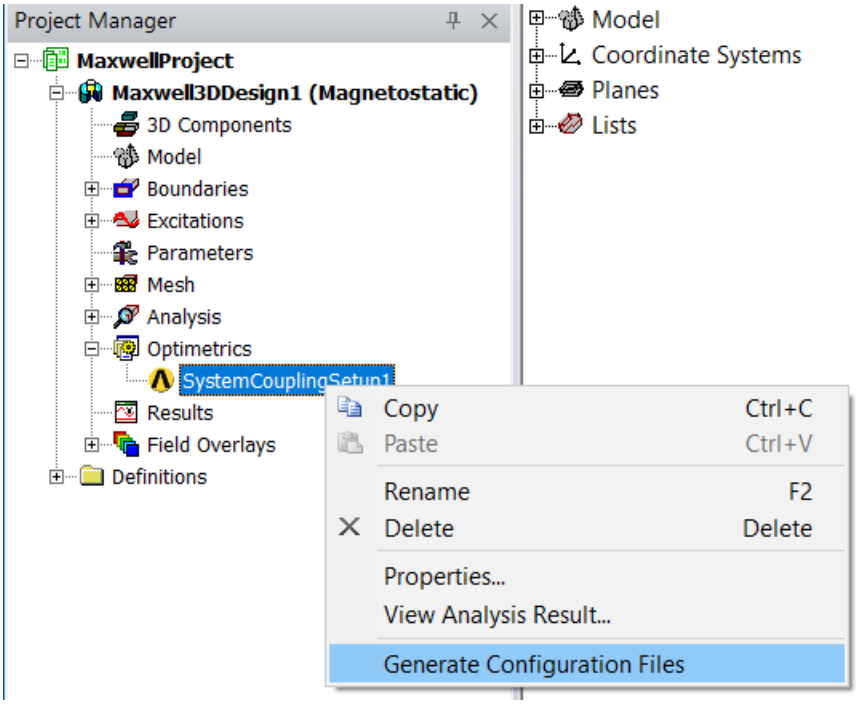
For our free-burning arc example, the force we defined on the Cathode object corresponds to Switch Arm Object Force in the System Coupling setup. The Switch Arm Displacement is defined with the Displacement variable that was defined in the Motion application.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The configuration files needed for System Coupling are automatically generated when you click **OK** in the System Coupling Setup dialog box, but if you want to regenerate the configuration files without reopening the dialog box, right click on the System Coupling Setup in the Project Manager, and select **Generate Configuration Files**:

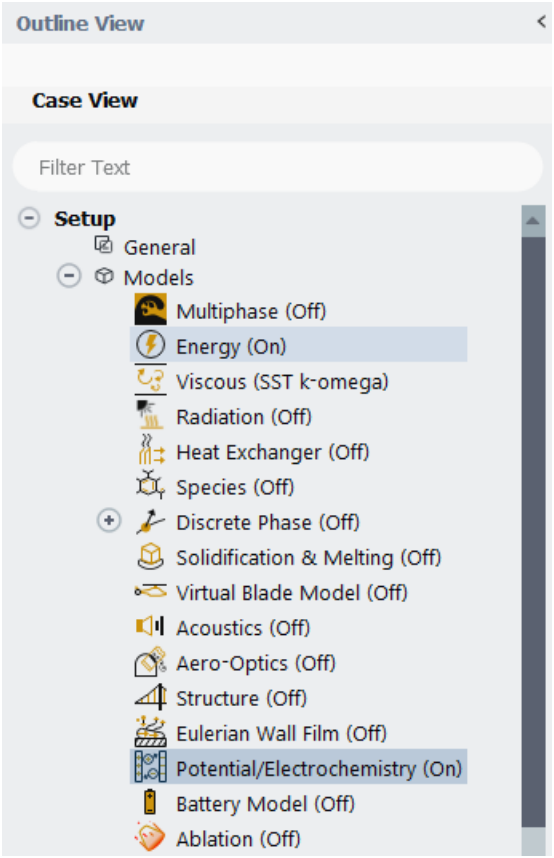
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Fluent Setup

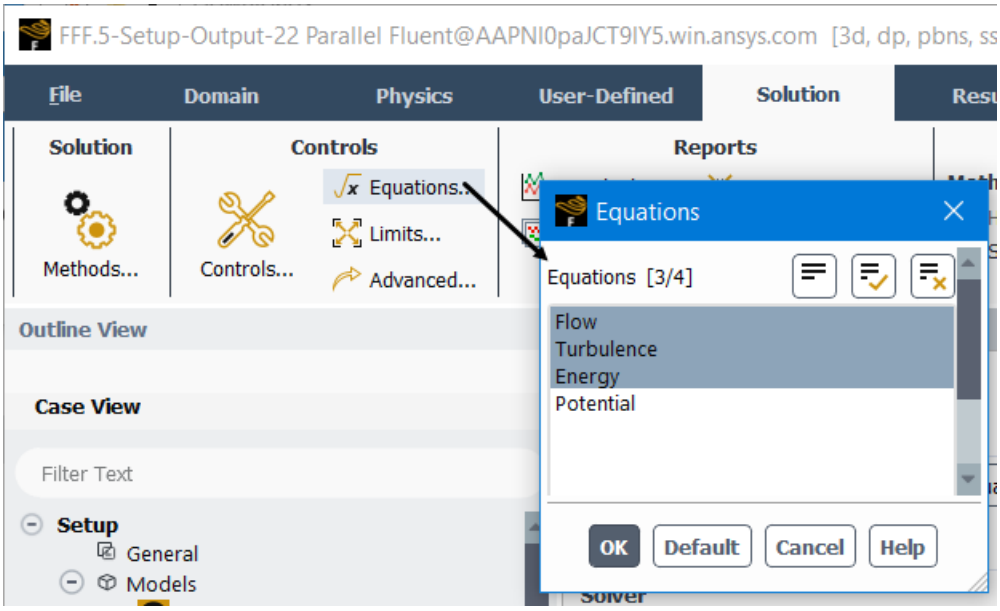
For the Fluent model, the materials should have the same properties as the Maxwell model. The Energy and the Potential/Electrochemistry models should be turned on. Potential is needed because it is used to calculate the Electrical Conductivity required by the Maxwell model.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

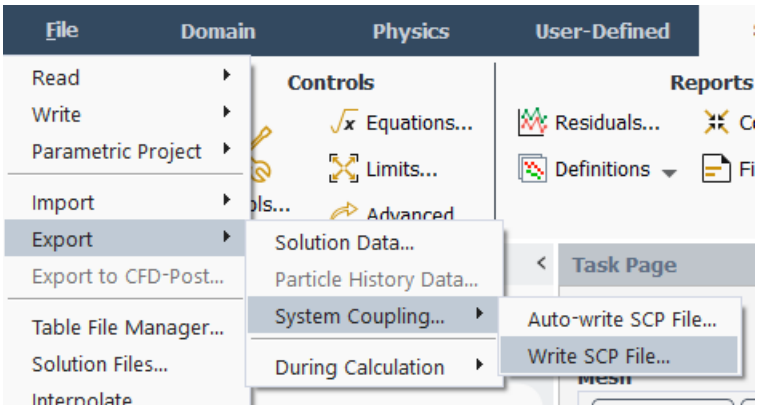


From the **Solution** panel, the **Potential** equation must be deactivated because electromagnetics will be simulated in the Maxwell application:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After setting any required properties and boundary conditions for your model, export the Fluent setup to a System Coupling file:



System Coupling Setup

Load the System Coupling participants one by one following the order **Ansys Electronics Desktop Maxwell** > **Fluid Flow** > **Motion**: click **Add Participant** icon , and in the dialog box that opens, find the input files for each participants and add one by one.

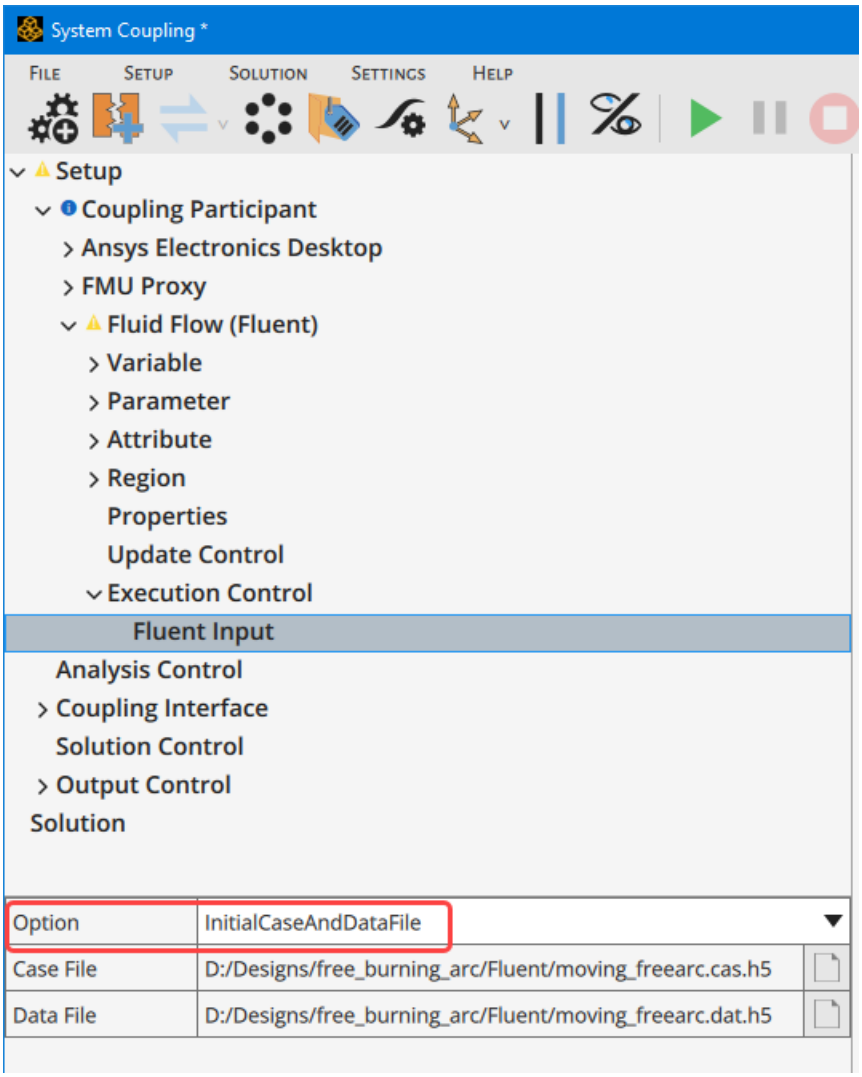
- The input file for AEDT is a \\MaxwellProject_Maxwell3DDesign_ SystemCouplingSetup1.scp file.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The input file for Fluid Flow is a \\Fluent\fluent.scp file.
- The input for Motion is a .FMU file.

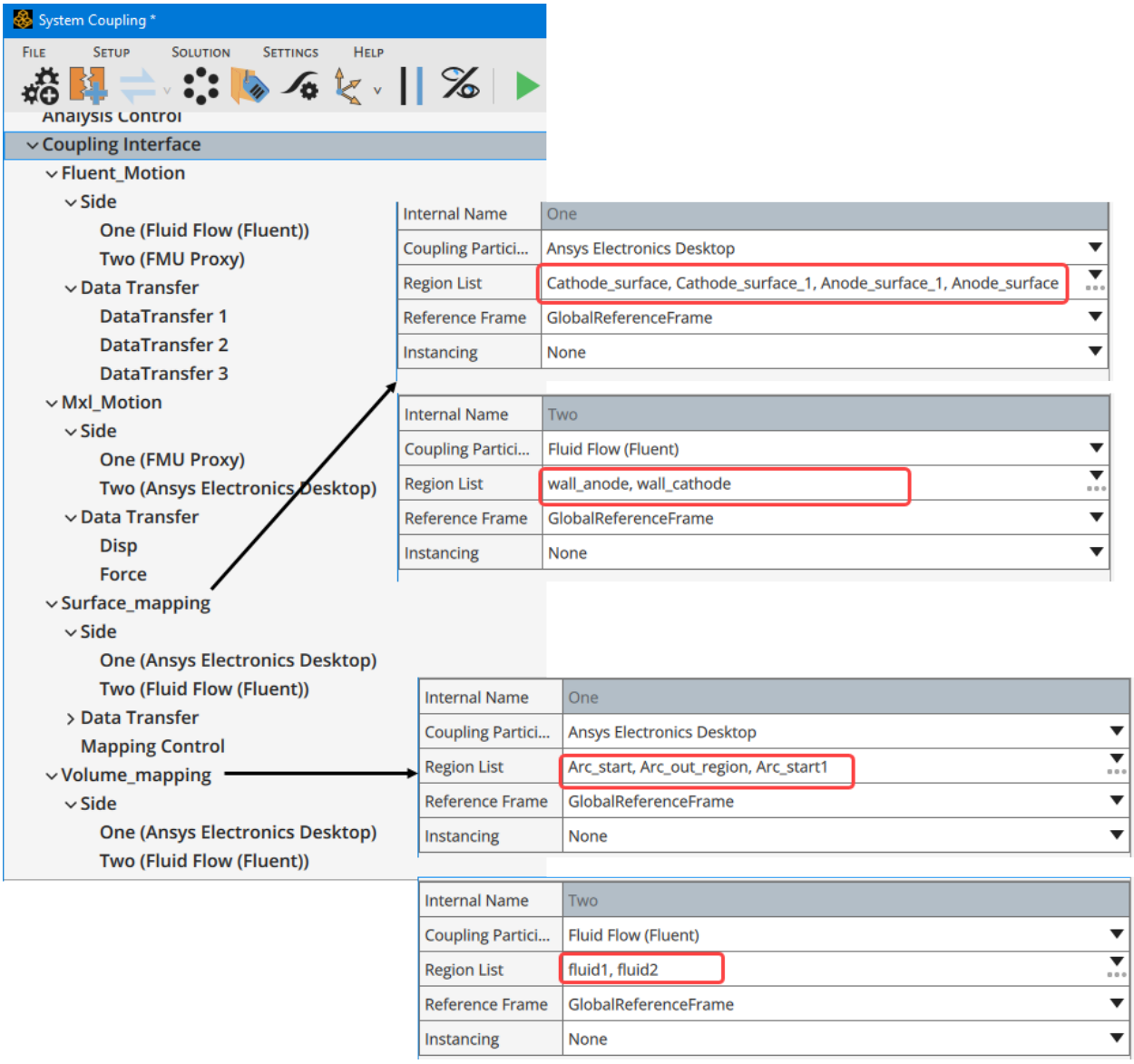
Note that the order in which they were added reflects the order in which they have to be solved.

For the **Fluid Flow\Fluent Input**, add the Fluent case and data files in from the \\Fluent directory:



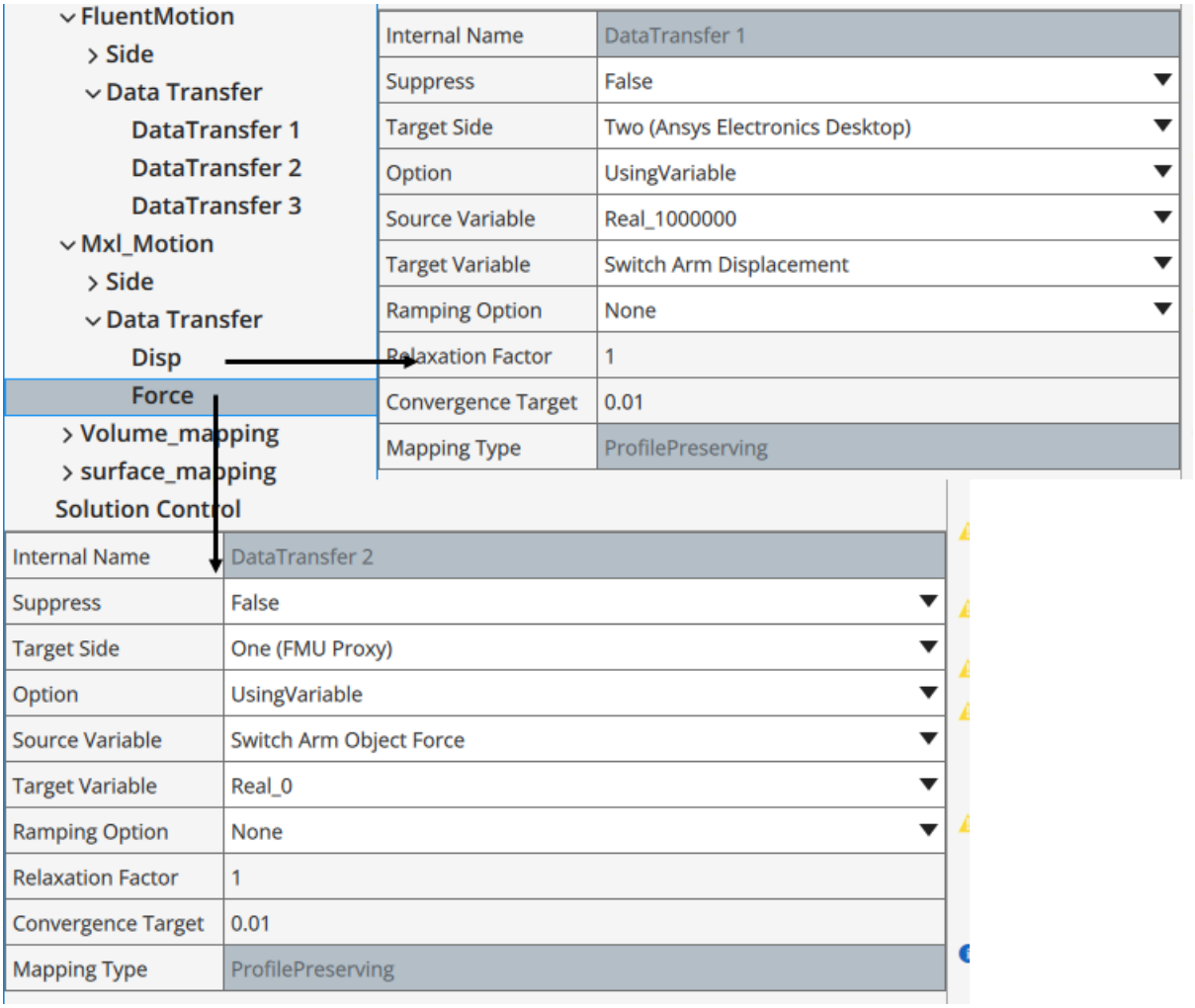
For the free-burning arc design, we need four Coupling Interfaces: two between Ansys Electronics Desktop (Maxwell) and Fluent, one between Ansys Electronics Desktop and Motion (FMU proxy), and one between Fluent and Motion. For the Maxwell\Fluent interfaces, select a region or regions for each side where the data will be exchanged:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



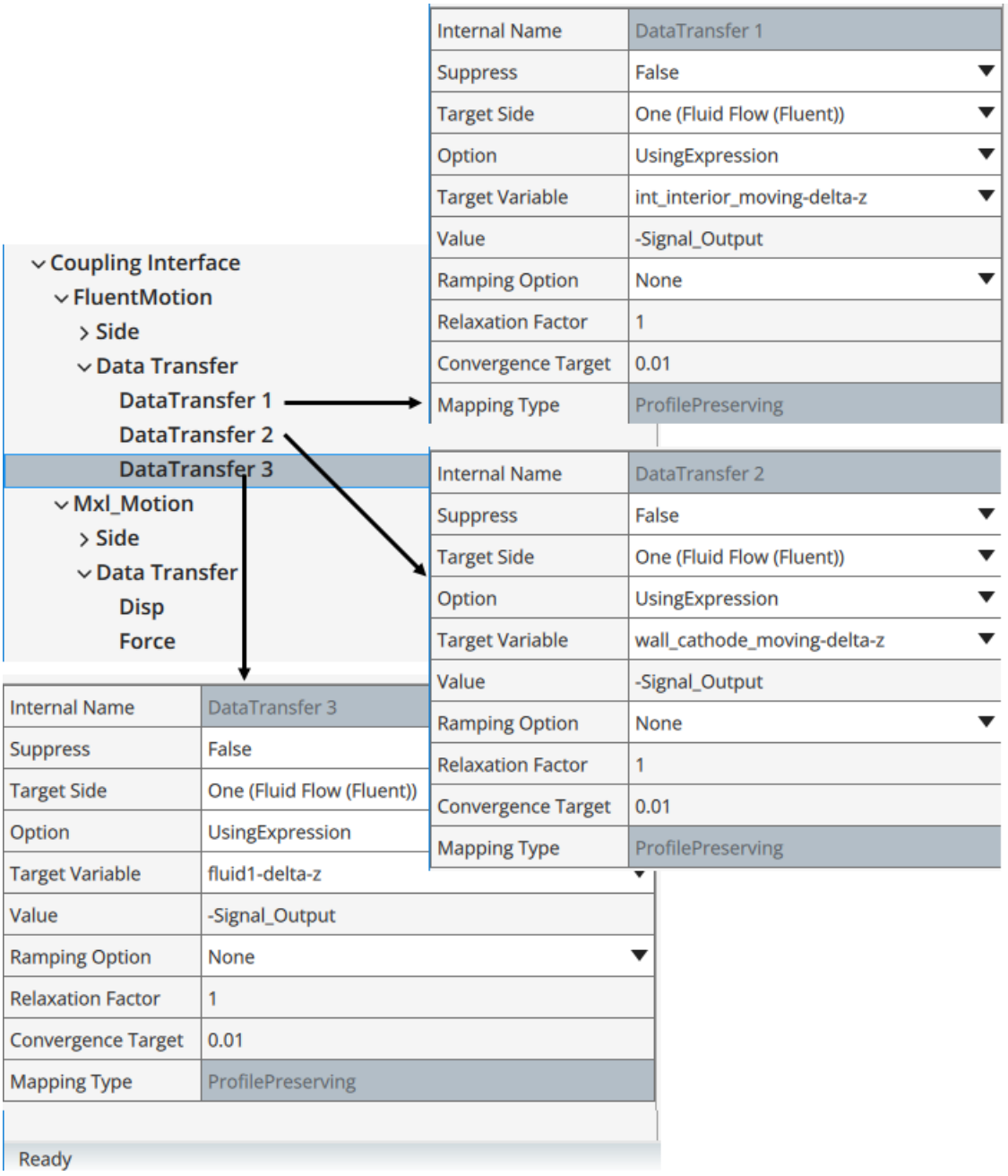
For each coupling interface, specify the data that will be transferred. For example, in the free-burning arc example, we transfer the displacement from the Motion application to the Maxwell application, and we transfer the force from the Maxwell application to the Motion application:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- The Fluent_Motion interface is used to transfer variable values from the Motion application to the Fluent application:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

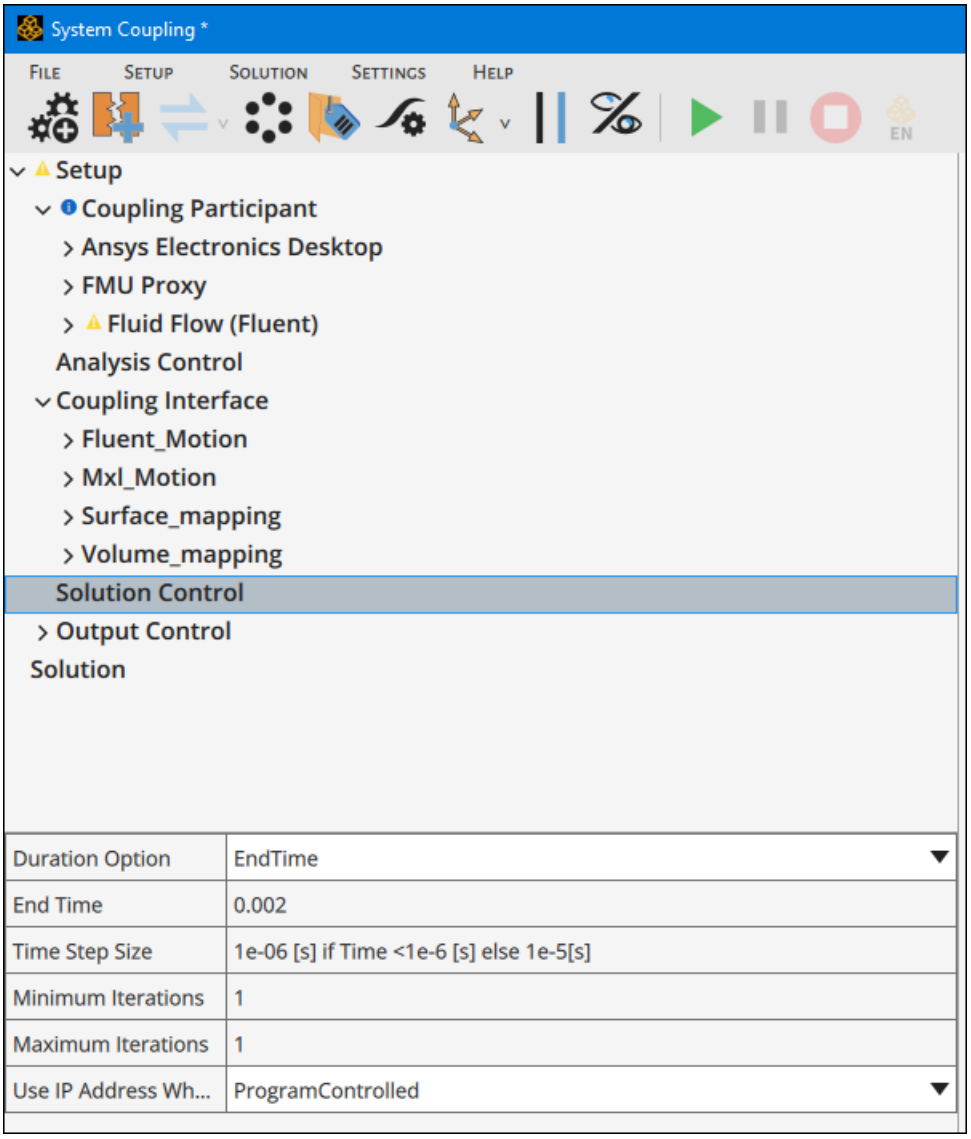


- For the Surface_mapping interface: SurfaceLoss is transferred from the Maxwell application to the Fluent application as heatflow.
- For the Volume_Mapping interface:
  - Electrical Conductivity is transferred to Side 1 (Maxwell) as conductivity.
  - LorentzForce is transferred to Side 2 (Fluent) as Lorentz_Force.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Loss is transferred from the Maxwell application to the Fluent application as heatrate.

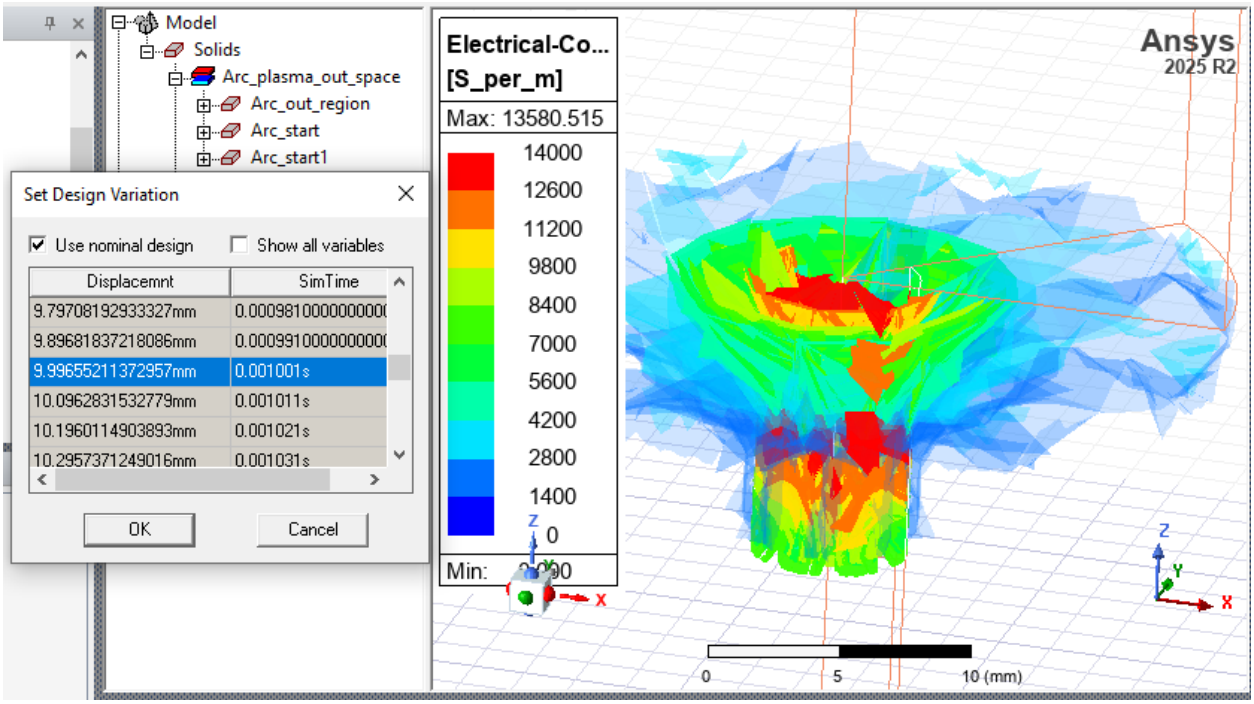
When the System Coupling is executed, it will run according to the end time, timestep and iterations specified in the Solution Control setup:

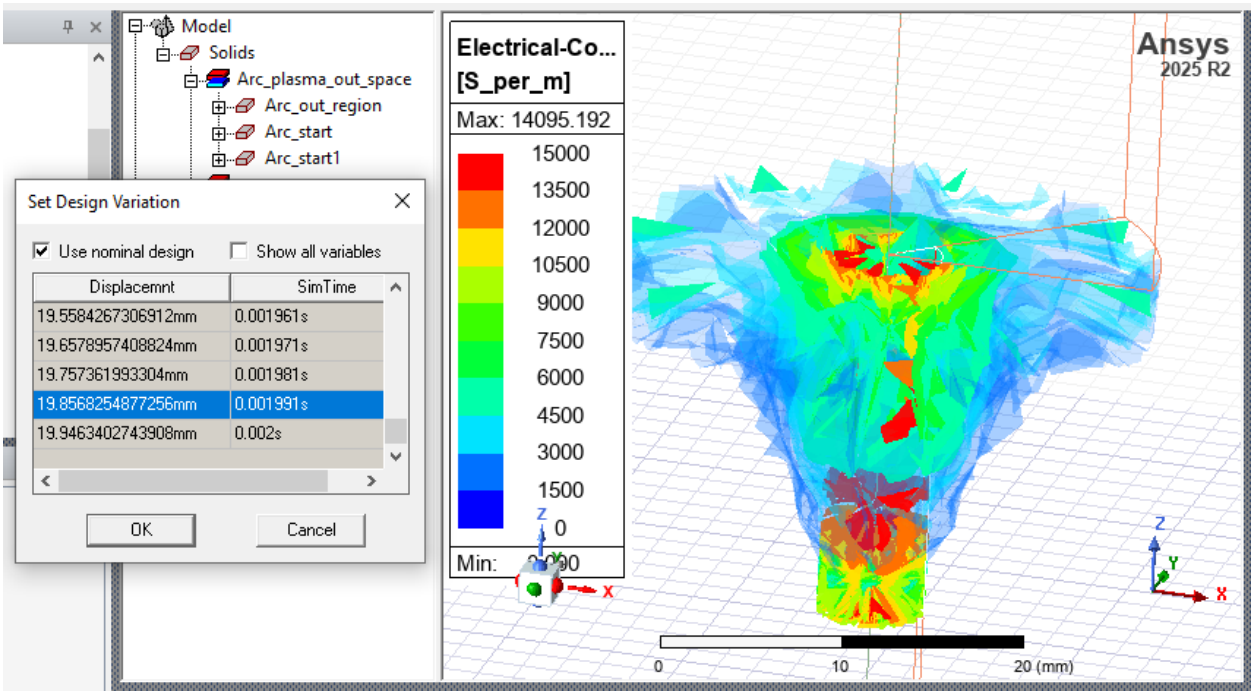


The simulation can be monitored from the **Command Console** tab, and plotted results will be available from the **Chart** tab when the simulation is finished.

The Electrical Conductivity as a function of displacement can be visualized in Maxwell:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: For detailed information on System Coupling in general, refer to the System Coupling User's Guide. For details on working in System Coupling's GUI or CLI, see the Using System Coupling's User Interfaces section.

Note: For more examples of Maxwell participating in cosimulations, review the coil-and-core induction heating, bar-and-coil induction heating, bus bar, and electric motor examples in System Coupling Tutorials.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

18 - High Performance Computing

Ansys Electronics Desktop affords High Performance Computing (HPC) options that allow you to speed up processing. HPC leverages multiple cores through matrix multiprocessing, distributed frequency points (called spectral decomposition method or SDM), domain decomposition (DDM), parallel hybrid FEM/IE solving or the finite antenna array DDM. In addition, hierarchical HPC solving is possible where frequency points can be distributed with each frequency point using multiple cores or machines for large scale DDM analysis at each frequency point, all in parallel. Maxwell intelligently determines which jobs are to be performed and how to distribute them for the simulation. It automatically apportions the jobs during the simulation process and makes optimum use of the available resources. This computing technology enables generating accurate solutions for large, complex, higher-fidelity models.

See the following topics for more information and configuration tips:

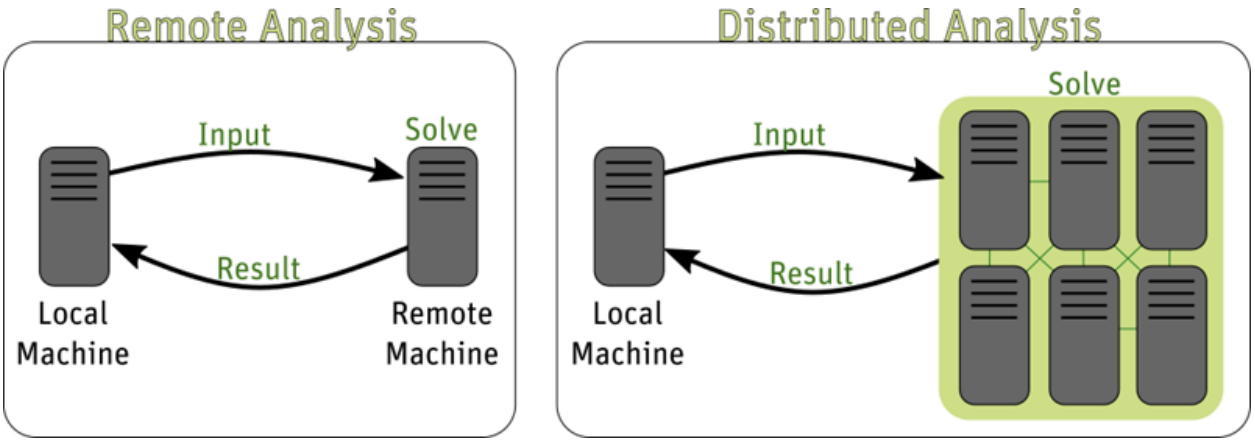
- Remote Analysis
- Distributed Analysis
- Ansys Cloud Burst Computing Support for HPC
- Large Scale DSO for Parametric Analysis
- Interactive Scheduler Jobs
- HPC Integration
- Multi-Step Job Submission
- Windows to Linux Job Submission
- Distribution Command Line Options
- GPU Acceleration

Remote Analysis

It is possible to solve a project on a different machine from the one on which you are running Ansys Electronics Desktop. This is particularly useful when you want to take advantage of a more powerful machine but it is not convenient to access that machine. This process involves configuring the machine that will perform the solve (the remote machine), as well as the machine from which the simulation is to be launched (the local machine).

This can also be extended into distributed analysis, where a specified analysis, if supported, is concurrently solved on multiple machines.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Important:

In both Remote and Distributed Analysis, communication between machines can drastically affect performance. Use of a high-speed network system, such as Gigabit or Infiniband, is recommended for optimal performance.

The rest of this topic covers the following:

- Prerequisites for Remote and Distributed Analysis
- Configuring the Local Machine to Solve Remotely
- Remote Analysis Options
- Running Remote Analysis

The **Export Options Files** command writes XML files containing options settings at all levels to the specified directory. **Tools** > **Options** > **Export Options** makes it easier for different users to use Ansys EM tools installed on shared directories or network drives. For some example use cases, see: Example Uses for Export Options Features.

Prerequisites for Remote and Distributed Analysis

1. You must have installed Ansys Remote Simulation Manager (RSM) or a supported High Performance Computing (HPC) management software program (See: High Performance Computing (HPC) Integration).

   For a list of currently supported HPC software, select the topic, *Ansys 2025 R2 - Job Schedulers and Queuing Systems Support* from the Platform Support page.

2. Ansys Electronics Desktop must be accessible from all remote machines as well as on the local machine. If the analysis uses MPI, then the path of the Ansys Electronics Desktop installation must be the same on all of the machines used for the analysis (remote and local). This may be a shared network path accessible from all hosts. Alternatively, it may

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

be a local installation on each host; in this case the installation path must be the same for all hosts. If using a shared network path, there should not be a local installation of the same Ansys Electronics version on any of the hosts. See Distributed Analysis for information on whether MPI is needed for the analysis.

3. If you use RSM, it must be accessible from all remote machines. In addition, the Ansys Electronics Desktop engines must be registered with each initialization of RSM.

   To do this on each machine:

   - From Windows, click **Start** > **Programs** > **Ansys EM Suite 2025 R2** > **Register with RSM**. You can also run **RegisterEnginesWithRSM.exe**, located in the product subdirectory (for example, C:\Program Files\ANSYS Inc\v252\AnsysEM\RegisterEnginesWithRSM.exe).

     In each case, you see a dialog box confirming the registration. Click **OK** to confirm.

   - From Linux, run **RegisterEnginesWithRSM.pl**, located in the product installation directory (for example, /opt/ansys_inc/v252/AnsysEM/RegisterEnginesWithRSM.pl).

     If the RSM service cannot run due to permission issues for the configuration file, it issues an error message and exits.

     If your product is not registered with RSM, the analysis will run locally.

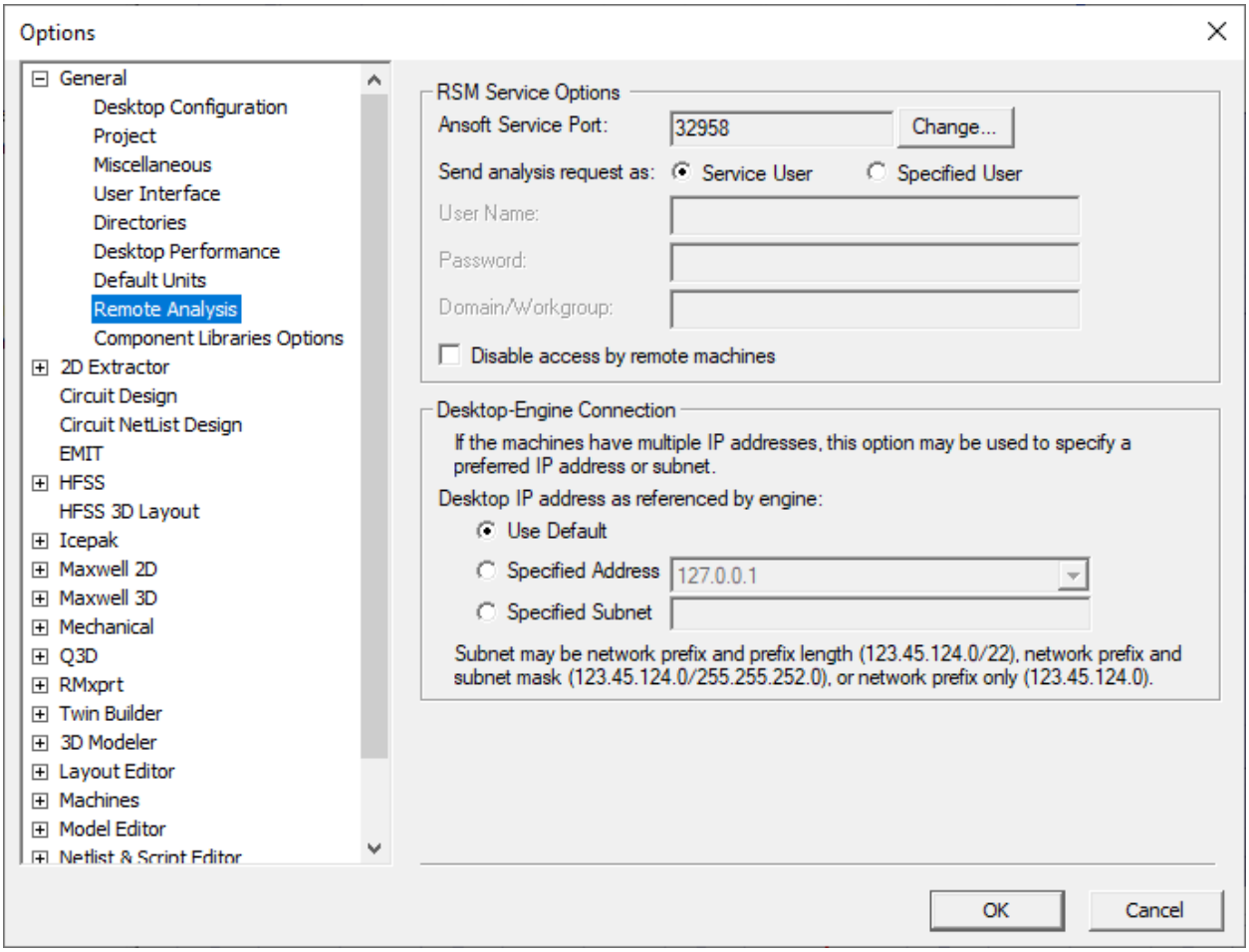
Configuring the Local Machine to Solve Remotely

To set Maxwell Analysis options, see: Distributed Analysis.

Remote Analysis Options

You can also set Remote Analysis Options from **Tools** > **Options** > **General Options** > **General** > **Remote Analysis**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Select whether to run simulation processes as the user running RSM (Service User), or sd a Specified User. If you select Specified user, you must provide the User Name, Password, and any Domain/Workgroup on which this user is defined. If the name or password is incorrect, the Messages window issues a warning, and the solver attempts to perform the analysis as the Service User.

Running Remote Analysis

When you run a simulation remotely, you should see a message in the Progress window identifying the design name and the specified remote machine. You will see Progress messages as the simulation continues. When the simulation is complete, you will see a message in the **Messages** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Troubleshooting

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

Problem:

When you try to solve from local to remote machine, a COM engine process starts on the remote machine, but the user interface hangs indefinitely.

This occurs if the remote solve option is enabled after the COM daemon is started, or when the option "Don't allow exceptions" is selected for the Windows firewall.

Resolution:

Remote solve needs either firewall exceptions to be turned on, or firewall to be completely turned off.

Problem:

When you try to solve from a local to a remote machine, you receive the following error message:

[error] Unable to locate or start COM engine on 'nomachine' : Unable to reach AnsoftRSMService. Check if the service is running and if the firewall allows communication. (10:57:13 PM Aug 13, 2019)

Resolution:

This message can happen if the machine is not present, the network connection is down, if there are firewall issues, or if the service is not running.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Problem:

A solve that is distributed to multiple hosts using MPI fails because the AnsysEM installation path is different on different hosts.

Resolution:

Ansys Electronics Desktop must be accessible from all remote machines as well as on the local machine. If the analysis uses MPI, then the path of the Ansys Electronics Desktop installation must be the same on all of the machines used for the analysis (remote and local). This may be a shared network path accessible from all hosts. Alternatively, it may be a local installation on each host; in this case the installation path must be the same for all hosts. If using a shared network path, there should not be a local installation of the same Ansys Electronics version on any of the hosts. See Distributed Analysis for information on whether MPI is needed for the analysis.

Problem:

The command `ansoftrsmservice stop` will not work as root.

Resolution:

Use systemctl command to stop the service:

```
sudo systemctl stop ansoftrsmservice
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Remote Solve Node = Windows

Error:

"Unable to locate or start COM engine on <remote node> : Unable to reach AnsoftRSMService. Check if the service is running and if the firewall allows communication."

Resolution:

1. Try disabling the firewall.
2. Confirm that you have not changed the Service Port in **Tools** > **Options** > **General Options** > **General** > **Remote Analysis**. If you have, change it back to the default (32958), restart Ansys Electromagnetics Desktop, and try to solve again.
3. Make sure that the local machine is able to contact the RSM port on the remote node. Open a command prompt on the local machine and type:
   `telnet <remote node name> 32958`
   If the terminal appears to hang, the connection was successful.
4. Make sure the user listed in the service is an administrator.
5. Make sure the COM engine is registered with the RSM Service. From the Windows menu, choose **Start** > **All Programs**> **Ansys EM Suite 2025 R2** > **Register with RSM** to register the engines.
6. If none of these steps fixes the problem, contact Ansys Support.

Error:

"Unable to locate or start COM engine on <remote node>: Engine is not registered with the Ansoft RSM service which is running on this machine."

Resolution:

1. To register the engine from the Windows menu, select **Start** > **All Programs** > **Ansys Electromagnetics Suite 2025 R2** > **Register with RSM**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Remote Solve Node = Linux

Error:

"Unable to locate or start COM engine on <remote node>: Unable to reach AnsoftRSMService. Check if the service is running and if the firewall allows communication."

Resolution:

1. Try disabling the firewall.
2. Confirm that you have not changed the Service Port in **Tools** > **Options** > **General Options** > **General** > **Remote Analysis**. If you have, change it back to the default (32958), restart Ansys Electromagnetics Desktop, and try to solve again.
3. Make sure that the local machine is able to contact the RSM port on the remote node. Open a command prompt on the local machine and type:
   `telnet <remote node name> 32958`
   If the terminal appears to hang, the connection was successful.
4. Check to make sure Remote Simulation Manager is running:
   a. Go to the rsm subdirectory of the Remote Simulation Manager installation directory, <RSM installdir>/rsm.
   b. Type
      `./ansoftrsmservice status`
   c. If the status query indicates that the service is stopped, type ./ansoftrsmservice start.
5. Make sure the COM engine is registered with RSM. Type:
   `./RegisterEnginesWithRSM.pl status`
   from within the Electronics Desktop installation directory. If the status query indicates "Not registered", type:
   `./RegisterEnginesWithRSM.pl add`
6. If none of these steps fixes the problem, contact Ansys Support.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Error:

"Unable to locate or start COM engine on <remote node>: Engine is not registered with the Ansoft RSM service which is running on this machine."

Resolution:

1. To register the engine, go to the Ansys Electromagnetics product installation directory and type:
   `./RegisterEnginesWithRSM.pl add`

Distributed Analysis

Distributed analysis allows users to split certain types of analyses and solve each portion of an analysis simultaneously on multiple machines. Simulation times can be greatly decreased by using this feature.

> **Note:** When using distributed analysis, the Temp folder must be accessible by all machines set up in the HPC configuration.

Maxwell supports the following form(s) of distributed analysis:

- Distributing rows of a parametric table, either as a regular DSO or as Large Scale DSO performed through command line.
- Distributing single or discrete interpolating sweeps.
- Maxwell 2D and Maxwell 3D designs support distribution of frequencies, transient solver, solution matrix and skew model.

> **Note:**
>
> Communication between machines in remote analysis and distributed analysis can drastically affect performance. Use of a high-speed network system, like Gigabit or Infiniband, is recommended for optimal performance.

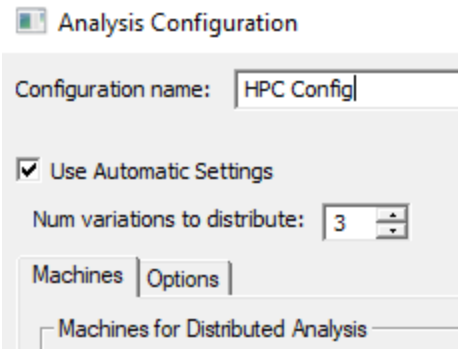
All types of distribution use MPI, except when only distributing rows of a parametric table, either as a regular DSO or as Large Scale DSO performed through command line. MPI may also be used if Auto mode is specified, and rows of a parametric table are distributed. MPI must be correctly configured if the distributed analysis uses MPI. See Setting HPC and Analysis Options for setting MPI Licensing and the appropriate Ansys Windows or Linux Installation Guide for details on installing MPI.

> **Note:** Maxwell Circuit and RMxprt designs do not support MPI.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Beta Feature: Parallel Component Mesh Adapt for 3D Component Array

If you enable the Beta Feature for Parallel Component Mesh Adapt, you can have parallel mesh adaption for 3D Component arrays. This feature works only with Use Automatic Settings enabled on the active Analysis Configuration.



When running a suitable design with this feature enabled, the Solution Profile will show multiple frequencies being adapted at the same time.

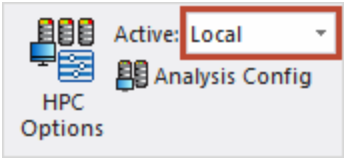
Configure a Distributed Analysis

To configure a distributed analysis, you must select a distributed machine configuration containing a list of machines to use for a simulation, based on memory and CPU considerations (See: Selecting an Optimal Configuration for Distributed Analysis). To create an new distributed machine configuration or to edit an existing one, see Editing Distributed Machine Configurations.

Before you can select a configuration, it must be active. See: Setting HPC and Analysis Options.

To select an existing, active configuration:

- From the **Simulation** tab, use the **Active** drop-down menu to select a configuration.



Distributed Analysis Configuration

After you have selected the active distributed machine configuration, you can edit its configuration in the **Analysis Configuration** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

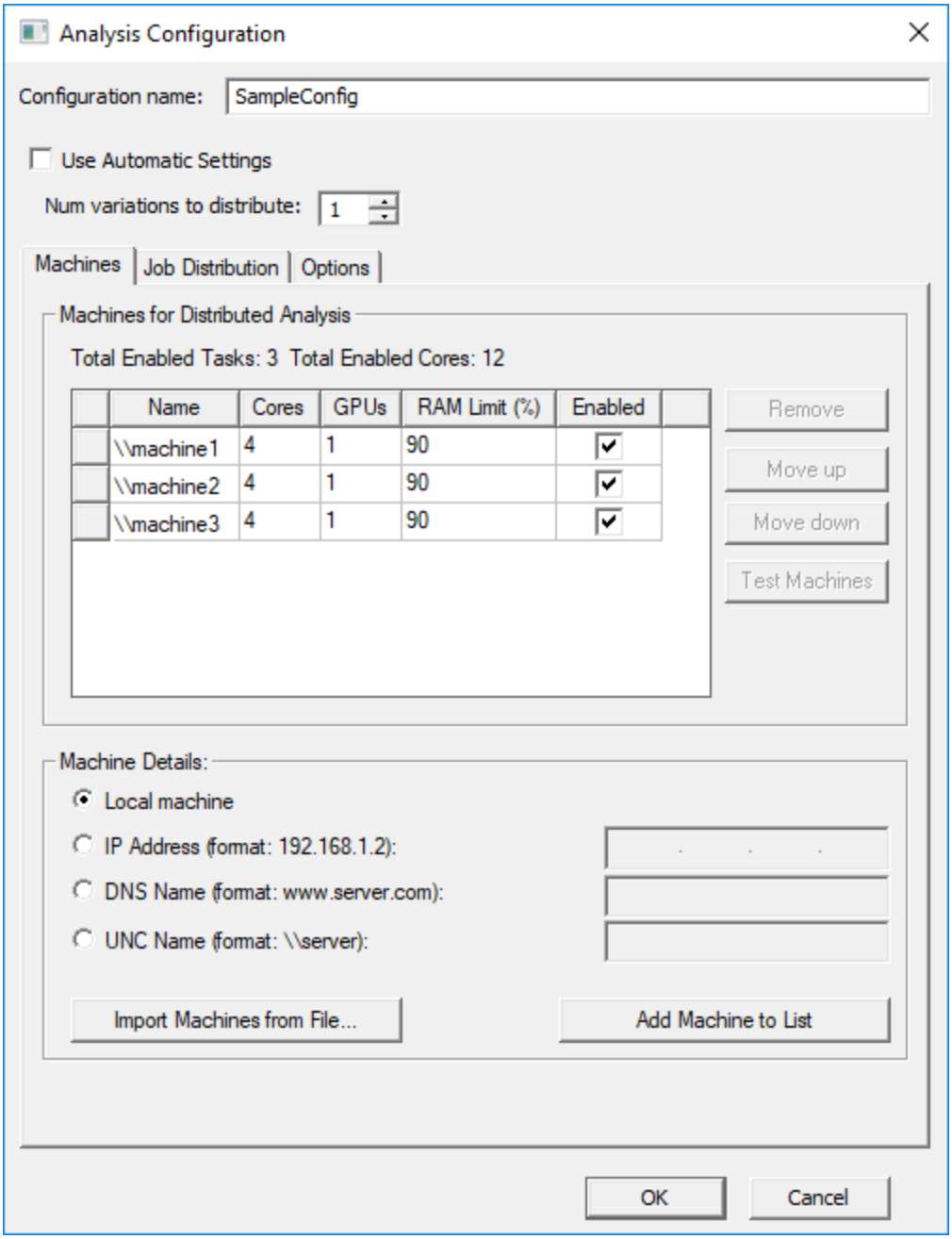
To edit the configuration:

1. Access the **Analysis Configuration** window one of three ways:
   - From the **Simulation** tab, click **Analysis Config**.
   - Click **Tools** > **Edit Active Analysis Configuration**.
   - From the **HPC and Analysis Options** window (opened from the **HPC Options** icon on the **Simulation** ribbon or **Tools** > **Options** > **HPC and Analysis Options**), select the configuration and click **Edit**.

   > **Note:**
   >
   > You can also access the Analysis Configuration window from the **HPC and Analysis Options** window by clicking **Add** or **Copy**. When using **Add** or **Copy**, the steps are the same as below, but you will need to specify a **Configuration Name**. The name cannot be empty and it cannot be a reserved name.

   The **Analysis Configuration** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> Available options may vary slightly depending on the solver and design.

2. If automatic settings are supported, the **Use Automatic Settings** check box allows you to select this option and set parameters based on the best use of available resources for the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

current analysis. Deselecting it enables the **Job Distribution** tab, where you can assign resources manually.

For Maxwell, this setting is currently only supported for 2D and 3D Transient and for 3D AC Magnetic designs.

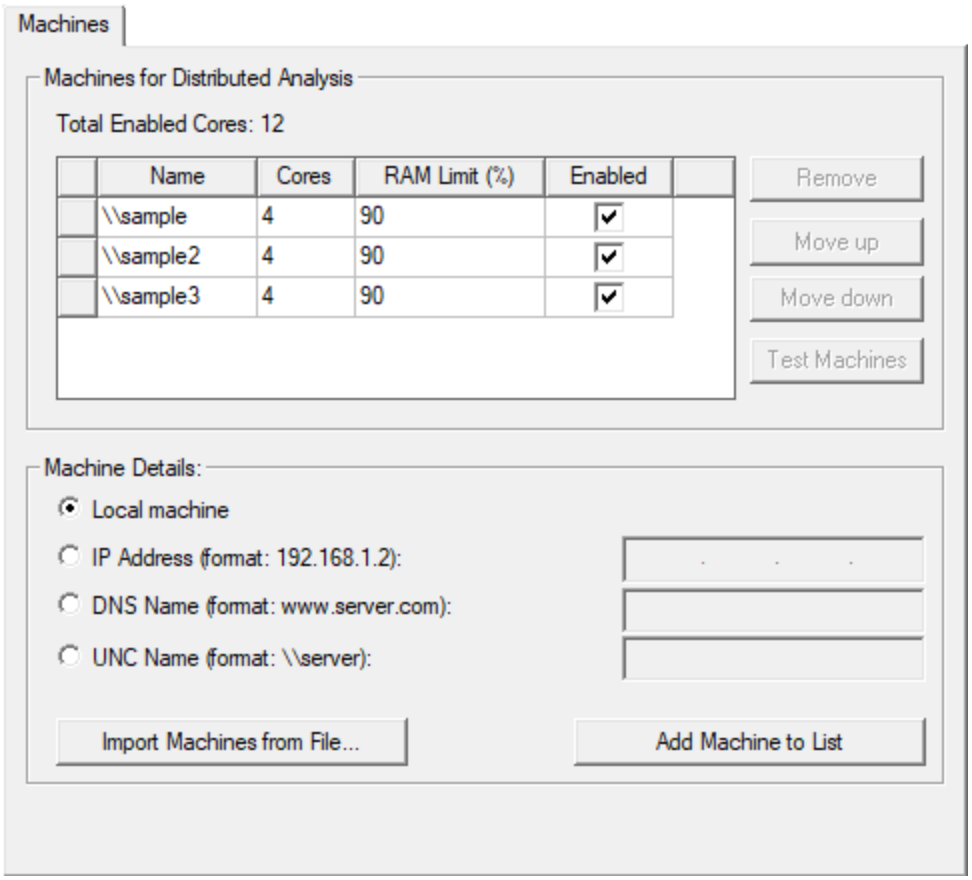
For 2D and 3D Transient designs, the **Use Automatic Settings** option distributes one excitation to each machine listed in the configuration. When this option is enabled, a minimum of 2 cores must be assigned. Each machine uses all cores assigned to it while solving its excitation. Unlike automatic frequency setup, the machines are assigned sequentially without regard to machine capability. For a 3D AC Magnetic design with **Use Automatic Settings**, you must assign a minimum of 4 cores.

3. Use the tabs to navigate through the rest of the settings:
    - **Machines** – contains the machine list. You can add, enable, remove, test, and reorder machines from the list.
    - **Job Distribution** – when **Use Automatic Settings** is deselected, this tab appears and allows you to select job distribution settings manually.
    - **Options** – allows you to specify additional options, depending on the design type.
4. Click **OK** to save the configuration settings.

Distributed Analysis Configuration - Machines Tab

On the **Analysis Configuration** window, the **Machines** tab allows you to provide machine information, either by specifying remote machine details, or by importing a list of machines from a file. You can then add, enable, remove, test, and reorder machines from the list.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Manually Adding a Machine

You can add the **Local machine**, or add a machine by **IP Address**, **DNS Name**, or **UNC Name**. Select the applicable radio button, enter the machine's information in the specified format, then click **Add Machine to List**.

> **Important:**
>
> The remote machines must have the same Ansys Electromagnetics Suite version installed on the same OS version, and have active RSM service.

The added machine(s) will appear under **Machines for Distributed Analysis**.

Importing a Machine List

You can import a machine list from a text file. Each line of the file should contain a machine's information in the following format:

```
<MachineName>:<NumTasks>:<NumCores>
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Machine List Details

The **Machines for Distributed Analysis** area lists machines in the order in which you entered them, irrespective of the load on the machines. To control the list order, select one or more machines, and use the **Move up** and **Move down** buttons. To remove one or more machines, select the machine(s) and click **Remove**.

The following columns can appear in the list:

- **Name** – the machine name.
- **Tasks** – this column allows you to specify the number of tasks a given machine will perform simultaneously. *Each separate solver or instance counts as one task.*

  If you have selected **Use Automatic Settings**, this column does not appear because there is no need for it.

- **Cores** – specifies the total number of cores that will be used on the given machine. The total number of Tasks and Cores are described just above the machine list.

  For distributed tasks, the software will allocate the total cores on a given machine to that machine's tasks. If a machine with 8 cores is running 2 distributed tasks, the software will automatically allocate 4 cores to each task. If it is running 4 distributed tasks, each gets 2 cores. And if it is running 3 distributed tasks, the first two tasks get 3 cores and the last task gets 2 cores. *The number of Cores must always be greater than or equal to the number of Tasks.*

  For a given variation (for example, frequency or geometry), you should make assignments so that each task has the same number of cores. This is because the solvers attempt to make each task computationally balanced. For example, with two machines, one with eight cores and another with four, assuming that the memory is proportionally equivalent, you could assign two tasks for machine 1, and one task for machine 2, giving all tasks the same number of cores.

- **GPUs** – supports GPU acceleration. See: Enabling GPU.
- **RAM Limit (%)** – specifies the maximum percentage of each machine's RAM you would like to be in use by the solver.
- **Enabled** – use the check boxes to enable or disable machines.

In general, Ansys Electromagnetics Suite solvers use machines in the distributed analysis machines list in the order in which they appear. If you select a distributed configuration (rather than Local) from the Toolbar menu and you launch multiple analyses from the same UI, Ansys Electromagnetics Suite solvers select the machines that are running the fewest number of engines in the order in which the machines appear in the list. For example, if the list contains 4 machines, and you launch a simulation that requires one machine, the solver chooses the first machine in the list. If another simulation is launched while the previous one is running, and this simulation requires two machines, the solver chooses machines 2 and 3 from the list. If the first

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

simulation then terminates and we launch another simulation requiring three machines, the solver chooses 1, 4, and 2 (in that order).

Testing Machines

When multiple users on a network are using distributed solve or remote solve, they should check the status of their machines before launching a simulation to ensure that no other Ansys EM processes are running on the machine. To do this, select one or more machines and click **Test Machines**.

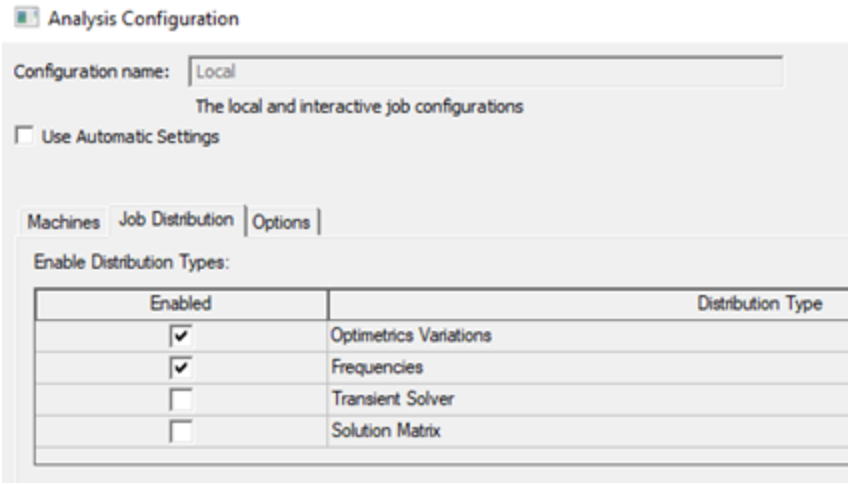
The **Test Machines** dialog box appears. The test goes through the current machine list and gives a report on the status of each machine. A progress bar shows testing progress. When the test is complete, click **OK** to close the dialog box. If you need to disable or enable machines based on the report, you can do so in the **Analysis Configuration** window.

Distributed Analysis Configuration - Job Distribution Tab

On the **Analysis Configuration** window, the **Job Distribution** tab allows you to manually enable specific types of job distribution and enable multi-level solves.

The **Job Distribution** tab is disabled if you selected **Use Automatic Settings**.

> **Note:**
>
> Different design types have different job distribution types.



The job distribution list allows you to specify which job distribution types to allow for the current analysis configuration. Use the check boxes enable and disable distribution types. At solve time, Maxwell automatically selects the best distribution type from the enabled types.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enabling a distribution type does not mean it will be used. It must be also allowed by the solve setup. If you enable a distribution type for a given setup, and distribution is allowed, the preview window updates to describe the distributions. Note that enabled distribution types apply to all setups of the given design type, so it is possible for different setups in a design to be solved using different distribution types.

For example, the **Transient Solver** distribution type that enables use of the Time Decomposition Method, will only be *used* with Maxwell 2D and Maxwell 3D transient designs. Note that the enabled distribution types will apply to all setups of the given design type, so it is possible for different setups in a design to be solved using different distribution types. The **Solution Matrix** distribution type that enables use of the Matrix-based Domain Decomposition Method will be *used* with both 3D AC magnetic designs and 3D magnetostatic designs.

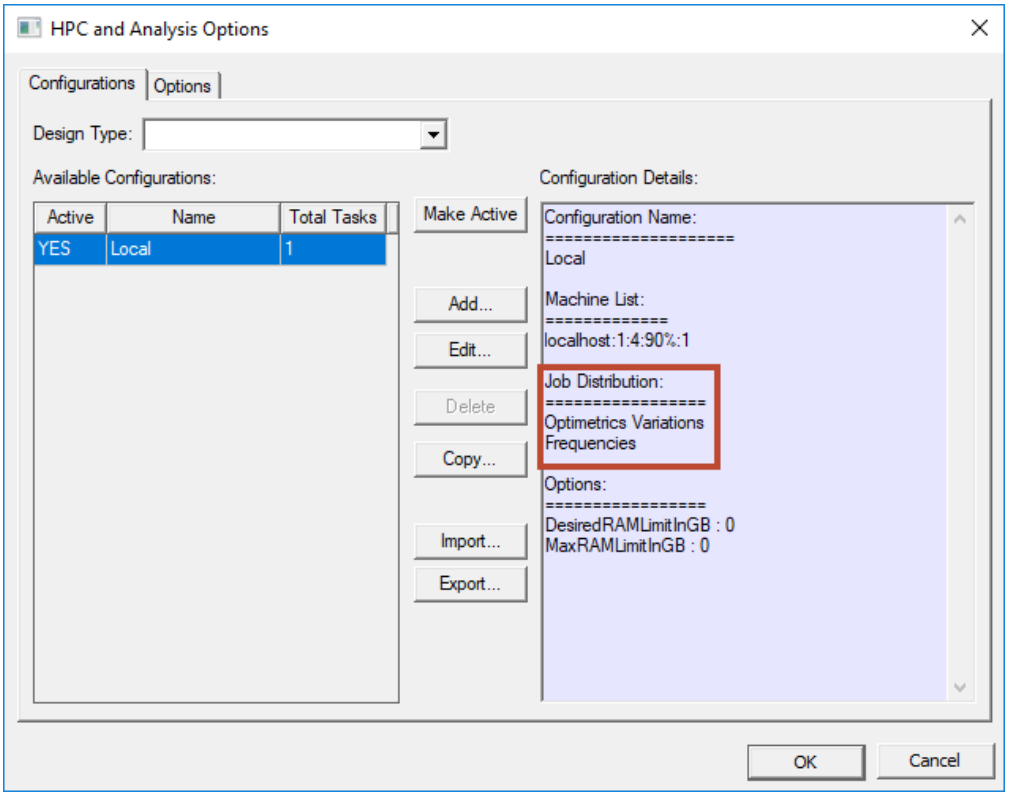
The concurrent initial mesh generation workflow with Distributed Mesh Assembly relies on the MPI based distribution technology inside the MeshAssemblyManager. The decision whether to launch sequential or parallel mesh generation is based on the combination of the number of individual meshes present, the HPC setting, and the number of tasks available.

- If there's only one single mesh or if curvilinear contact exists: then normal initial mesh workflow with single G3dmesher will be launched.
- If there are more than one individual mesh and no curvilinear contact exists:
    - If Use Automatic Setting is not selected, under the **Job Distribution** tab, if "Mesh Assembly" is not enabled, then meshes will be generated sequentially. If "Mesh Assembly" is enabled and user assigns more than 1 tasks, then parallel mesh generation will be launched.
    - If Use Automatic Setting is selected and user assigns more than 1 core, then parallel mesh generation will be launched.
    - If Use Automatic Setting, the number of MPI tasks will be the number of cores user assigned or the number of geometries to be meshed, whichever is smaller. If there are more geometries than cores, the geometries will be dynamically assigned to tasks. If there are more cores than geometries, the remaining available cores will provide some of the tasks with multi-thread capabilities.

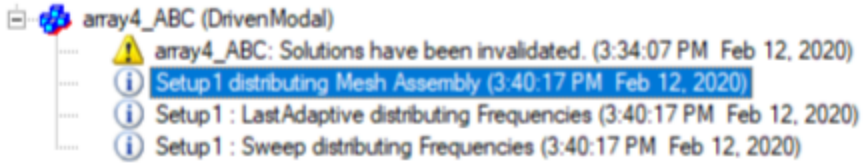
All the detailed progress information from the Mesher is suppressed and the progress will report the number of meshes being finished. All the mesh profiles will be available under profile report and the mesh feedback will also be available under mesh feedback tool.

Enabled distribution types are listed when you select a configuration in the **HPC and Analysis Options** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

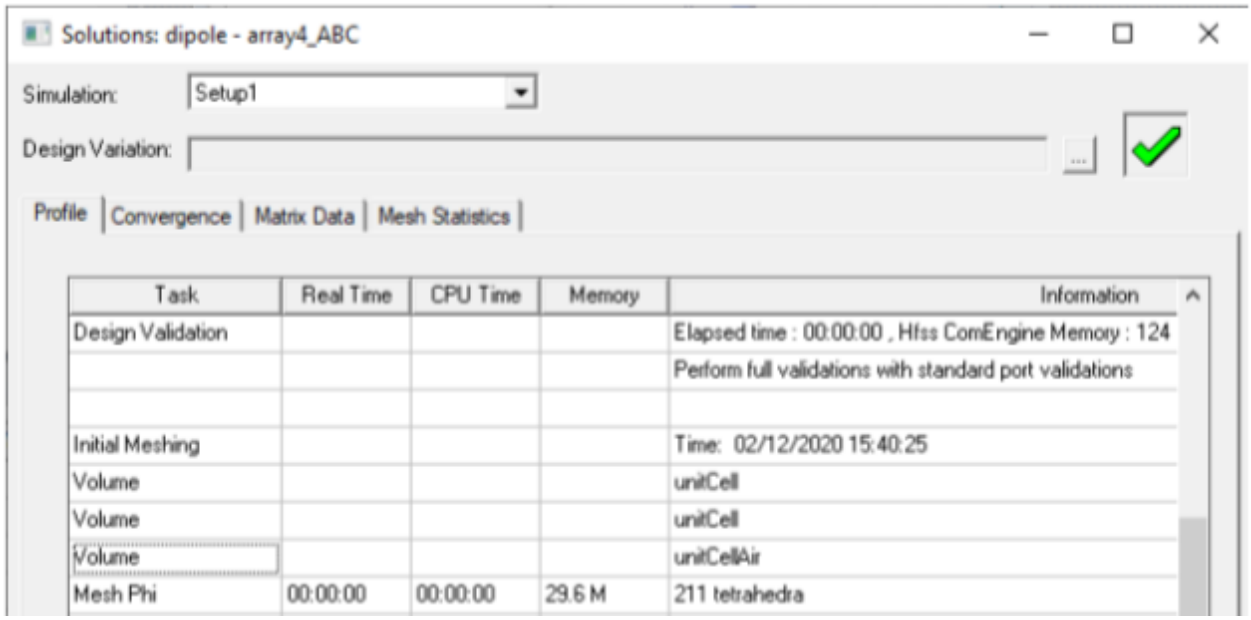


When you run a simulation, the **Messages** window describes distributions.



When you view the Solution Profile, distributions for unitCell show as parallel Volume Tasks.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



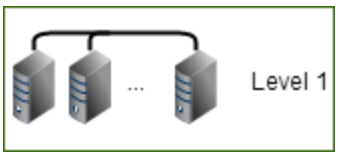
Distribution Levels

For products and designs that support two-level distribution, you can select either single or two-level distribution.

If you select single level, one distribution type will be applied at each stage of the solution process. If multiple types are available, the higher-level solution will generally be distributed. All machine tasks will be used by the single-level distribution.

Single-level Distributions

In a single-level distribution, one distribution type is applied at each stage of the solution process. Common stages include LastAdaptive, Sweep, and Parametric. All machine tasks will be used by the single-level distribution.



Supported distribution types include:

- Optimetrics Variations
- Frequencies
- Mesh Assembly

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Transient Solver
- Solution Matrix

Solver distributions require MPI. See: Setting HPC and Analysis Options.

Parallel distribution types (such as Optimetrics Variations, and Frequencies) do not require distribution. If these types are not able to distribute, the simulation can run sequentially.

Memory distribution types (such as Transient Solver and Solution Matrix) are set to require distribution. If these types are enabled, the software will assume that distribution is necessary to extend the simulation scale or add fundamental solution capabilities.

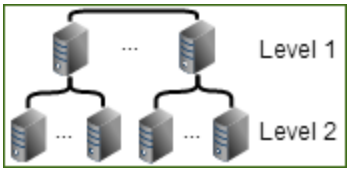
When multiple distribution types are available, the higher level solution will generally be distributed. The priority order is : Optimetrics Variations, Transient Solver, Frequencies, Mesh Assembly and Solution Matrix. If Iterative Solver is enabled in the AC Magnetic solve setup, the priority order will be: Solution Matrix, Optimetrics Variations, Transient Solver, and Frequencies. For example, Optimetrics Variations will be distributed when both Optimetrics Variations and Transient Solver are selected.

Two-Level Distributions

Selecting **Enable two-level** enables the **Distributed solutions at first level** box.

In a two-level distribution, the first level distributes the specified number of solutions. Each solution will then use a subset of machine tasks to distribute the second level. A solver distribution type must be available for the second level; otherwise, single-level distribution will be applied. The Transient Solver distribution type must be available for the second level for a Transient design; otherwise single-level distribution will be applied. The Solution Matrix or Frequencies distribution type must be available for the second level for an AC magnetic design; otherwise single-level distribution will be applied.

For two-level distribution, the total number of tasks must be greater than or equal to the number of tasks for level 1.



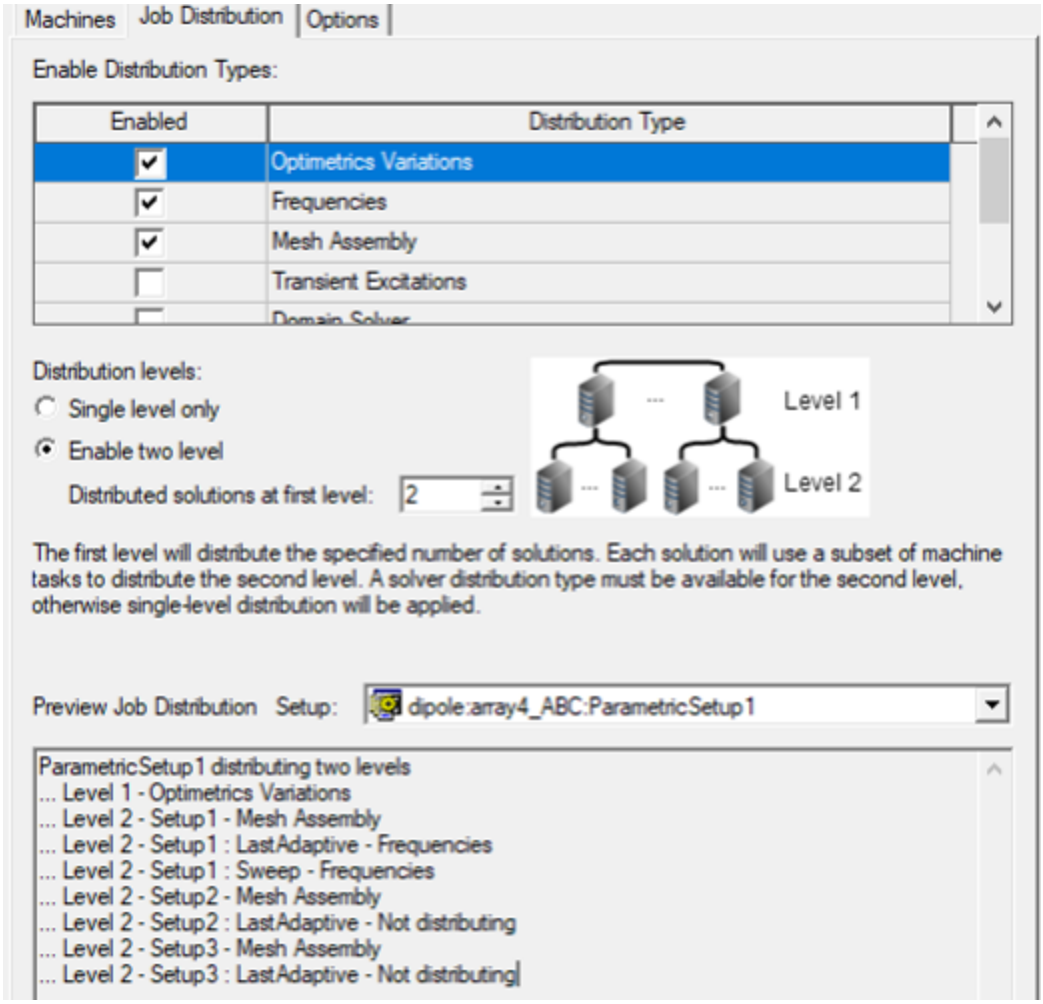
The following is an example of a two-level distribution:

- A parametric setup distributing **Optimetrics Variations** as level 1, and a **Frequencies** distribution as level 2.
- A parametric setup distributing Optimetrics variations as level 1, and Mesh Assembly distribution as level 2.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Analysis Configuration** window displays a preview of the job distribution, if allowed for that configuration:

Here is another example:



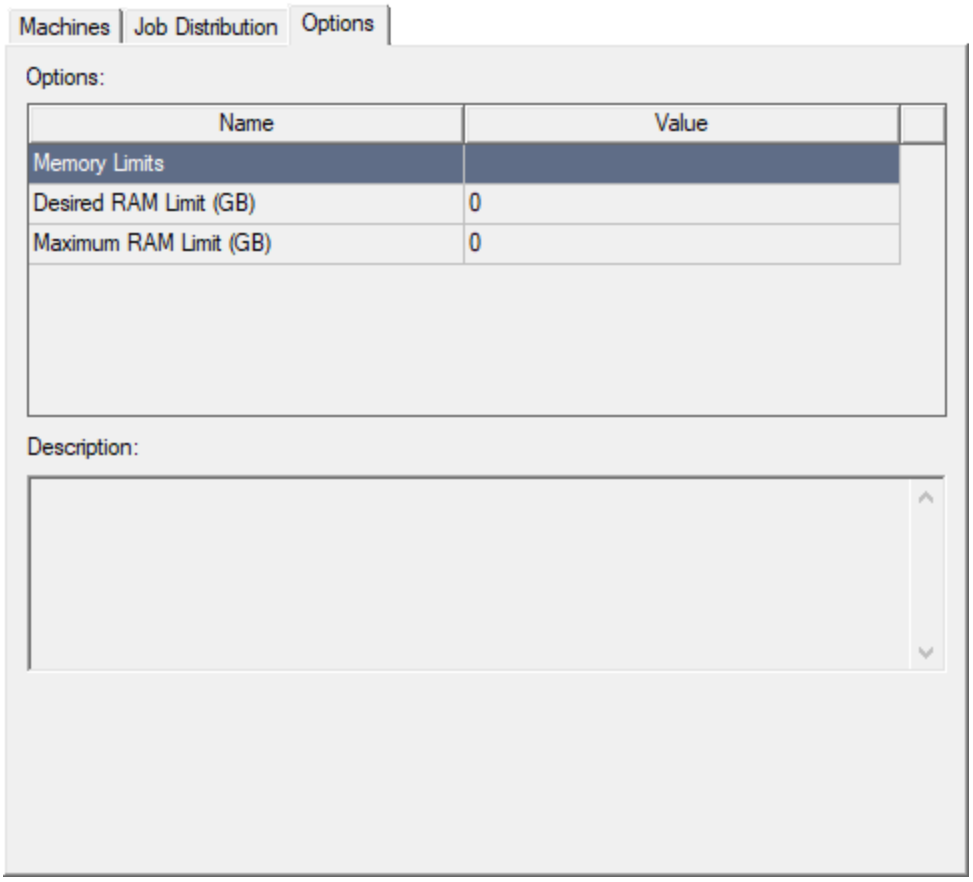
For more information, see: Two-Level Distribution Guidelines.

Distributed Analysis Configuration - Options Tab

On the **Analysis Configuration** window, the **Options** tab allows you to specify options for the current analysis configuration.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Different design types have different options.



These options settings will be in effect only when all the following are true:

- A design is being solved whose design type matches the analysis configuration design type.
- The analysis configuration is the active configuration for its design type.
- You have not specified corresponding `batchoptions` on the command line.

Relation to Batchoptions

Command line `batchoptions` can be used to override the options specified by the active configuration.

Analysis configuration option settings can be overwritten by specifying the option name and value inside a `-batchoption` string.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To add batchoptions from the UI, click **Maxwell** > **Submit Job**. Under **Analysis Options**, click **Add**. This opens the **Add Batchoption** window.

See: Running Electronics Desktop from the Command Line and Batchoptions Command Line Examples.

Adding Configurations or Accepting Edits

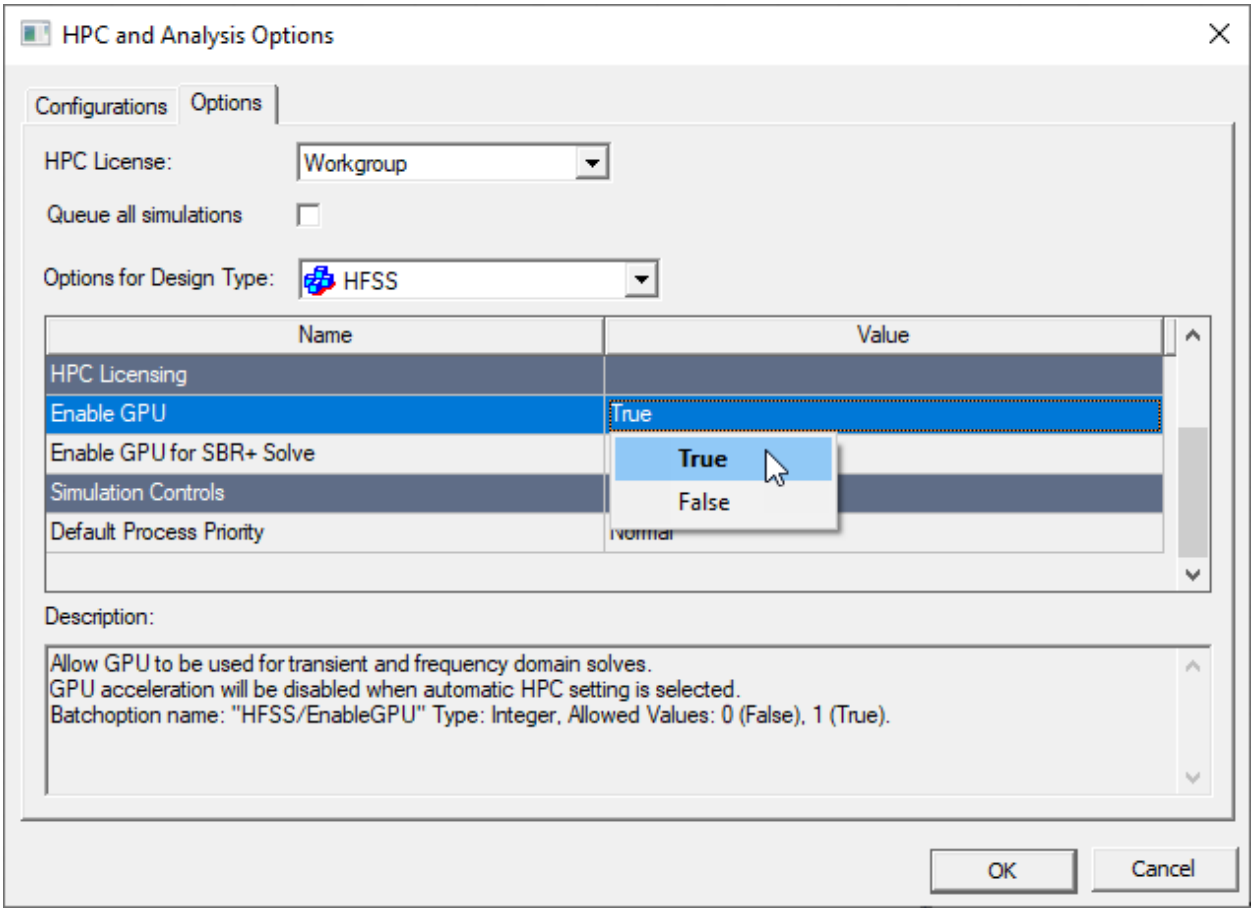
Click **OK** to accept the changes and close the **Analysis Configuration** window. Only machines marked **Enabled** appear in the machine list.

Regardless of the machine(s) on which the analysis is actually run, the number of processors and RAM Limit (%) settings, and the default process priority settings are now read from the machine from which you launch the analysis. See: Setting HPC and Analysis Options.

Enabling GPU

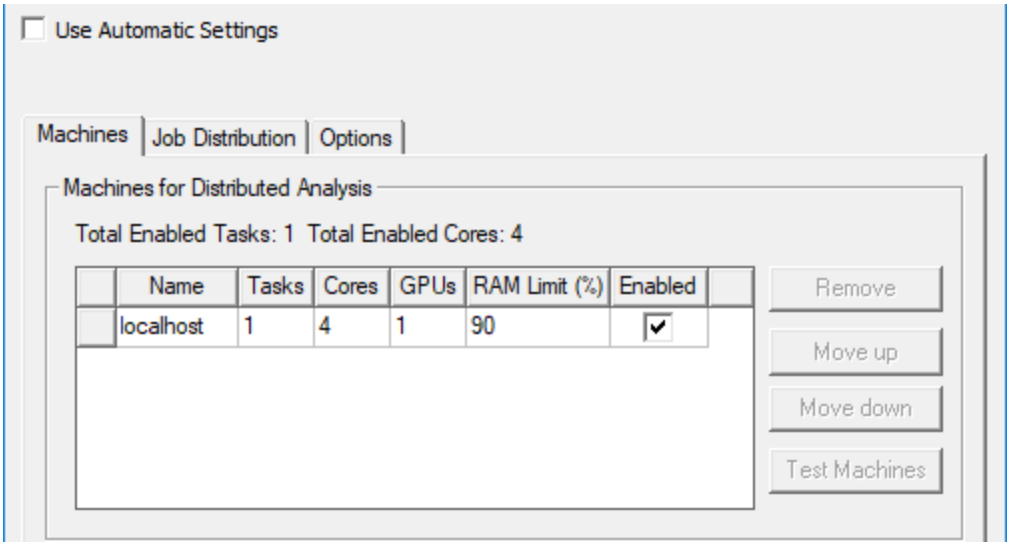
Ansys Electronics Desktop supports GPU acceleration for transient, frequency domain, and Maxwell 3D AC magnetic matrix simulations. For details on the requirements for GPU use, see: GPU Acceleration.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To enable use of GPU or MIC acceleration, you can set **Enable GPU** to **True** and set **Use Automatic Settings** to **off** in the window. The matrix solver automatically determines if all cores should be used, or if one GPU should be used to give the best performance. For example, for a Maxwell AC Magnetic Solution Matrix, if you specify 4 cores for the simulation, the 3D AC magnetic solver will use 4 cores in parallel during matrix assembly while the matrix solver will use either 4 cores or 1 GPU.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For solvers that can use GPUs, when you uncheck **Use Automatic Settings**, the analysis configuration lets you specify the number of GPUs to be used by each machine, along with the existing settings for cores and RAM Limit. This makes sense, because GPUs, cores, and RAM are all resources that the user is managing, and GPUs and cores both contribute to HPC count. Each machinehas tasks and GPUs. The GPUs are distributed among the tasks, similarly to how cores are distributed. For example, if a machine has 3 tasks and 2 GPUs, then the first two tasks will get one GPU each, and the third task will get no GPU.

For design types like HFSS that support GPU for different types of solve (frequency, transient, SBR+), the GPU setting when **Use Automatic settings** is unchecked applies to all types. To use GPU for some solves and not for others, you can create multiple configurations.

Determining When a GPU or MIC is Being Used

You can determine if a GPU or MIC is being used for acceleration by viewing the **Solutions** dialog box, **Profile** tab. If a GPU is successfully locked for use by the solver process, the profile shows the GPU's CUDA device ID and its name.

If the GPU is not used, the Solution Profile indicates that, and the CPU cores are used.

When GPU is used:

- GPU must be enabled for HFSS/Maxwell.
- Both Windows and Linux are supported.
- Only complex symmetric matrices can be solved by CUDA.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The matrix (number of unknowns to be solved) must be large enough. By default, its dimension should be larger than 2,000,000.
- The times using CUDA and CPU only are estimated. If GPU is faster, it will be used. If not, the solver falls back to multi-core CPU.

GPU and CPU:

- The estimation is based on the structure of the matrix, not just the dimension.
- The generation and model of GPU are considered. The newer the model is, the faster the GPU estimation is.
- The clock rate and number of cores of CPU are considered. The higher the rate is, the more the cores are, the faster the CPU estimation is.
- Finally, the faster device (GPU or CPU) is selected based on the estimation.

Why GPU is Not Used

- The estimation is based on the whole matrix, not just the size. Having larger matrices doesn't mean GPU will be used.
- The bandwidth is not useful either. It is possible to have two matrices with exactly the same size and bandwidth, one favoring GPU and the other CPU.
- If GPU is of older models while CPU is not, GPU may not be used.
- If CPU has high clock rate, GPU may not be used.
- If CPU has many cores, GPU may not be used.
- If Use Automatic settings in Analysis Configuration is checked.
- Nvidia GPU drivers are downloaded from certain repository (e.g. RedHat) instead of directly from Nvidia website

Multiple GPUs

When there are multiple GPU cards in the same machine, the first solver process (hf3d, 3dedy) locks the first GPU card available (not locked or used by other processes, not used for display, etc.), the second process locks the second card available, and so on. After all cards are locked, solver uses CPU only.

Selecting Optimal Configurations for Distributed Analysis

This section provides some basic guidelines for configuring a distributed analysis, including how to choose the number of tasks for Distributed Direct and Distributed Iterative solvers, and when to use two-level distribution. In this section, assume that a fixed number of machines and cores are available, and that the goal is to use them as efficiently as possible.

In the majority of situations, configuring a Distributed Solution Option (DSO) in combination with the multiprocessing option to take optimal advantage of the available hardware results in improved speed.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

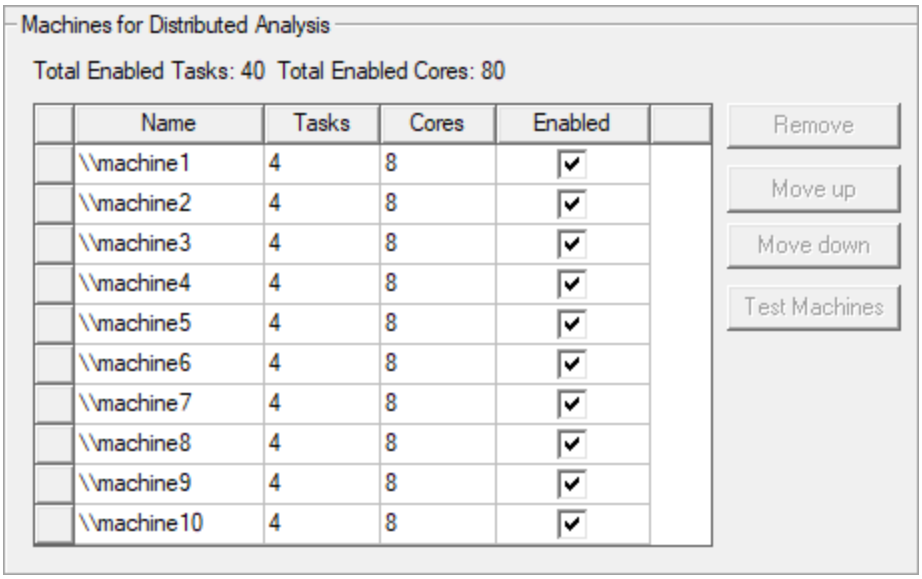
Note:

This topic assumes that there is enough hard drive space and memory for DSO simulations. For multiple DSO simulations on a single machine, the total memory needed is the sum of the memory used by each simulation. For example, consider a discrete frequency sweep in which each frequency point needs 3.5GB. A quad core system with 8GB of RAM would rely heavily on swap, which is highly inefficient. In this case, updating the number of tasks in the machine list is ideal.

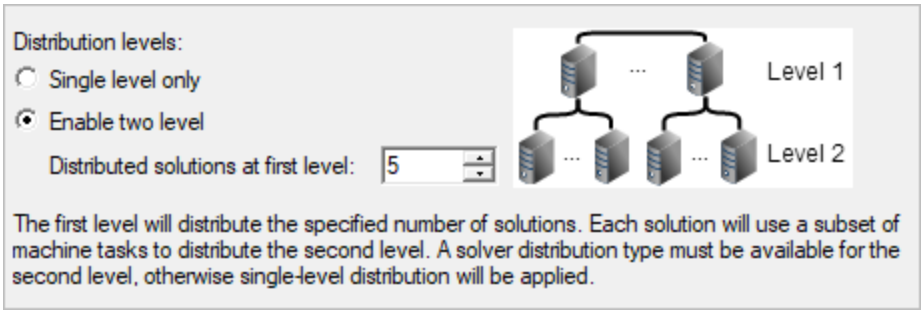
Two-Level Distribution Guidelines

Ansys recommends that you focus resources for the highest level of distribution (level one). In other words, allocate the resources so that a maximum number of frequencies or variations are solved in parallel. First, determine the minimum number of machines required to solve one frequency or variation based on your prior knowledge of the type of model being solved. Then allocate the resources so that a maximum number of solutions can be solved in parallel.

For example, suppose that a given model requires at least 100 GB of memory to solve using DDM. Assume that you have 10 machines, each with 60 GB and 8 cores. One solution will require two machines for the DDM solver. This means that, at most, 5 frequencies can be distributed. Each DDM solution requires at least 3 tasks, but since more cores are available you should set 4 tasks per machine, resulting in 2 cores for each Domain . A machine configuration based on the machines and number of tasks described above is shown in the figures below.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Distributed Direct Solver Guidelines for HPC Configuration

The dependence of Distributed Direct solver on number of tasks is very similar to the Maxwell solver. For distributed direct solver, the number of tasks increases the total memory but also typically increases the speed of the simulation.

Some general guidelines:

1. If memory usage is critical, use 1 task/machine using all cores.
2. If memory usage is not critical, use 4 cores/task for optimal speed.

Distributed Iterative Solver Guidelines for HPC Configuration

The distributed iterative solver distributes excitations for faster simulations and to access less memory to solve a large problem. Each task corresponds to a number of excitations and, therefore, more excitations are solved in parallel when the number of tasks increases.

The first consideration for the number of tasks is the memory requirement. Each task uses a similar amount of memory, and the total memory usage doubles as the number of tasks doubles. Thus the number of tasks on each compute node should be restricted to avoid an "Out of Memory" failure.

The second consideration for the number of tasks is the number of excitations. Maxwell designates one of the tasks as the master, which does not participate in the iterative process. ***Thus the number of tasks should be no more than the Total Number of Excitations + 1***.

In summary, assuming that you have enough memory, if the number of excitations N is less than the total number of cores M and you choose 4 cores per task, then the number of tasks is ceil (N/4) + 1, where ceil() stands for the ceiling function and the extra "1" is dedicated to the master task; if the number of excitations N is more than the total number of cores M and you choose 4 cores per task, then the number of tasks is ceil(M/4). For example, if you have a design with 64 excitations and you have 32 cores available, you could define a possible setup having 8 tasks with 4 cores per task.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Large Scale DSO for Parametric Analysis

Large Scale DSO for parametric analysis operates through a non-graphical batch application called desktopjob. You can run the desktopjob command line to perform parametric analysis DSO. The command-line interface supported by this batch program is similar to the command line used for regular DSO jobs.

Large Scale DSO is used for large scale parallel jobs, which either fail or scale poorly as regular DSO jobs. A Large Scale DSO job does not support the output of full parametric results, but produces reduced datasets corresponding to predefined rectangular plots. The extracted columns of data are saved as CSV files. Typically, there is one CSV file per-trace, per-variation. These CSV outputs can be used directly in downstream applications (for example, Microsoft Excel). They can also be imported as dataset solutions for post-processing. Non-rectangular plots of the design (such as statistical eye or digital plot) are not extracted. In order to produce a new output you must rerun the analysis.

> **Note:**
>
> For a machine with *n* cores, it should be expected that running *n*, single core, distributed simulations in parallel will encounter additional overhead due to the need to spawn *n* unique solve processes. Therefore, it should not be expected to observe an *n* times speed up over the time taken to run *n* analyses in series with a single core. The relative impact of this overhead increases as the size of the simulation and time to complete a single solve shrink.

The basic Large Scale DSO process involves

1. Preparing the model for Large Scale DSO Analysis.
2. Submitting the Large Scale DSO job through **Tools** > **Job Management** or via command line.
3. Monitoring the job's progress.
4. Postprocessing the results.

For details, refer to the following sections:

- Prerequisites for Large Scale DSO
- Job Management Interface for Large Scale DSO
- Command Line Syntax
- Deployment/Configuration
- Tutorial Example for HFSS
- Job Outputs
- Job Monitoring

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Process Limits for Large Scale DSO on Windows
- Known Issues/Troubleshooting for Large Scale DSO

Large Scale Distributed Solve Operation could submit a parametric setup to be solved in multiple machines, each machine may launch multiple EM-Desktop processes to solve the assigned variations (Design Points). Variations are distributed to each task (EM-Desktop process) equally, regardless of the machine hardware and each variation's complexity. In practice, some tasks may finish earlier than others, in some extreme case some tasks may hours behind fastest task. DSO can redistribute tasks when a task finishes before other task. Variations are removed from slow tasks and reassigned to fast tasks. If you abort a task, they can be reassigned to the running task, when the running task finish its original assignment. For more information, see Large Scale DSO theory.

Large Scale DSO offers two new batchoptions related to the redistribution ability.

LargeScaleDSO/VarRedistribution, where 0 disables redistribution (default), and 1 enables it.

LargeScaleDSO/RedistributionLimit, is a positive integer specifying the minimum estimated remaining time (in minutes) for variations to redistribute to another task. The default is 3.

Aborting a Large Scale DSO Simulation

To abort the whole Job, select the **Abort** button on the Job Monitor dialog.

To abort using the Job progress bar, click the button next to the Job progress bar, Click the **Abort** menu item in the popup.

To abort all tasks in a Node(host), click the button next to the node progress bar., and click the **Abort** menu item in the popup. Aborted Variations will be redistributed to other running Nodes, if redistribution is turned on.

To abort an individual task, click the button next to the node progress bar, and click the "Detail" menu item in the popup. In the Task status dialog box, click the **Task** button in the grid. In the variation status dialog box, click **Abort** button. Aborted Variations will be redistributed to other running tasks, if redistribution is turned on.

To terminate a hanging EM-Desktop process, the hanging ansysedt.exe process won't respond to the first abort command. Send a second abort command to terminate the hanging process. Remaining variations will be redistributed to other running tasks, if redistribution is turned on.

For an EM-Desktop process crashed or killed by Windows Task manager or other tools, the task status will be shown as aborted, Remaining variations will be redistributed to other running tasks, if redistribution is turned on.

Prerequisites for Large Scale DSO

General Prerequisites

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Ansys Electronics Desktop must be installed on the cluster which runs either a supported scheduler or Ansys RSM.
- The cluster is compatible with Large Scale DSO requirements.
- Every node of the cluster supports the disk space (in temp directory) and memory requirements of multiple engines that run in parallel.
- All the machines allocated to Large Scale DSO job must all come from the same platform, either Windows or LINUX.
- On the Windows platform, Ansys RSM is started as an admin account, rather than as a system account.

  > **Important:**
  >
  > Large Scale DSO does not support RSM Service running with system login credentials.

- On each machine of the cluster, the 'desktopjob' application is registered with Ansys RSM service using the command shown below:

  Windows: `<installation-directory>/<platform> desktopjob.exe -regserver`

  LINUX: `<installation-directory>/<platform> desktopjob -regserver`

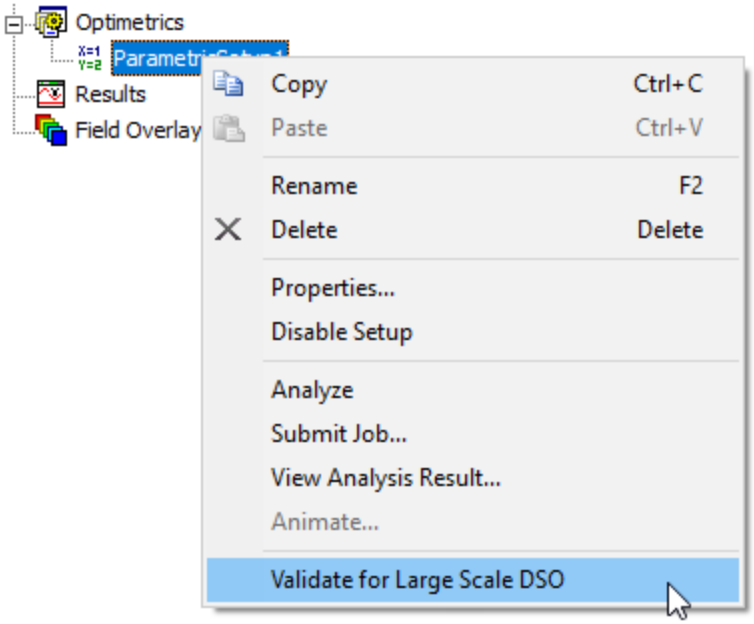
Job Management Interface for Large Scale DSO

Large Scale DSO jobs run only in non-graphical batch mode, irrespective of the scheduler environment. This is in contrast to a Regular DSO job, which can run in graphical mode. This consideration implies that projects corresponding to Large Scale DSO job must be saved and closed prior to job submission. Secondly, the command to submit a Large Scale DSO job is only available through **Tools** > **Job Management** or via a command window, while a Regular DSO job can be run in an RSM environment by right-clicking directly on the parametric setup. The Job Management window is accessed by running Ansys Electromagnetics product Desktop on the designated Postprocessing Node of the cluster. The Desktop provides UI commands for scheduler selection, job submission and job monitoring/control.

When you have selected the scheduler, perform the following steps to submit a Large Scale DSO job:

1. Set up and prepare the model on your local workstation. Right-click the desired solution setup and select **Validate for Large Scale DSO**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Correct any errors and save the project.
3. Close Electronics Desktop.
4. Copy the project (or folder, if the project references external files) from a personal workstation to a shared drive on cluster.

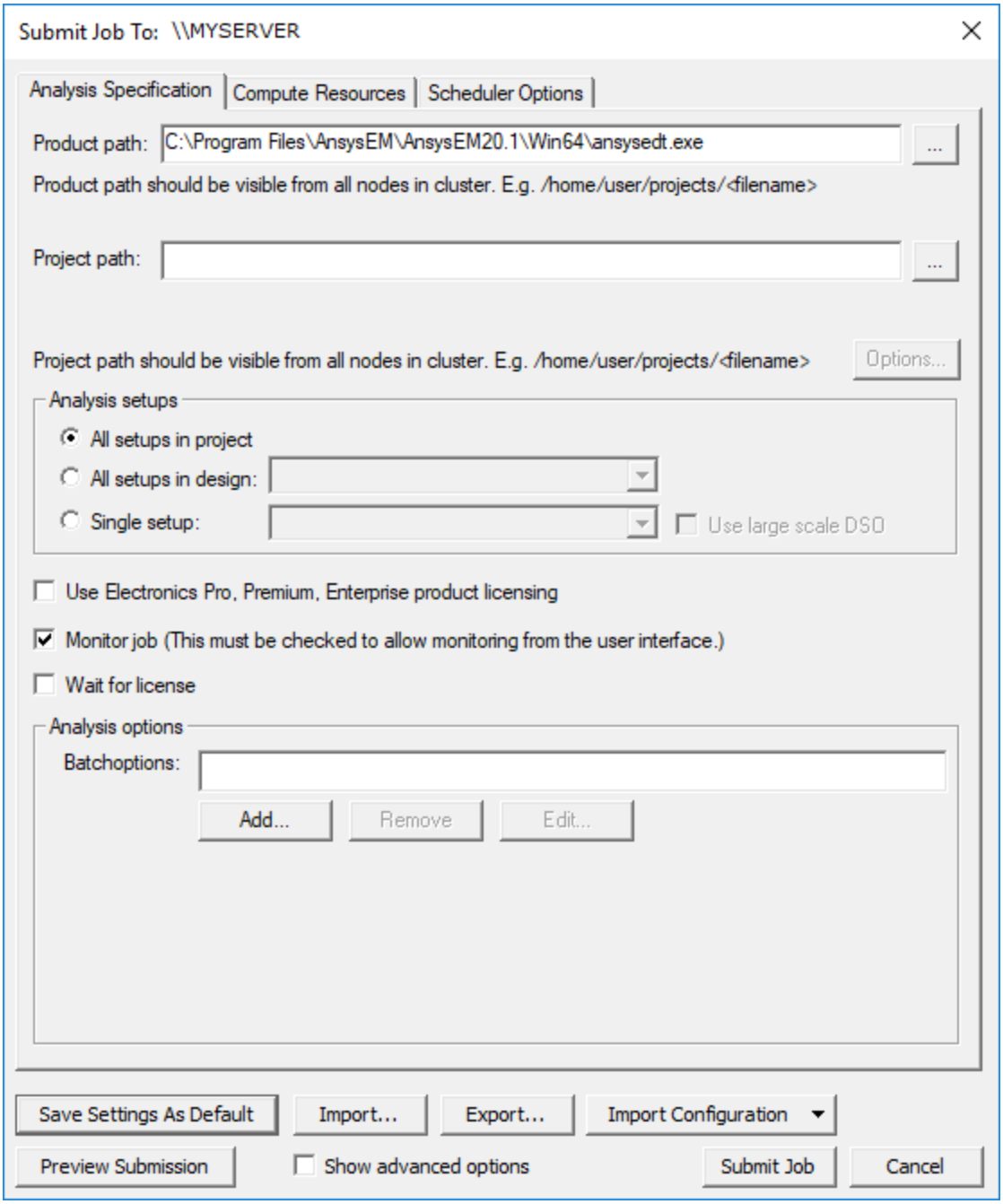
   In the RSM environment, you must specify a machine list (See: HPC and Analysis Options).

   In a Linux scheduler environment, a cluster must have a designated Postprocessing Node.

5. Open a remote desktop session (or equivalent) on the node corresponding to the first machine in the machine list (the designated Postprocessing Node on Linux).
6. Launch Electronics Desktop on that node and open the project.
7. Verify that the model has been prepared correctly.
8. Close the project.
9. Submit the job one of several ways:
   - Click **Tools** > **Job Management** > **Submit Job**.
   - Click **Project** > **Submit Job**.
   - Click **Maxwell 3D**, **Maxwell 2D**, or **RMxprt** > **Submit Job**.
   - Select the **Simulation** tab and click the **Submit** icon.

   The **Submit Job To** window appears, on the **Analysis Specification** tab.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

Options vary slightly depending on the selected scheduler. See: HPC Integration.

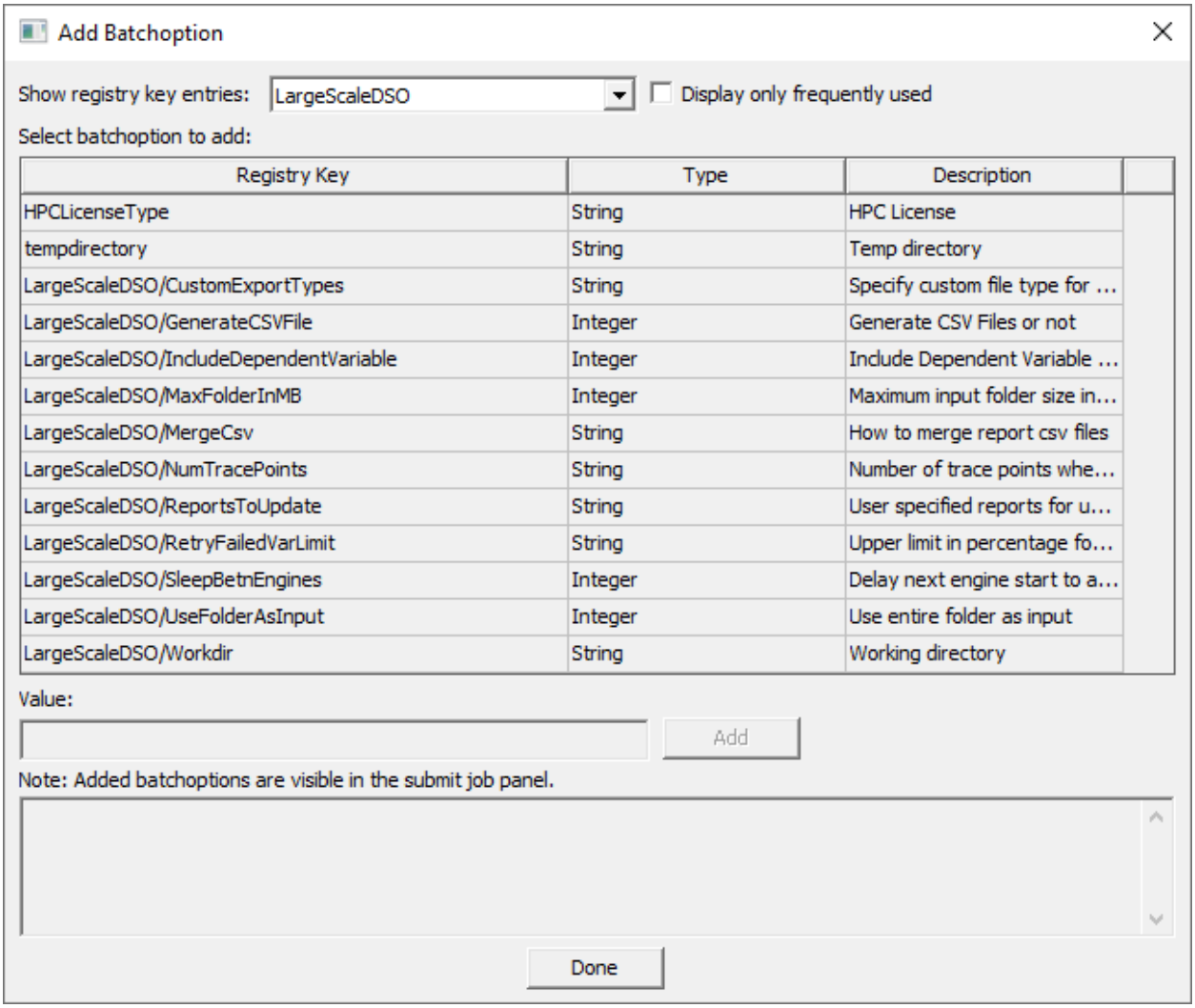
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10. Specify information for all fields:
    - Most options will not be enabled until you select a **Project path**.
    - Under **Analysis setups**, select setups for analysis.

      For Large Scale DSO, select a **Single setup** from the drop-down menu and ensure the **Use large scale DSO** check box is selected.

    - Determine whether to **Use Electronics Pro, Premium, Enterprise product licensing**, whether to **Monitor job**, and whether to **Wait for license.** If you wish to monitor jobs through **Tools** > **Job Management**, you must enable job monitoring.
    - Add **Batchoptions**, if desired.

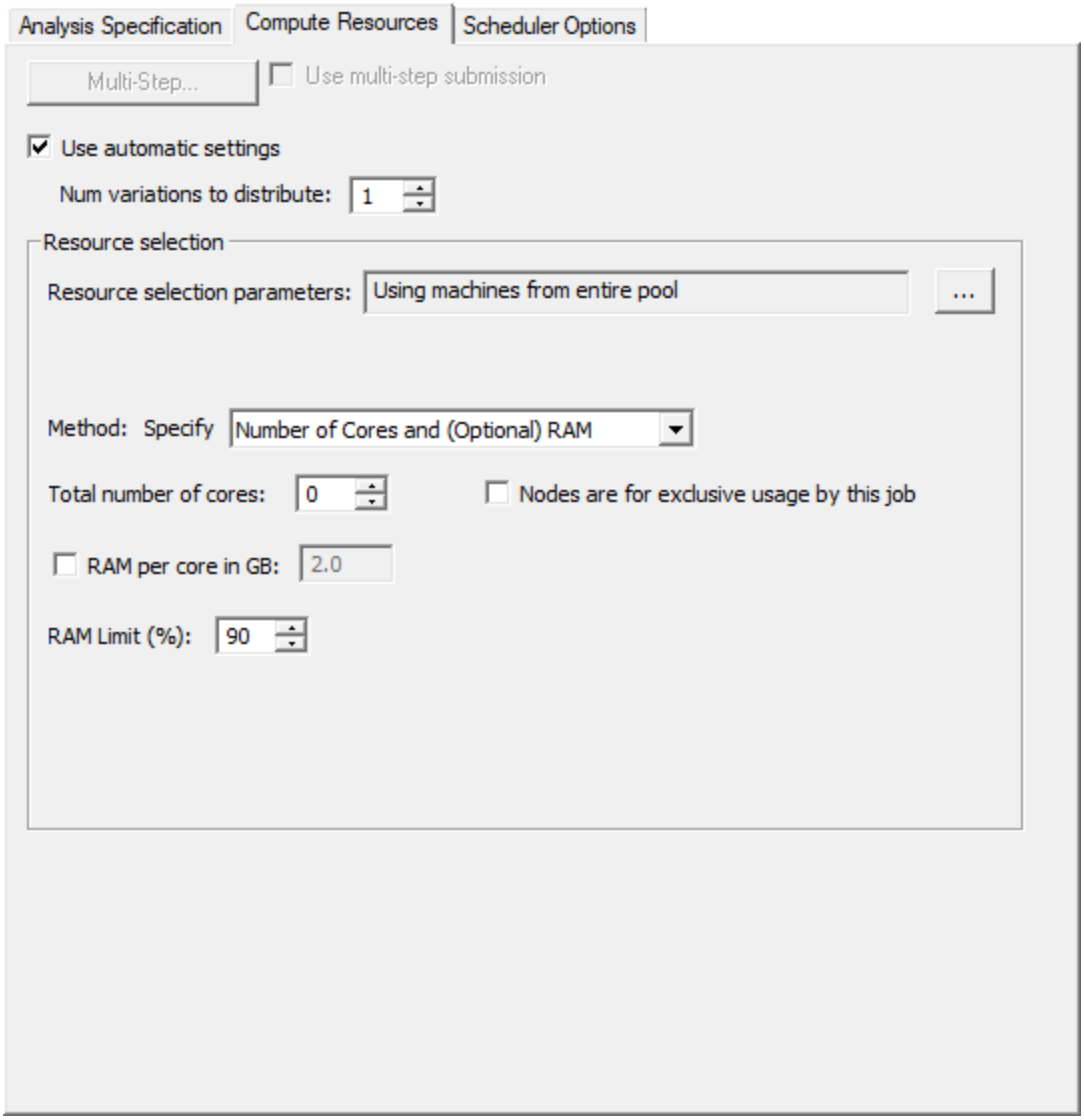
      The following shows batchoptions specific to Large Scale DSO.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11. Select the **Compute Resources** tab.

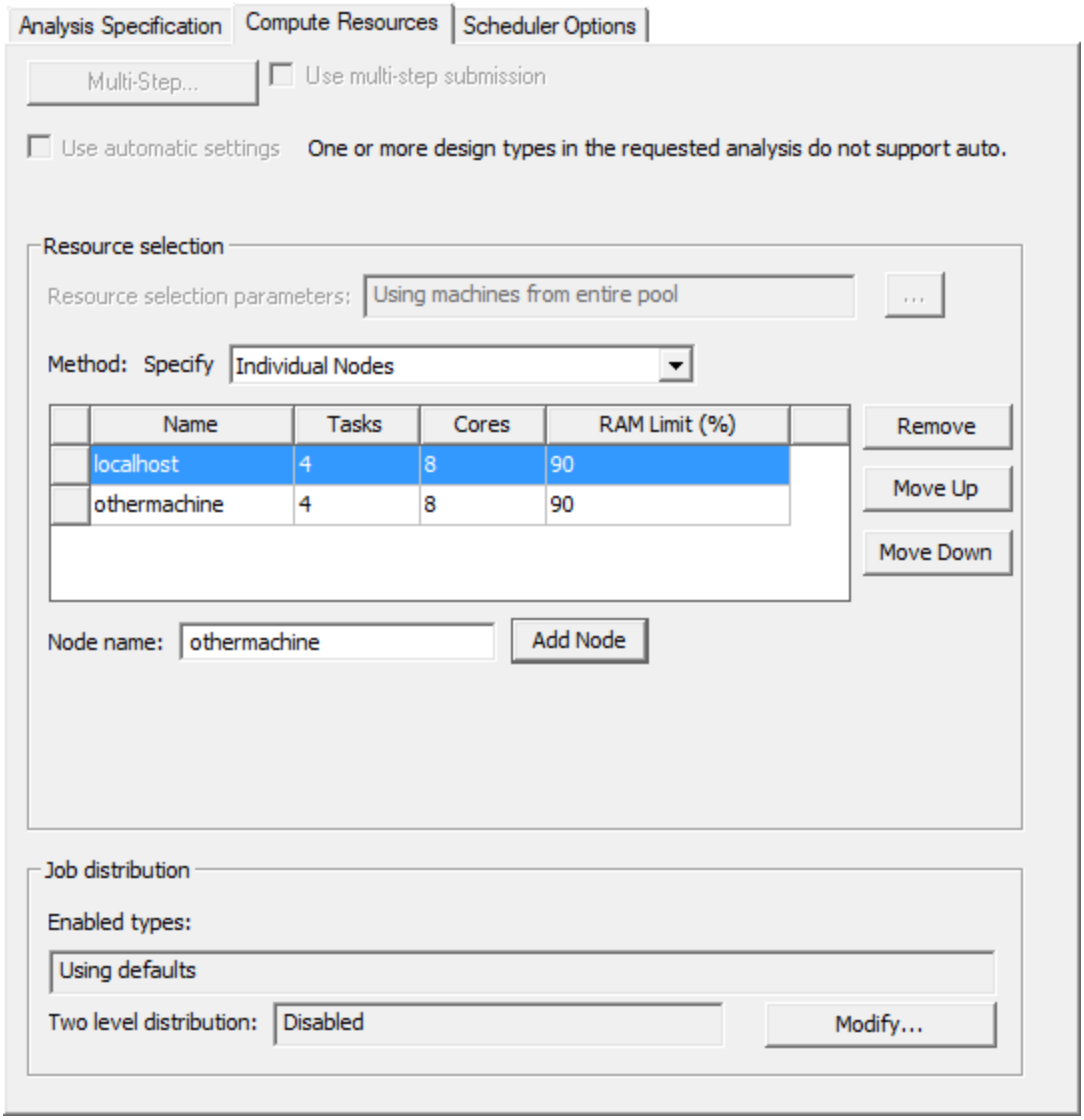
    When available, **Use automatic settings** may be selected. If it is selected, also select the Number of variations to distribute.



    The values you specify represent minimal requirements for each condition that can interact in leading to the total resources the Scheduler derives from them. A submission preview shows the number of resources assigned.

    When automatic settings are not available or not selected, additional options appear:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For RSM Large Scale DSO jobs submitted from the Job Submission panel, localhost must be the first node in the resource selection panel. Otherwise, the job will fail.

In the **Job distribution** area, you can enable or disable **Two-level distribution** by clicking **Modify**. See: Two-level Distribution Guidelines.

12. If desired, click **Preview Submission** to view a summary of the commands to be sent to the scheduler. The text can be copied to the clipboard.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13. To submit the commands to the scheduler, click **Submit Job**.

    > **Note:**
    >
    > The RSM environment does not support queuing, so clicking **Submit Job** starts it immediately.

14. If you enabled job monitoring, you can monitor the job via **Tools> Job Management > Monitor Jobs**. See: Monitoring Jobs.

Large Scale DSO Command Line Syntax

Large Scale DSO operates through a non-graphical batch application called `desktopjob`. You can run the `desktopjob` command line to perform parametric analysis DSO. The command-line interface supported by this batch program is consistent with the command line used for current DSO jobs. `desktopjob -help` lists all available command-line options as shown below:

Command Line Syntax:

```
desktopjob.exe <options> <project-path-on-shared-drive>
```

Note that the project path can be to an archive file.

Options:

- **-help**

  Prints the help text.

- **-cmd**

  Specifies the command to run.

  Available choices: dso

- **-ng**

  Runs the analysis in non-graphical mode.

- **-monitor**

  Outputs progress and messages to standard output/error.

- **-waitforlicense**

  Queues the job until licenses are available.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **-preserve**

  Preserves the local storage space of the distributed job for investigation into the job's run. If local storage directory (for example, the temp directory) is provisioned by scheduler, ensure it is also configured to preserve the job's local storage. *This storage should be deleted manually*.

- **-batchoptions**

  Overrides the Tools/Option entries through either a batchoptions file or batchoptions string.

  -batchoptions specific for Large Scale DSO include:

  To retry failed variations, new batch option LargeScaleDSO/FailedVarRetryCount.

  For each task to re-simulate its failed variations when its assigned variations are finished, specify the FailedVarRetryCount batch option with positive integer. Default 0; Value-Zero(0) will disable the re-simulation for failed variations. Value ranges from 0 or positive integer.

  Redistribution batch option, LargeScaleDSO/VarRedistribution .Value 0 disables redistribution (default). Value 1 enables redistribution.

  Redistribution limit batch option, LargeScaleDSO/RedistributionLimit. Minimum estimated remaining time (in minutes) for variations to redistribute to another task. Value must be positive integer. Default 3 minutes

  Example:

  ```
  -batchoptions <config-file-on-shared-drive>
  -batchoptions "'name1'='val1' 'n2'='v2' "
  ```

- **-jobid**

  Specify a custom job ID for the job. The job's output is organized into a folder with job ID name. This parameter is ignored when run under a scheduler.

- **-machinelist**
  - In the context of **Ansoft RSM**:

    Specify machines for distributed analysis. Machine list is specified either inline (as a comma separated machine names) or through a file. Multiple cores are specified by repeating the name of machine or by embedding number of cores in the machine name, using a colon separator.

    Example 1:

    ```
    -machinelist "list=m1,m1,m1,m2,m2,m3"
    ```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example 2:

```
-machinelist "list=m1:3,m2:2,m3"
```

Example 3:

```
-machinelist "list=m1:1:3,m2:2:2"
```

Example 4:

```
-machinelist "file=machines.txt"
```

- In the context of **a scheduler such as LSF**:

  Specify the portion of total machines for distributed analysis. Use remaining for overhead or shared memory multiprocessing.

  Manual Example:

  ```
  -machinelist "Num=10"
  ```

  Auto Example:

  ```
  -machinelist "NumCores=40"
  ```

  Note: In a scheduler environment, the machinelist argument can be omitted, but the results may not be as expected or desired. If there is no machinelist argument, the scheduler-assigned resources will be used, assuming one core per task. E.g., if the scheduler assigns 40 cores, there will be 40 variations solved in parallel, with only one core used per variation. Since one core per task may not make sense in many cases or may not be what the user desires, a warning message that machinelist is missing will be output, but the job will proceed.

- **-auto**
  - Run the leaf jobs in auto mode. See Submitting Large Scale DSO Job Examples.
- **-numdistributedvariations**:

  Specify the number of parallel leaf jobs to run in auto mode. Required when running under a scheduler. If not specified when not running under a scheduler, then the number of tasks for each machine must be specified in the machine list. The total number of tasks will be the number of parallel variations (leaf jobs) to run.

  Example:

  ```
  -machinelist NumCores=40 -auto -NumDistributedVariations 10
  ```

- **-usefolderasinput**

  Choose this option if the job's input represents an entire folder rather than just the project file.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **-maxfolderInMB**

  Specify the maximum size input folder that is allowed for a valid job (in MBytes). By default, the maximum size allowed for input is 10MB. Specify a value of 0 to remove this size restriction and enable inputs of any size. This option applies when `-usefolderasinput` is used.

- **-workdir**

  Specifies the shared drive folder for status and result files generated by analysis. By default, the results folder of input project is used as the work directory.

- **-meregecsv: [acrossDPs | singleDP | both]**

  across DPs: Merge report csv files for all design-points (variations). One file is created per trace, across all variations.
  singleDP: Merge csv files within a single design-point (variation). One file is created per variation, per a set of traces that can be merged.
  both: Merge all traces that have the same primary sweep for all design-points (variations) into one csv file.
  Interpolation note: If primary sweep values are not uniformly spaced, mergecsv is enabled with traced values and are re-sampled uniformly using '-batchoptions' syntax as shown below:

  ```
  -batchoptions
  "'LargeScaleDSO/NumTracePoints'='PrimarySweepName:200'""
  -batchoptions
  "'LargeScaleDSO/NumTracePoints'='ReportName1:Trace1:100;ReportNam
  e1:Trace1:100;ReportName1:TraceName2:200'"
  ```

- **-abort**

  Abort a running job identified through the job's working directory. Example: `-abort <projectresultsfolder-path>/>jobid>`. For a complete discussion of methods for aborting jobs or specific tasks, see the discussion of Aborting a Large Scale DSO Simulation under Large Scale DSO for Parametric Analysis.

- **-repackageresults**

  Choose this option to add simulations results to the input archive file. Note: this option only applies if an archive file is provided as input.

- **-batchsolve**

  Solves the specified parametric setup.

  Syntax for the setup:

  `<design-name>:Optimetrics:<parametric-setup>`

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Large Scale DSO Job Outputs

A Large Scale DSO analysis does not support the output of full parametric results. Instead, it extracts a subset of results using predefined rectangular plots, which are created by the user before running the job. The extracted columns of data are saved as CSV files. Typically, there is one CSV file per-trace, per-variation. The outputs can be either imported as datasets for post-processing in the desktop also as function of parametric variations, or used directly in downstream applications (for example, Excel, or custom programs that parse .csv files).

> **Note:**
>
> Non-Rectangular plots of the design (for example, statistical eye and digital plot) are not extracted.

Output Location and Organization

The results of a Large Scale DSO job are located in the `<workdir>/<jobid>/results` folder. If `workdir` is not specified on the job command line, it is same as the input project's results folder. For example, the default `workdir` corresponding to `\\shared\projects\tee.aedt` is `\\shared\projects\tee.aedtresults`. Within this results folder, there is one folder per variation. The name of the variation's folder is an integer number corresponding to variation's index in the parametric table. For example, a variation folder named '4' has results for the fifth row of parametric table, while a variation-folder named '0' has results for the first row of the table. Each variation's folder contains a CSV file for each trace.

CSV File Contents

The initial header rows of the CSV file define the solved variation. For each such row, the first column contains a variable name and the second contains a variable value. The row following variation rows has the name of primary sweep and the name(s) of extracted quantities. Subsequent rows contain data—quantity values as a function of primary sweep.

These examples provide context:

- **Traces of an S-parameter Report** – The data portion of the CSV file contains two columns of data: the first contains Freq values and the second contains values for the trace's s-parameter component.
- **Trace of a Far Field Report** – Suppose there is a far field report with a trace (magrE), whose primary sweep is phi and whose secondary sweep is theta. Further suppose that two values of theta are chosen and all values of phi are chosen. For this trace, the data portion of CSV file contains three columns of data: the first containing phi values, the second containing magrE values for the first value of theta, and the third column containing values for second value of theta. The magrE output columns are titled as 'magrE_crv1' and 'magrE_crv2' respectively.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Advanced Sweeps** – In the case of a trace with special primary sweep (such as the trace of a time domain quantity), one CSV file is created per curve of trace, per variation. These CSV files always have two columns, irrespective of the number of values chosen for secondary/higher sweeps.

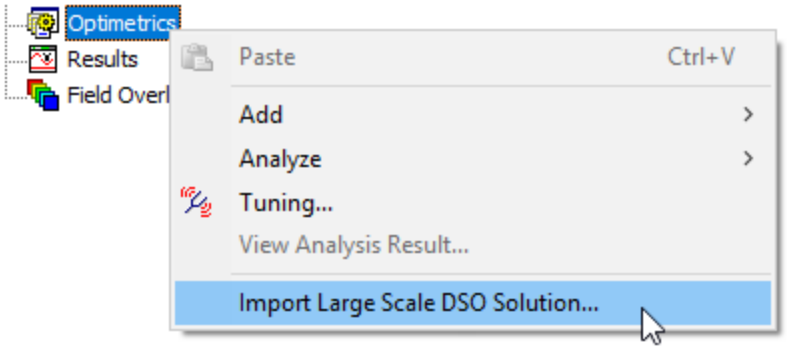
Postprocessing Large Scale DSO Dataset Solutions

In Ansys Electronics Desktop, the **Import Large Scale DSO Solution** command allows you to post-process Large Scale DSO dataset solutions.

Importing a Large Scale DSO Solution

To import a Large Scale DSO solution:

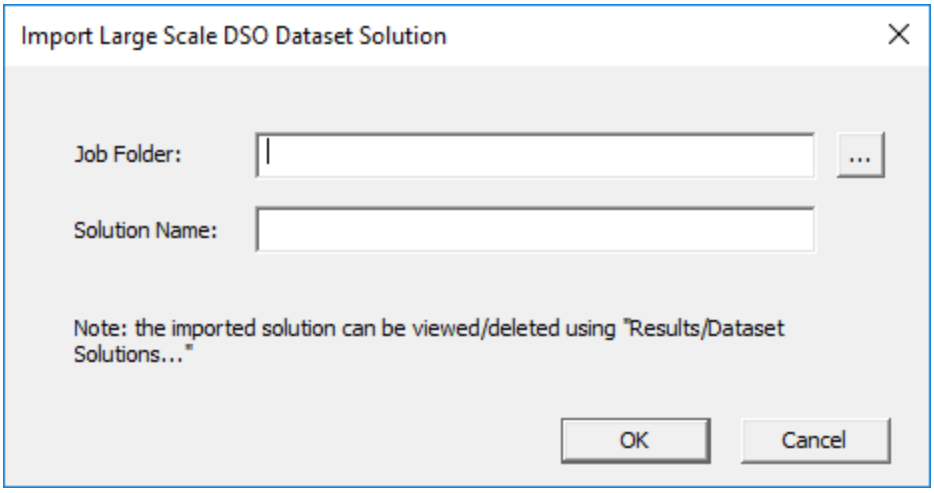
1. In the Project Manager, right-click **Optimetrics** and select **Import Large Scale DSO Solution**.



> **Note:**
>
> You can also access this window by right-clicking **Results** > **Dataset Solutions**.

The **Import Large Scale DSO Dataset Solution** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click the ellipsis button (**...**) and browse to select a job folder. To select a results dataset, double-click the results folder name. Results folders are organized by the scheduler prefix (for example, RSM) and job number.
3. Click **Open** to return to the **Import Large Scale DSO Solution** window.
4. Review the **Job Folder** path and **Solution Name**, then click **OK**.

   The dataset is imported.

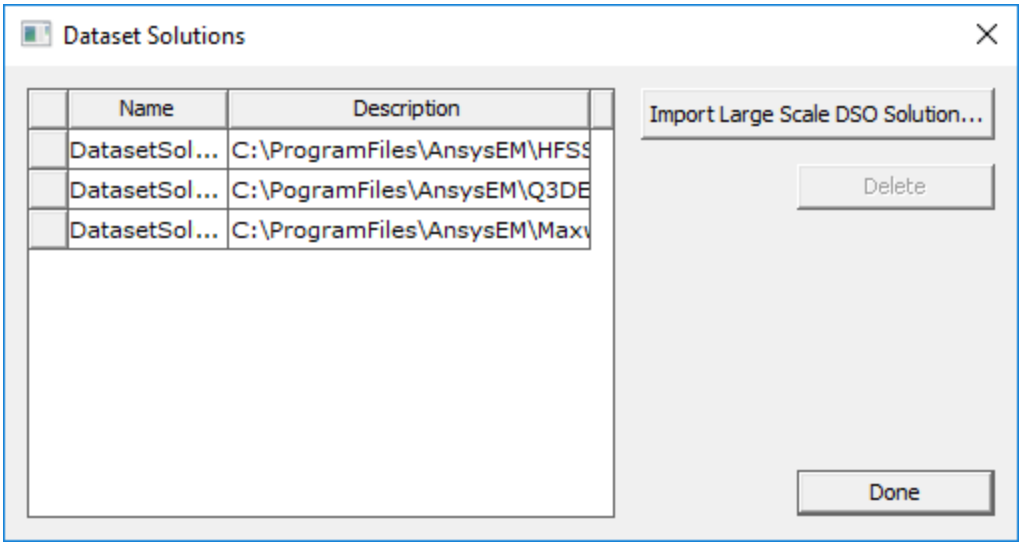
Viewing Imported Solutions

To view a list of imported datasets:

1. In the Project Tree, right-click **Results** and select **Dataset Solutions...**

   The **Dataset Solutions** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Selecting a dataset enables the **Delete** button, which you can click to remove it.

2. Click **Done** to close the window.

Creating a Dataset Report

After importing one or more DSO solutions, you can create a dataset report.

To do so:

1. In the Project Tree, right-click **Results**. Select **Create Dataset Report** > **[Report Type]**.

   The **Report** window for the report type opens. See Creating Reports.

Cloning a Dataset Solution

If you reopen a project that was solved using Large Scale DSO, you can quickly clone the solution report:

1. In the Project Tree, under **Results**, right-click the report and select **Clone from Dataset Solution** > **[Solution Name]**.

   The cloned solution appears. You can now reuse the existing report definition rather than creating a new report.

Large Scale DSO Results Database Organization

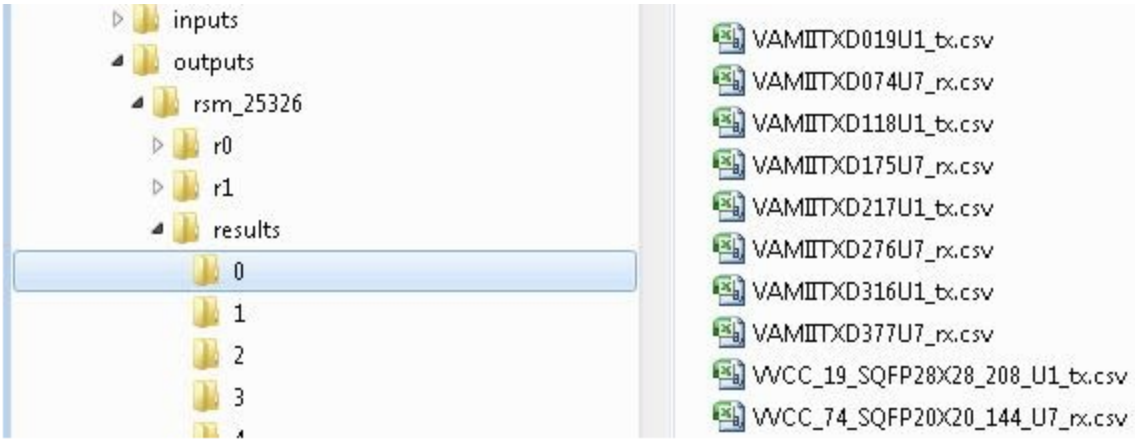
A large scale DSO analysis does not support the output of full parametric results. Instead, it extracts 'subset' results using predefined Rectangular plots, which are created by the user before the job is run. The extracted columns of data are saved as CSV files. Typically, there is

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

one CSV file per-trace, per-variation. (Note: Non-Rectangular plots of the design (for e.g. statistical eye, digital plot) are not extracted)

The results of a large scale DSO job are located in the *<workdir>/<jobid>/results* folder. If 'workdir' is not specified on the job command-line, it is same as the input project's results folder. For example the default 'workdir' corresponding to *\\shared\projects\relay.aedt* is *\\shared\projects\relay.aedtresults*. Within this results folder, there is one folder per variation. The name of the variation's folder is an integer number corresponding to variation's index in the parametric table. For example, a variation-folder named '4' has results for the fifth row of parametric table, while a variation-folder named '0' has results for the first row of the table.

Below is a sample results folder showing the contents corresponding to results of first variation. There are ten CSV files corresponding to ten predefined traces:



Large Scale DSO Job Monitoring

The **Monitor Job window** allows you to monitor the progress and status of Large Scale DSO jobs, including information on variations solved so far, variations currently solving, and the number of variations remaining.

Additional Resources for Large Scale DSO Monitoring

Large Scale DSO avoids detailed intra-variation monitoring, as it increases network traffic for large-scale jobs. Additional monitoring resources include:

- **Cluster Monitoring Tools** – Standard cluster monitoring tools are ideal for job-neutral resource monitoring as they use negligible network bandwidth.
- **Detailed Monitoring of Analysis of a Variation** – For detailed monitoring, you may want to examine a job's log files. Large Scale DSO writes detailed logs about the machines where engines are running and the local storage location of per-engine distributed databases. You can log in to individual machines for deeper probing of each distributed engine.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following logs are available:

- **Per-Node Logs** – There is one desktopjob.log file per node assigned to the job. This log contains information regarding the node such as name, local storage folder, and number of engines started on this node. It is located in <workdir>/<jobid>/r<nodeIndex>. For example, <workdir>/<jobid>/r0 contains the desktopjob.log corresponding to the engines running on the first node of job, while <workdir>/<jobid>/r2 contains the log corresponding to engines running on the third node.
- **Per-Engine Logs** – There is one desktopjob.log file per distributed engine. It is located in <workdir>/<jobid>/r<nodeIndex>/r<taskIndex>. For example, <workdir>/<jobid>/r0/r0 contains the log corresponding to first engine running on first node, while <workdir>/<jobid>/r1/r2 contains the log corresponding to the third engine running on the second node. Engine unique information (such as local storage of this engine) is logged here.
- **Parametric Analysis Log** – This log file is located in'<workdir>/<jobid>/r<nodeIndex>/r<taskIndex> and corresponds to Desktop's local-machine parametric batchsolve. It is available only at the end of analysis and contains information regarding the variations solved by this engine and any info/warning/error messages.
- **Root Log** – This is the top-level desktopjob.log file that logs job distribution information such as hierarchical activation and the list of nodes assigned to this job.

For a complete discussion of methods for aborting jobs or specific tasks, see the discussion of Aborting a Large Scale DSO Simulation under Large Scale DSO for Parametric Analysis.

Large Scale DSO Deployment/Configuration

LINUX Cluster Configuration

- 'Temp directory' configuration.

  Temp directory is either on 'local storage' or on storage that has equivalent speed characteristics. The I/O rates of the storage should be invariant to network traffic

  Temp directory on a host has sufficient space to hold results database for the variations that are solved on it. (Note: This storage is freed at the end of the analysis.)

  The amount of required space depends on the number of engines per node and the cumulative variations solved on this node

  The amount of required space depends on the project's compression-options. For example, if 'Save Fields' of a parametric setup is OFF, the space requirement is smaller by the amount of space taken up by field solution data.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Ansoft RSM environment: In the case of supported scheduler environments, there is no extra configuration needed. In the case of Ansoft RSM environment, following additional steps are needed:

  Ansoft RSM must be running on all the nodes of cluster. The credentials of 'RSM service' allow read/write to shared drive. Reason: the remote engine processes are launched using the credentials of RSM service

  Registration of 'desktopjob.exe' with RSM service: 'desktopjob' program must be registered with Ansoft RSM using 'desktopjob -regserver". To ensure that the registration is successful, check that the 'desktopjob' entry in '<RSM-installation-folder>/AnsoftRSMService.cfg' file is valid.

> **Note:**
>
> LINUX specific critical note: Edit AnsoftRSMService.cfg and replace 'desktopjob.bin' with 'desktopjob'

Major limitation: In the Ansoft RSM environment, Large Scale DSO can only be enabled for one product.

Troubleshooting hints (Ansoft RSM environment only): "shared drive read/write" requirement is a new constraint introduced in Large Scale DSO. So if user runs into a situation where Regular DSO jobs run and Large Scale DSO jobs fail, one possible cause for the failure: RSM service does not have privileges to read and write to project folder located on shared-drive.

Windows Cluster Configuration

All the above steps apply, except for steps that are stated as LINUX-specific. Additional instructions:

- Ansoft RSM and Ansys Electromagnetics products are either installed locally on each node of cluster OR installed on a single shared-drive available to all nodes of cluster.
- Registration of 'desktopjob.exe' with RSM service.
- Network installation: desktopjob.exe is registered with RSM service once, on any of the nodes of cluster
- Local installation: Since each node has it's own RSM installation, desktopjob.exe must be registered with RSM on each node.

> **Important:**
>
> Ansoft RSM service must be started using the credentials of a non-system 'admin' account, which has read/write permissions to project's shared drive. If RSM service runs as 'system' user, large-scale-dso jobs will fail

Heterogeneous Cluster Configuration

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Limitation: Currently heterogeneous cluster (with both linux and windows nodes) is not supported. This is due to the shared drive requirement.

Large Scale DSO Tutorial Example

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

This section provides an example of the use of Large Scale DSO to distribute parametric variations of an HFSS model across the nodes of a cluster or to multiple cores of a single machine.

This example presumes that your configuration satisfies the Prerequisites for Large Scale DSO. For this example, pre-suppose that we have a Windows cluster. Further suppose that the shared drive folder, which contains the input projects and computed results is at the location *\\sjo7na1\hfssprojs*.

Major steps for Large Scale DSO Example

1. Prepare the model for Large Scale DSO Analysis
2. Submit the Large Scale DSO Job
3. Post process the results

Prepare the Model for Large Scale DSO Analysis

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

1. Launch the desktop.
2. Create the input project on the shared drive.

   For this example, start with the standard HFSS OptimTee.aedt example and copy it to the shared drive.

   Copy "<installation-directory>\<platform>Examples\RF_Microwave\OptimTee.aedt" to "\\sjo7na1\hfssprojs\OptimTee.aedt"

3. Set up the parametric table.

   For this example, use the existing 'ParametricSetup1' as the parametric setup to solve.

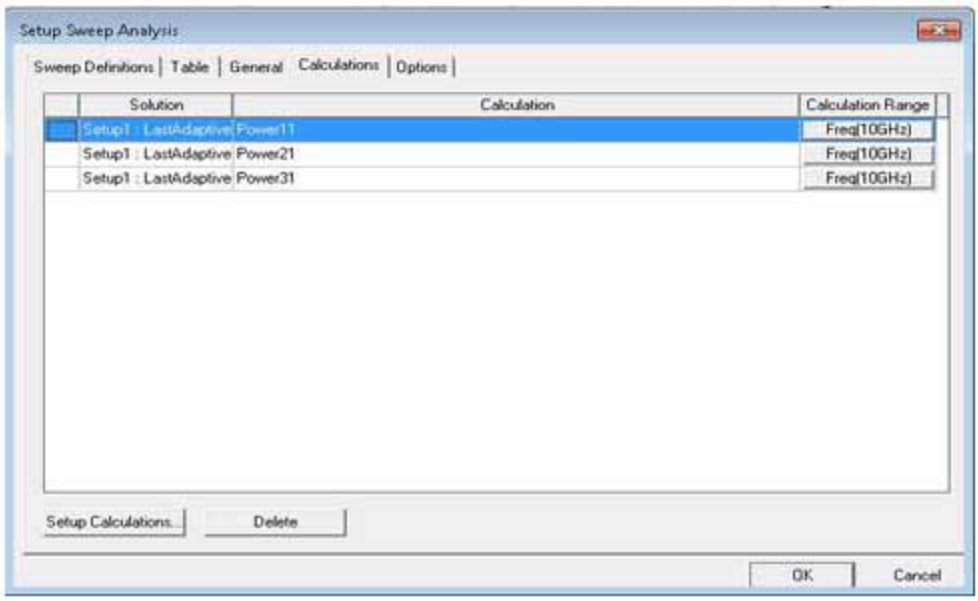
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Follow the steps below to prepare the outputs.

   Outputs from Large Scale DSO come from predefined rectangular plots that are created *before* the Analysis command is issued. Non-rectangular plots of the design (digital plots, for example) are not extracted. The result data of the rectangular plots should be of datatype "double". Other result types (e.g. complex numbers) are not supported.

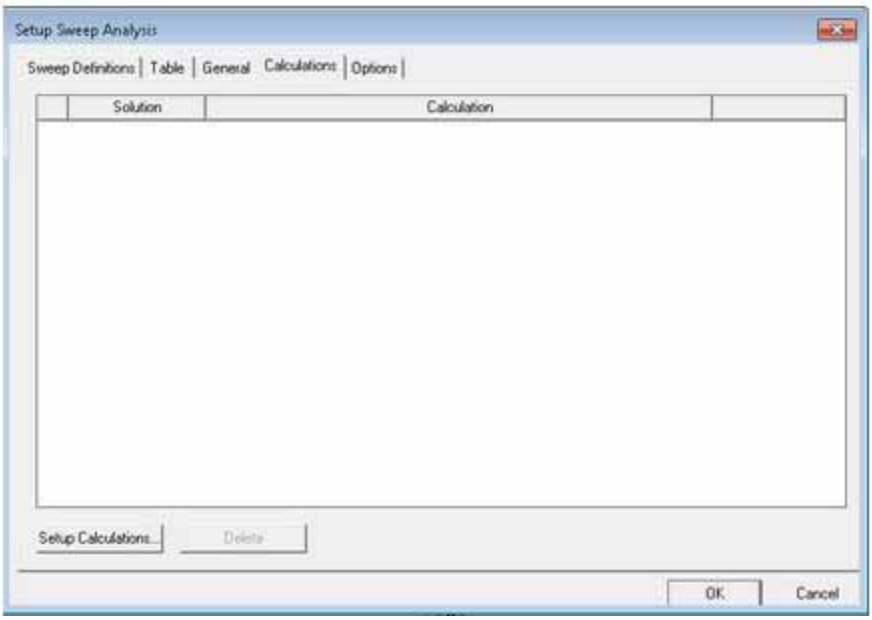
   a. Because these DSO outputs come solely from Rectangular Plots, delete all other postprocessing setups, and then turn off Save Fields And Mesh as shown below.

      Open ParametricSetup1 for editing. You will see that the OptimTee parametric setup contains three calculations, as shown below.



   b. Delete all three calculations. When you have done so, the Setup Sweep Analysis Calculations tab looks like this:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



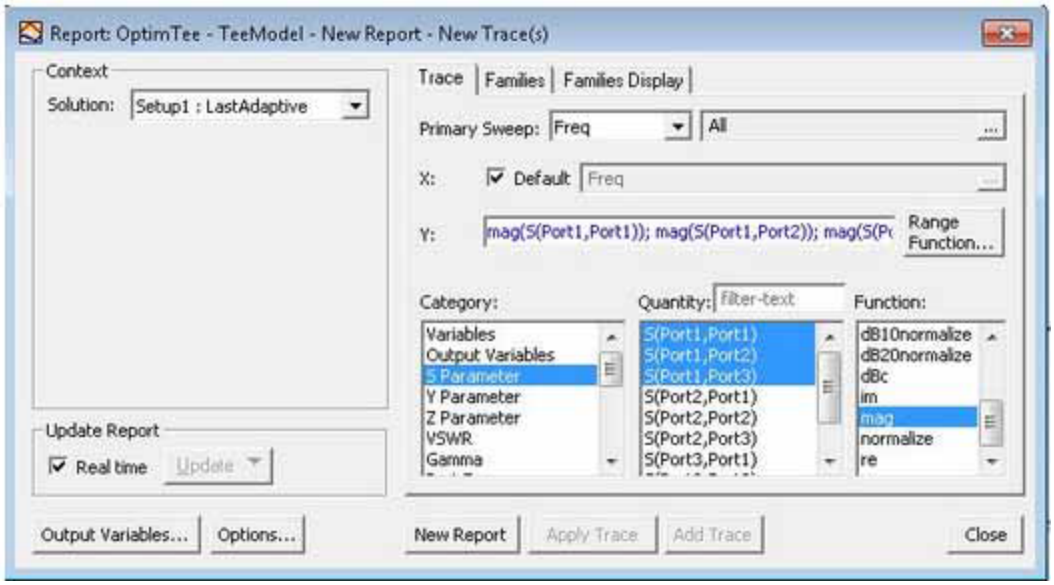
c. Click on the **Options** tab and uncheck Save Fields And Mesh, as shown below:

5. Use the Reporter to define outputs.

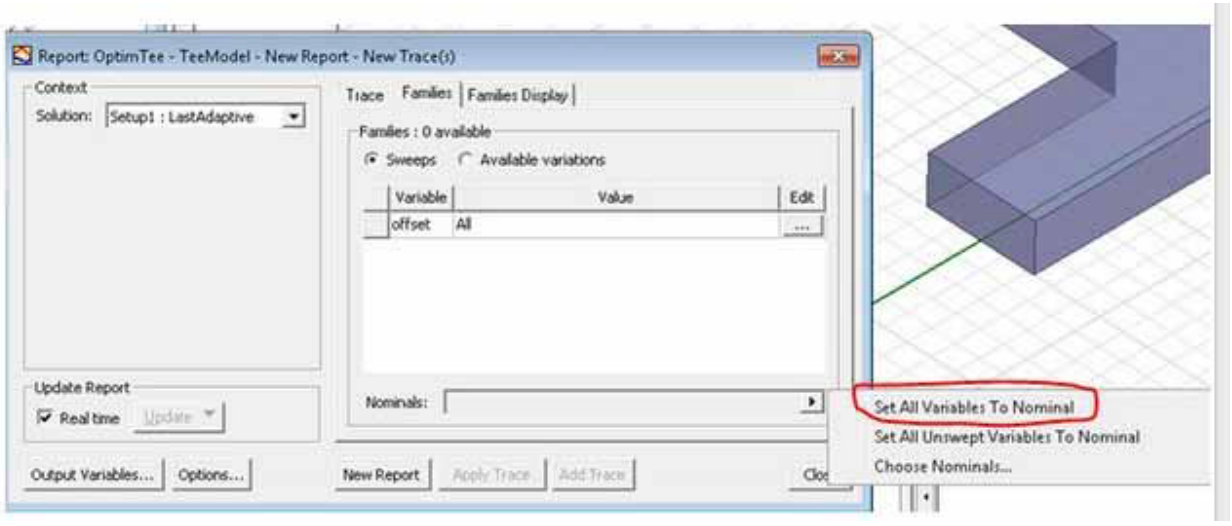
   For this example, you add six traces that correspond to six csv outputs of the Large Scale DSO job: mag(S11), mag(S12), mag(S13), Power11, Power12, and Power13.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. As shown below, select three quantities: mag(S11), mag(S12), and mag(S13).



b. Click on the **Families** tab and ensure that all variables are set to Nominal, as shown below.



Note:

For Large Scale DSO, outputs are not extracted correctly unless all variables on the **Families** tab are set to nominal.

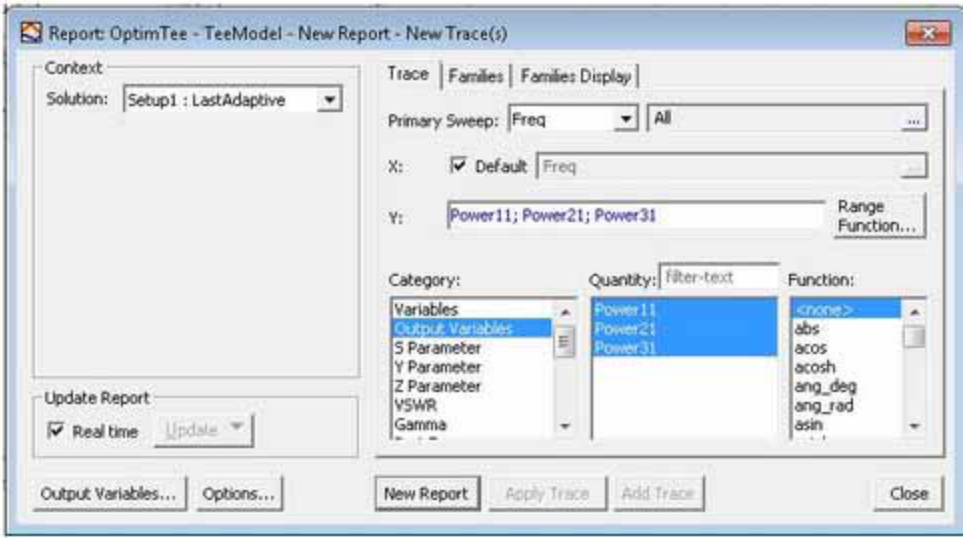
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Click the **New Report** button.

   Use the Report to create a Power distribution plot that has three power distribution traces.

d. To create an S-parameter plot that has three traces, click **Results** > **Create Modal Solution Data Report** > **Rectangular Plot**.

e. As shown below, select three quantities: Power11, Power21, and Power31.

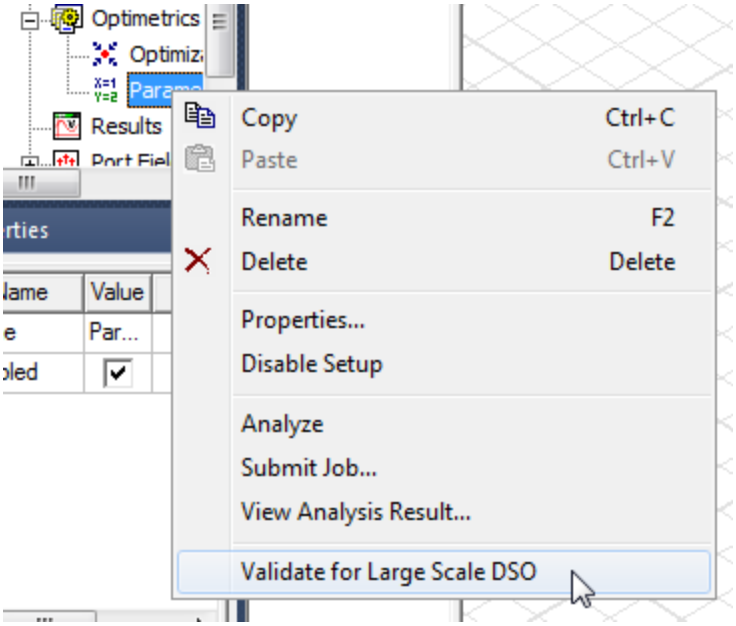


f. Click on the **Families** tab and ensure that all variables are set to **Nominal**.

g. Click the **New Report** button.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Right-click the parametric sweep, and select **Validate for Large Scale DSO**.



7. A dialog reports any errors. These may occur if the steps just taken are neglected:

   ```
   [error] Please remove all the calculations in Parametric Setup/Calculation page.
   ```

   ```
   [error] Please turn off "Save Fields And Mesh" in Parametric Setup/Options page.
   ```

   ```
   [error] No rectangular plot exist in the design. Please create a rectangular report.
   ```

8. Correct any errors, if necessary to pass validation.
9. Save and Close the project.

Next

Submit the Large Scale DSO Job: Examples

Submit the Large Scale DSO Job: Examples

This section includes examples of submitting a Large Scale DSO job using Ansoft RSM and a Scheduler.

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

Using the Ansoft RSM Environment

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Before submitting the job make sure that the input project is not open in any Ansys Electronics Desktop window.
2. For this example, suppose that there are two quad-core machines on your cluster with the names "m1" and "m2". Further suppose that there are two engines per machine, for a total of four parallel engines. Let the number of processors allocated to each engine be 1.
3. From a command prompt, issue the following command:

   ```
   "<installation-directory>\ANSYS
   Inc\v252\AnsysEM\desktopjob.exe" -cmd dso -machinelist
   "list=m1:2,m2:2" -batchoptions
   \\sjo7na\aedtprojs\hfssoptions.txt -batchsolve
   "TeeModel:Optimetrics:ParametricSetup1"
   \\sjo7na\aedtprojs\OptimTee.aedt
   ```

   where the file \\sjo7na\hfssproj\hfssoptions.txt has the following contents:

   ```
   $begin Config
     'Desktop/Settings/ProjectOptions/NumberOfProcessors'=1
     'HFSS/Preferences/SaveBeforeSolving'=0
     'HFSS/Preferences/HPCLicenceType'='pack'
   #end 'Config'
   ```

4. Suppose the above job is assigned ID "jobID".

Using a Scheduler Environment (such as LSF)

1. Suppose you want to solve variations using four parallel engines, each engine being assigned a single core.
2. From a command prompt, run the following command:

   ```
   bsub -n 4 "<installation-directory>\ANSYS
   Inc\v252\AnsysEM\desktopjob.exe" -cmd dso -batchoptions
   \\sjo7na\hfssprojs\hfssoptions.txt -batchsolve
   "TeeModel:Optimetrics:ParametricSetup1"\\sjo7na\hfssprojs\Opti
   mTee.aedt
   ```

   where the file \\sjo7na\hfssproj\hfssoptions.txt has the same contents as the RSM example above.

3. Suppose the above job is assigned an ID "jobid"

For example with 2 nodes of 44 cores each (with multiple distributed variations per machine):

MANUAL: `bsub -n 88 <installation-directory>\v<version>\<platform>\desktopjob -cmd dso -machinelist num=8 -monitor -useelectronicsppe=1 -ng -batchoptions " 'Maxwell 3D/NumCoresPerDistributedTask'=11 'Maxwell 3D/RAMLimitPercent'=90`

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
'HPCLicenseType'='Pack'" -batchsolve
TeeModel:Optimetrics:ParametricSetup1 OptimTee-v232-1.aedt
```

AUTO: `bsub -n 88 <installation-directory>\v<version>\<platform>\desktopjob -cmd dso -machinelist numcores=88 -auto -NumDistributedVariations 8 -monitor -useelectronicsppe=1 -ng -batchoptions " 'Maxwell 3D/RAMLimitPercent'=90 'HPCLicenseType'='Pack'" -batchsolve TeeModel:Optimetrics:ParametricSetup1 OptimTee-v232-1.aedt`

For example, with 16 nodes of 44 cores each (note each distributed variation will span 2 machines):

AUTO: `bsub -n 704 <installation-directory>\v<version>\<platform>\desktopjob -cmd dso -machinelist numcores=704 -auto -NumDistributedVariations 8 -monitor -useelectronicsppe=1 -ng -batchoptions " 'Maxwell 3D/RAMLimitPercent'=90 'HPCLicenseType'='Pack'" -batchsolve TeeModel:Optimetrics:ParametricSetup1 OptimTee-v232-1.aedt`

Next

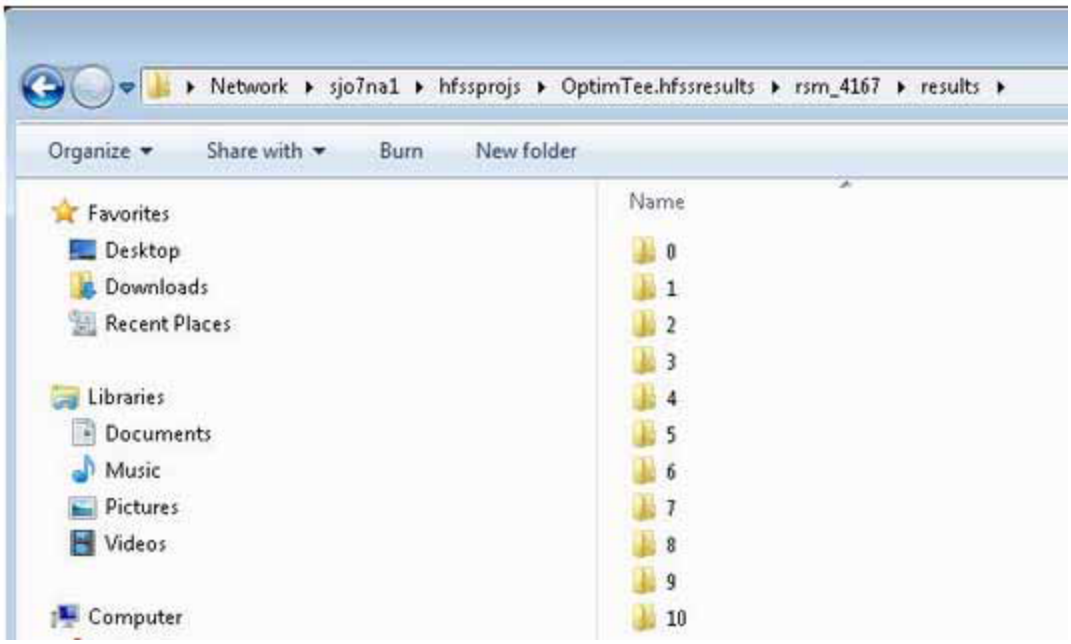
Large Scale DSO Example: Post Process the Results

Large Scale DSO Example: Post Process the Results

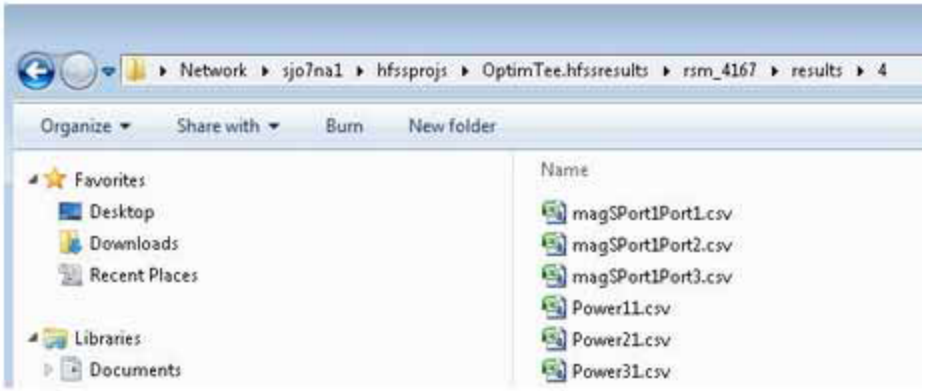
Once the job is done, output is available in the ~\OptimTee.aedt\jobid\results folder. Each variation creates a subfolder, which in turn has one csv file per trace of each report. See the detailed information regarding job monitoring and the location of the analysis logs.

The following figure shows the results for 10 variations as located in 10 folders.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



There are six csv files corresponding to three S-parameter traces and three power distribution traces, as shown below for the fourth variation.

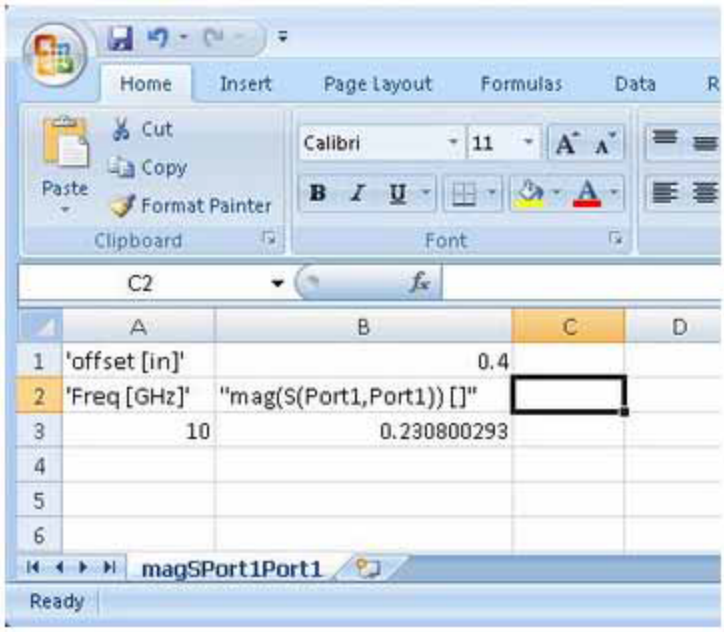


You have three options for postprocessing csv files.

- Import Large Scale DSO Dataset Solution
- Use Microsoft Excel or any other application that has csv post processing functionality.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

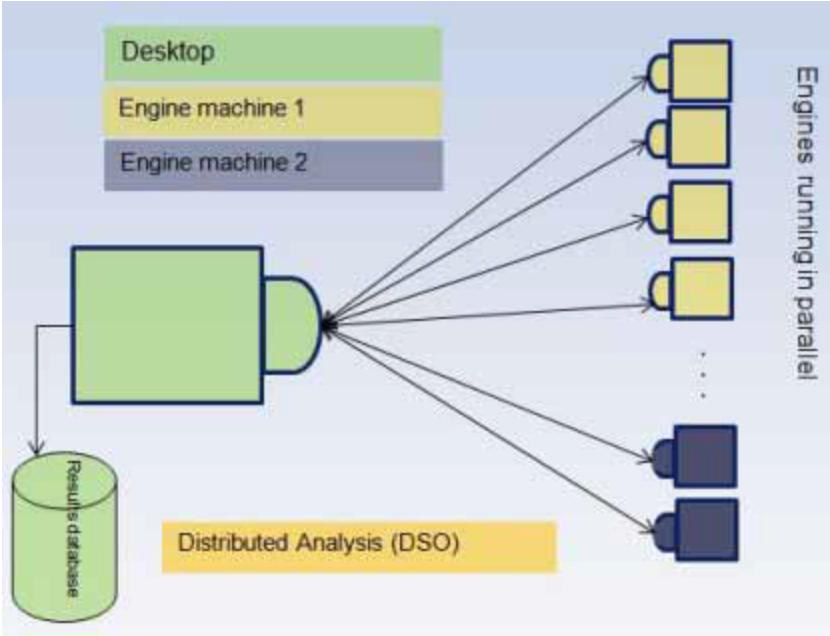
- Parse the csv output into your custom program, for any downstream flow.



Large Scale DSO Theory

The parametric analysis command in Desktop computes simulation results as a function of model parameters, such as geometry dimensions, material properties and excitations. The parametric analysis is either performed on a local machine, where each variation is analyzed serially by a single engine, or distributed across machines through a DSO license. Desktop's DSO analysis runs multiple engines in parallel, thus generating results in a shorter time. In the Regular DSO algorithm, the parametric analysis job (Desktop) runs on master node, which in turn launches one or more distributed-parallel engines on each machine allocated to the job. Desktop distributes parametric variations among these engines running in parallel. As variations are solved, the progress/messages and variation results are sent back to Desktop, where they are persisted into the common results database on master node, as illustrated below:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A single engine can now span multiple machines with auto multi-level DSO.

Regular DSO Bottleneck

As per above illustration, Regular DSO's speedup is limited by the resources of the centralized 'Desktop' bottleneck. It's been observed that DSO becomes unreliable at a certain point, as the number of engines and number of variations is increased. The term 'large scale parallel' can be used to define this tipping point. For a given model, a 'large-scale parallel' job denotes scenarios, in terms of the number of distributed-parallel engines and the number of parametric variations, where the Regular DSO runs into centralized bottlenecks that result in one or both of: progressively smaller speedups, unreliability.

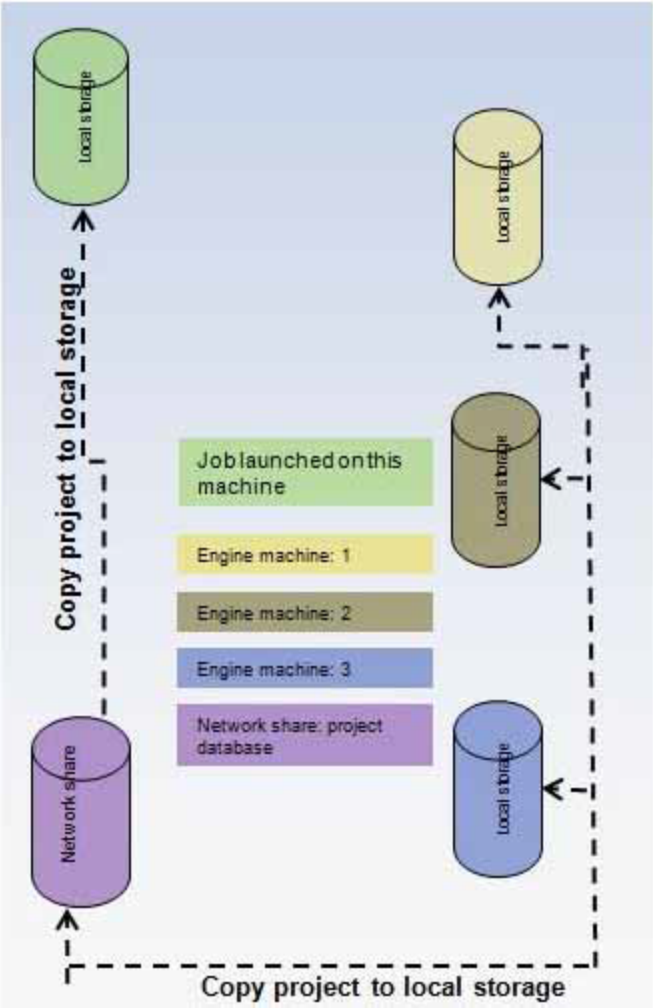
With the advent of economical availability and timely provisioning of compute resources, product designers have access to large compute clusters to run their simulations. And they are throwing larger and larger number of compute resources at simulation jobs, in order to obtain results faster. The parametric DSO needs to meet this challenge and target linear speedup for 'large-scale parallel' jobs. The Large Scale DSO feature is targeted toward 100% reliability and linear speedup of large-scale parallel DSO jobs.

Key Algorithms/Concepts for Large Scale DSO

- **Embarrassingly Parallel algorithm**: Large Scale DSO exploits the embarrassingly parallel nature of parametric runs. The solve of each variation is made fully independent of another variation, by replacing Regular DSO's centralized database with per-engine distributed databases
- **Distributed databases**: For each distributed-parallel engine, the database of the input model is cloned to local storage of engine's compute node (as illustrated in the following

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

picture). Parallel analysis is performed on these cloned models and the analysis results also go the corresponding location on local storage. As each engine has it's own database, there is no 'shared database' contention between any two engines. There is negligible network traffic as analysis/computations are contained in a single machine and use just the 'local resources'.



- **Hierarchical activation of engines**: In the Large Scale DSO algorithm, engine activation is done hierarchically, where the overhead of launching of engines is also shared across the nodes. Such hierarchical activation improves reliability of the activation phase of large-scale parallel jobs. A new 'desktopjob' program encapsulates hierarchical activation, as illustrated in the following pictures. In this approach, the 'root' desktopjob (Root DJ) activates a 'level-one' child desktopjob (DJ(L1)) for each unique node allocated to the DSO job. Each level-one desktopjob activates one or more 'leaf' desktopjobs (DJ(L2)) equal to the number of distributed engines per node. A leaf desktopjob in turn runs Desktop in batch mode, to perform local-machine parametric analysis to solve the variations assigned to this engine.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Decentralized Load balancing:** A parametric table is divided into regions of equal number of variations, with the number of regions equal to the number of engines. In the following illustration, the analysis of 30 parametric variations is distributed among 5 engines. Each engine solves it's assigned region as a 'local machine' parametric analysis. Large Scale DSO job is considered done once all engines are done with their assigned variations.
- **Distributed results postprocessing**: As engine is done with the solve of a variation, it extracts results for the solved variation before progressing to the analysis of next variation. The extracted results are saved to the local storage. When the engine is done with analysis of all variations, the extracted results are transferred from local storage to the results folder of the input project.

Redistribution Feature

Large Scale Distributed Solve Operation could submit a parametric setup to be solved in multiple machines, each machine may launch multiple EM-Desktop processes to solve the assigned variations(Design Points). Variations are distributed to each task(EM-Desktop process) equally, regardless of the machine hardware and each variation's complexity. That may resultin some tasks finishing earlier than others, in some extreme case some tasks may hours behind fastest task.

Now, redistribution occurs when a task finished its own assignment, it calls back to the L2 to ask for new assignment, L2 forward the request to L1, L1 forward it to L0, L0 may pick one of the slow task to remove some variations or return with no more assignment. When a task is picked to remove unsolved variation, L0 calls L1(may be different L1), L1 forward it to the selected L2, L2 makes a request to the EM-Desktop to remove some variations from its queue. EM-Desktop returns the removed variation indexes, or error code if it fails. In L0 if error is returned, it marks the selected task failed to respond the remove request, to void picking it again. L0 returns the result through L1-L2 back the EM-Desktop.

The Ansys Electronics Desktop will enqueue the new assignment and then start solving those new variations. If the return data indicates error, it makes another call to L2 to get new assignment. If no more variations return, the Ansys Electronics Desktop finishs the simulation and exit.

Large Scale DSO Known Issues/Troubleshooting

Node Order

For Large Scale DSO jobs that are submitted from job submission panel using RSM, localhost must be the first node in the resource selection node list, otherwise Large Scale DSO solve with RSM will fail. Set the order in the **Submit Job To** window, **Compute Resources** tab:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Name	Tasks
	localhost	1
	othermachine	1

Parallel Task Limitation for LS-DSO Parametric Variations

There some limitations of running short parametric runs in parallel and using a subset of cores and/or running across multiple machines instead of just one. LS-DSO does help in getting close to linear scalability for parametric runs in many scenarios, it has some overheads and typically they are small in comparison to total time of run. However, a few factors in particular scenarios can make them prominent.

The startup of each task involves copying the project and launching ansysedt to solve a subset of parametric table. Typically, one task solves several variations and this startup cost is negligible in comparison to total time. Consider a case where each task is solving only single variation. Additionally the actual solve time of one variation is relatively small, 3-5 minutes. These factors make the startup cost significant. There is also some variance in solve times of different variations. When all variations are solved in parallel, the total time is determined by slowest variation, even though the expectation intuitively might be relative to average time. That could make the perception of overhead worse in this case.

Another important factor is the number of parallel tasks on each machine relative to the total number of cores on the machine. Each task is effectively running an ansysedt of its own and solving subset of parametric table on a copy of project. The solve process does involve significant disk IO. If you run too many tasks on a single machine, they may end up competing with each other for single disk, and that could cause them to slowdown. The conflict in memory access may also become a factor.

In a particular case, 26 tasks were run on one 26 core machine, and this caused significant slowdowns. Spreading the tasks across machines helped scalability as an example with a 32 core machines in a Linux Cloud, we ran this job with 26 tasks on a single machine and then another one over two machines with 13 tasks per machine. The single machine job finishes in about 14 mins, while the job on two machine finishes in about 8.5 mins. This indicates that spreading tasks across machines helps with scalability by minimize file read/write conflicts.

Job Restart

There is no provision for stopping and restarting a job. A new job does not reuse solved results; it always solves all rows in the table. An abort or failure of a job restarts from the beginning, unless a new parametric table with the unsolved rows is created.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Linux-Only Issues

- Deployment/Installation errors (such as mainsoft-related) are not captured. If there is such an issue, the Large Scale DSO job will fail without useful messages in the logs.
- Report-based extraction fails if traces and parametric-setup are not prepared as per the Getting Started guides.
- Job status: The exit code of job doesn't indicate success or failure correctly. The error messages from multiple log files needs to be combined to determine the reason for failure. In many situations, the reason for a failure is apparent only after re-running the job after turning ON the 'debug logging'.
- In some LINUX scenarios, the analysis appears to finish successfully with valid results, except that the exit code is '134'. In this case, although the exit is abnormal, the failed exit code can be ignored.
- Load Balancing: For models with 'unbalanced variations table' (variations that take considerably different amount of time to solve are clustered in few regions of table), job will take longer time to solve than a Regular DSO as the job's overall completion time is determined by the slowest solving region. Workaround: rearrange the rows in the parametric table so that each region takes a similar time to solve.
- GM Specifics: the model used for 'Report-based extractor' jobs is NOT compatible with the 'Ansys-extractor-for-GM' jobs. A valid model for Ansys-extractor-for-GM cannot contain any of: reports, overlay plots, Optimetrics calculations.

Interactive Scheduler Jobs

This document includes information, guidelines, and caveats for users running interactive scheduler jobs on Linux.

In most cases, jobs run under a scheduler run as a batch job. These jobs may be submitted using the Ansys Electromagnetics Desktop job submission GUI, using cluster job submission commands on a command line or using a cluster GUI, if available.

Some customers use an alternative method for submission of scheduler jobs. For convenience, we call such jobs "interactive scheduler jobs." In this approach, the user submits an interactive job to the scheduler. From the interactive job prompt, the user launches an Ansys Electromagnetics Desktop product, which starts in interactive (GUI) mode, not batch mode. The user selects a project and then runs one or more analysis commands using the GUI. The intent is that these analysis commands should use all resources allocated to the job, whether on the same host as the GUI or on other hosts.

This approach is supported on Linux, where the user may set up an X Window System server for interacting with the Ansys Electromagnetics Desktop product GUI. The user needs to configure

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the cluster environment and/or the interactive environment so that the user may view and interact with the product GUI. This approach is not supported on Microsoft Windows.

Specifying Options for Interactive Scheduler Jobs

DSO Configuration for Interactive Scheduler Jobs

Design Type Options for Interactive Scheduler Jobs

Specifying Options for Interactive Scheduler Jobs

Batch scheduler jobs use command line options for specifying options rather than GUI controls. For interactive scheduler jobs, options may be specified on the command line, using GUI controls, or obtained from the registry. The machines specified in the **Configurations** tab of the **HPC and Analysis Options** window will be ignored for interactive scheduler jobs. If the command line used to launch the product contains a list of specific machines (using option -machinelist list=...), then the job will use the specified machines and cores. If the command line used to launch the product does not contain a specific machine list, then the machines and cores allocated to the job by the scheduler will be used for the job. In general, the job distribution settings for interactive scheduler jobs should be specified on the command line, instead of using the **HPC and Analysis Options** window.

- The UI shows the settings that will be used for analysis, even if they come from the command line, not the registry.
- If you make changes to the settings in the UI, and the changed settings will be used for the analysis, even if the changed settings conflict with the command line (including batchoptions).
- If you makes no changes to the settings in the UI, then the command line settings (including batchoptions) will be used for analysis.

Batchoptions settings that are automatically generated for batch jobs submitted using the AnsysEM job submission GUI will need to be manually included in the product command line. Some use cases require that certain settings be made on the command line when the product is launched, rather than using the GUI.

Batchoptions for Interactive Scheduler Jobs

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

For interactive scheduler jobs, only a limited set of batchoptions are supported. These batchoptions include the DSO configuration options, the design-type-specific options, and the following additional design-type-specific options that are not currently listed in the user interface or command line help windows:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **CreateStartingMesh:** Create the starting mesh only. Do not solve any other steps. Often used for multi-step jobs where the mesh creation is solved in a separate step using fewer resources than the rest of the analysis.
- **NumCoresPerDistributedTask:** Specifies the number of cores that are allocated to the job to use for each distributed task, used when running jobs under a scheduler. The scheduler communicates the number of cores allocated to the job on each host. This setting is used to determine how to allocate the cores to each task.
- **RAMLimitPercent:** Specifies the percentage of system RAM that the analysis is expected to use on each host. Running out of memory can cause processes to be killed by the Operating System or other failures. The analysis will be gracefully terminated if it requires more than this limit. Typical limits are about 90%, to allow some RAM for the Operating System and other services. If there are multiple jobs running on a single host, then the RAM limit should be reduced accordingly. For example, if there are two jobs running on a host, then the limit could be 45%, which would allow 10% of the RAM for the Operating System and other services.
- **RAMLimitPerCoreInGB:** This is an alternative way to specify the RAM limit, instead of using the RAMLimitPercent. In this case, the RAM limit is the limit per core multiplied by the number of cores allocated to the job on each host. Typically used for jobs running under a scheduler to ensure that the scheduler memory limit and the ansysedt memory limit are consistent.
- **SolveAdaptiveOnly:** Solve adaptive passes only. Do not solve any other steps. Often used for multi-step jobs where the adaptive passes are solved in a separate step using fewer resources than the rest of the analysis.
- **TotalNumOfCores:** The total number of cores for the job. This option is only used for EKM (Engineering Knowledge Manager).
- **ValidateOnly:** Only validate the specified setup, design, or project. Do not analyze the specified setup, design or project.

Any other batchoptions will result in a warning message and will be ignored.

Command Line Example:

```
desktopjob.exe" -cmd dso -jobid RSM_27086 -machinelist
list=localHost:2:2:90%
-monitor -ng -batchoptions "
'LargeScaleDSO/MergeCsv'='acrossDPsAndTraces'"
-batchsolve TeeModel:Optimetrics:ParametricSetup2 E:\work\2018\LS_
DSO\OptimTee.aedt
desktopjob.exe" -cmd dso -jobid RSM_25248 -machinelist
list=localHost:2:2:90%
-monitor -ng -batchoptions "
'LargeScaleDSO/MergeCsv'='acrossDPsByRow'"
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
-batchsolve TeeModel:Optimetrics:ParametricSetup2 E:\work\2018\LS_
DSO\OptimTee.aedt
ansysedt -batchoptions " 'TempDirectory'='C:\\TEMP'
'HFSS/SelectedDSOConfiguration'='Local'
'Desktop/Settings/ProjectOptions/DoAutoSave'=1
'LargeScaleDSO/MaxFolderInMB'=100 "
```

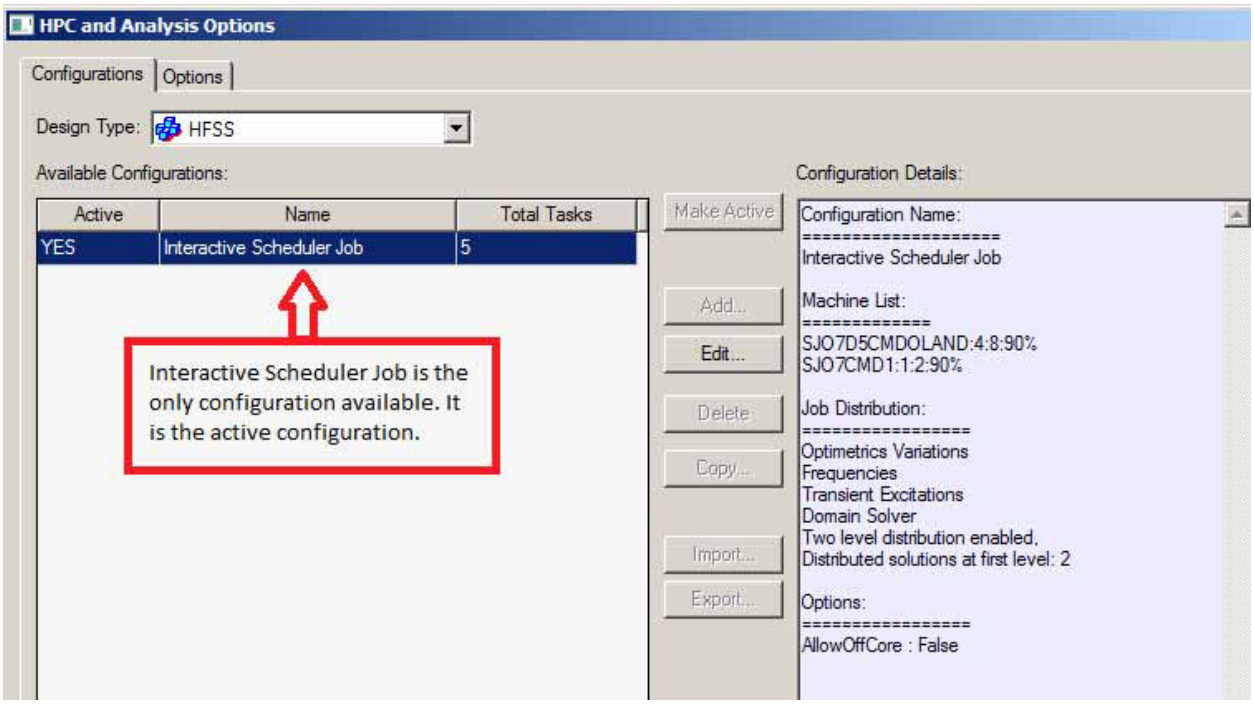
DSO Configuration for Interactive Scheduler Jobs

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

When running an interactive scheduler job, there is only one DSO configuration available for each design type. Each configuration is named "Interactive Scheduler Job". This configuration is always the active configuration for an interactive scheduler job. and it is the only configuration displayed in the list of available configurations shown in the "HPC and Analysis Options" dialog box. No configurations can be added or removed, but the "Interactive Scheduler Job" configuration may be modified using the "Edit" button, which pops up the "Analysis Configuration" dialog.

Because the "Interactive Scheduler Job" configuration is the only configuration accessible for interactive scheduler jobs and it is not accessible in other modes, there is no sharing of the "Interactive Scheduler Job" configuration settings with other modes.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



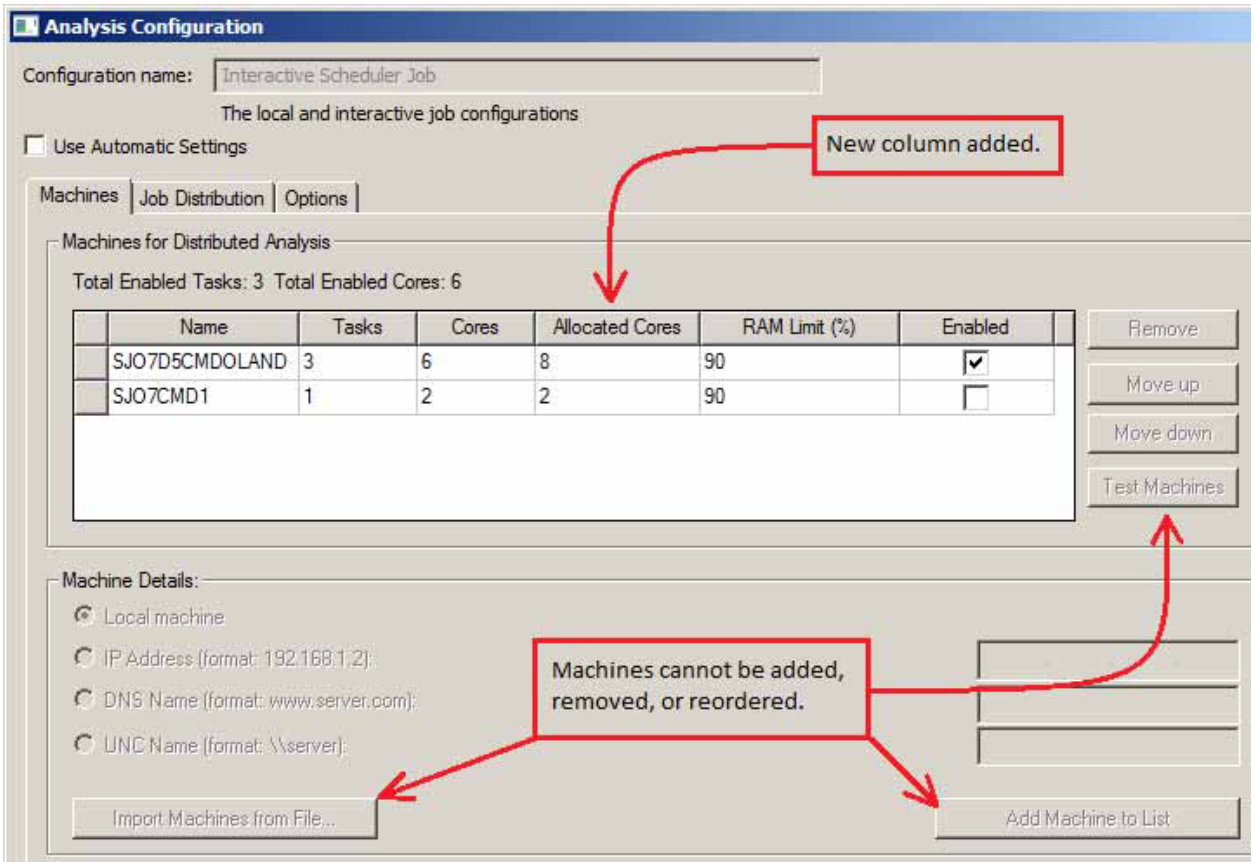
One major difference between interactive scheduler mode and batch mode or normal interactive mode is:

- In interactive scheduler mode, DSO configuration settings and design type options in the UI override command line options and batchoptions.
- When not in interactive scheduler mode, DSO configuration settings and design type options specified on the command line (including batchoptions) override UI settings.

Analysis Configuration - Machines Tab

The most obvious changes for interactive scheduler jobs are visible in the **Machines** tab of the **Analysis Configuration** window. The grid of machine information is prepopulated with the list of machines allocated to the job. It also contains an additional column in interactive scheduler mode. This column, "Allocated Cores", indicates the number of cores allocated to the job by the scheduler on each host. You cannot add or remove machines from the list, or modify the allocated cores for any machines. You can modify the tasks, cores, and RAM limittasks and cores for any machine, or specify that a machine is enabled or disabled.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Analysis Configuration - Job Distribution Tab

The **Job Distribution** tab of the **Analysis Configuration** window only appears if the **Use Automatic Settings** check box is not checked. The state of this check box, and the settings shown on this tab are initialized from the command line or from the registry. The command line options that affect the **Use Automatic Settings** check box or the job distribution settings are:

- -distributed
- -local
- -auto
- -machinelist

The registry contains the last value of these settings for an interactive scheduler job for the same user on the same host. Settings on the command line override settings from the registry. Any changes in the GUI will override the initial settings, even if the initial settings are from the command line. Any changes in the GUI also update the registry settings for the "Interactive Scheduler Job" configuration for the current design type.

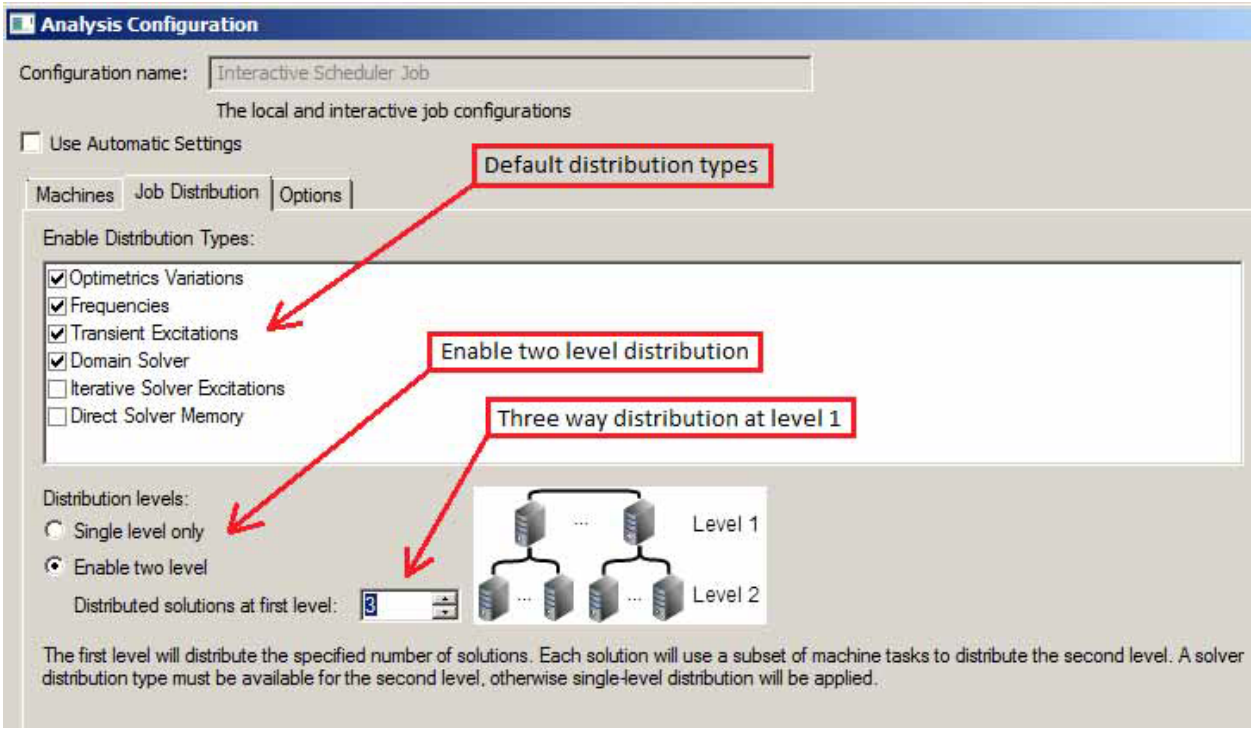
Example

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command line:

```
ansysedt -distributed includetypes=default maxlevels=2 numlevel1=3
```

Initial "Job Distribution" settings:



Analysis Configuration - Options Tab

You can use the **Options** tab of the **Analysis Configuration** window to examine or modify the DSO configuration options. The DSO configuration options are handled like the design type options, except that the Interactive Scheduler Job configuration settings are not shared with other modes.

The settings on this tab are initialized from the command line (using the -batchoptions command line option) or from the registry. The registry contains the last value of these settings for an interactive scheduler job for the same user on the same host. Settings on the command line override settings from the registry. Any changes in the GUI will override the initial settings, even if the initial settings are from the command line. Any changes in the GUI also update the registry settings for the "Interactive Scheduler Job" configuration for the current design type.

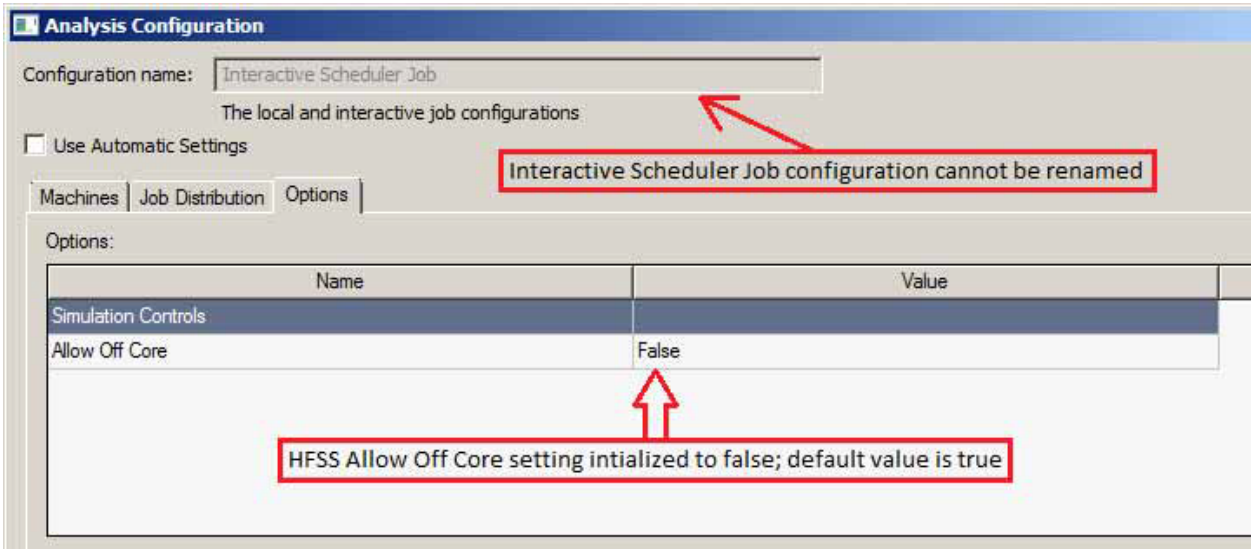
Example

Command line:

```
ansysedt -batchoptions 'HFSS/AllowOffCore'=0
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Initial Analysis Configuration Options:



Design Type Options for Interactive Scheduler Jobs

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

You can use the **Options** tab of the **HPC and Analysis Options** window to examine or modify the design type options. The settings on this tab are initialized from the command line (using the -batchoptions command line option) or from the registry. The registry contains the last value of these settings for the same user on the same host. Settings on the command line override settings from the registry. Any changes in the GUI will override the initial settings, even if the initial settings are from the command line. Any changes in the GUI also update the registry settings for the current design type.

These settings are shared between interactive scheduler mode and other modes.

Example

The following example includes two command line options, one that is applicable to HFSS ('EnableGPU') and one that is applicable to all design types ('HPCLicenseType'). These opitons would be included as part of the complete job submission command line syntax.

Command Line Excerpt:

```
-batchoptions 'HFSS/EnableGPU'=1 'HPCLicenseType'='Pack'
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Distribution Command Line Options

The user should include options specifying how the job should be distributed in the command line. These command line options will override options specified using the **HPC and Analysis Options** dialog box or obtained from the registry. See Running Ansys Electronics Desktop from a command line for more information on command line options.

For both scheduler batch jobs and interactive scheduler jobs, the -MachineList num=<num distributed tasks> format is the most common way to specify the number of tasks for the job. The other formats (-MachineList list=... or -MachineList file=...) allow the user to specify the number of tasks and cores to use on each host. These formats may be useful with clusters of heterogeneous machines, by allowing the user to specify different numbers of tasks or cores for different hosts. If either of the latter two formats is used, the user must ensure that the hosts and cores specified on the product command line are compatible with the hosts and cores allocated to the job.

Batchoptions for Interactive Scheduler Jobs

For interactive scheduler jobs, only a limited set of batchoptions are supported. These batchoptions include the DSO configuration options, the design-type-specific options, and the following additional design-type-specific options that are not currently listed in the user interface or command line help windows:

- **CreateStartingMesh:** Create the starting mesh only. Do not solve any other steps. Often used for multi-step jobs where the mesh creation is solved in a separate step using fewer resources than the rest of the analysis.
- **NumCoresPerDistributedTask:** Specifies the number of cores that are allocated to the job to use for each distributed task, used when running jobs under a scheduler. The scheduler communicates the number of cores allocated to the job on each host. This setting is used to determine how to allocate the cores to each task.
- **RAMLimitPercent:** Specifies the percentage of system RAM that the analysis is expected to use on each host. Running out of memory can cause processes to be killed by the Operating System or other failures. The analysis will be gracefully terminated if it requires more than this limit. Typical limits are about 90%, to allow some RAM for the Operating System and other services. If there are multiple jobs running on a single host, then the RAM limit should be reduced accordingly. For example, if there are two jobs running on a host, then the limit could be 45%, which would allow 10% of the RAM for the Operating System and other services.
- **RAMLimitPerCoreInGB:** This is an alternative way to specify the RAM limit, instead of using the RAMLimitPercent. In this case, the RAM limit is the limit per core multiplied by the number of cores allocated to the job on each host. Typically used for jobs running under a scheduler to ensure that the scheduler memory limit and the ansysedt memory limit are consistent.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **SolveAdaptiveOnly:** Solve adaptive passes only. Do not solve any other steps. Often used for multi-step jobs where the adaptive passes are solved in a separate step using fewer resources than the rest of the analysis.
- **TotalNumOfCores:** The total number of cores for the job. This option is only used for EKM (Engineering Knowledge Manager).
- **ValidateOnly:** Only validate the specified setup, design, or project. Do not analyze the specified setup, design or project.

Any other batchoptions will result in a warning message and will be ignored.

Example:

```
ansysedt -batchoptions " 'TempDirectory'='C:\\TEMP'
'HFSS/SelectedDSOConfiguration'='Local'
'Desktop/Settings/ProjectOptions/DoAutoSave'=1
'LargeScaleDSO/MaxFolderInMB'=100 "
```

Batchoptions

Batchoptions may be specified in the command line used to launch the product. Any valid batchoptions specified in the command line will override the associated registry settings. Batchoptions also override options specified using **HPC and Analysis Options** window or other dialog boxes used to specify options.

Setting the Number of Cores per Distributed Task

When submitting a job using the Ansys Electromagnetics Desktop job submission GUI, the number of cores per distributed task for a job is specified using the batchoption with pathname '<*DesignType*>/NumCoresPerDistributedTask', where <*DesignType*> is the design type to analyze. The batchoption setting is automatically included in the product command line when the job is submitted to the scheduler.

For interactive scheduler jobs, the user must include the associated batchoption setting or settings in the product command line when the product is launched. Multiple batchoption settings are required if the user analyzes multiple design types using the same product process. Batchoptions are the only way to specify this setting for batch jobs. There is an alternative to using the NumCoresPerDistributedTask batchoption for Interactive Scheduler Jobs, . The user may specify the total number of tasks and the total number of cores for each machine using the **Machines** tab of the **Analysis Configuration** window. You may use **Edit** in the **HPC and Analysis Options** to open the **Analysis Configuration** window for the "Interactive Scheduler Job" configuration.

Setting the Ram Limit Per Core in GB

When cores and RAM per core are requested, the cores could be allocated in an arbitrary/non-uniform way across nodes that themselves could be non-uniform/heterogeneous. For example, nodes could range from 4 cores to 20 cores and from 64 GB to 384 GB. In such environments,

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the RAM percentage set in the HPC configuration, Job Submission Compute Resources, or batch option may not be appropriate. In such cases, you can use the batchoption for RAMLimitPerCoreInGB. This setting specifies the Maximum amount of RAM used for each core allocated by the scheduler in GB. This batchoption cannot b e combined with RAM limit percent and is only valid when solving in a Linux scheduler environment.

The scheduler GUI automatically passes this new batch option instead of percent limit. From a scheduler GUI, such a request is available only for auto.

You can also use this new batch option for command line submission. The desktop does the computations and passes the percent limit to product/solver.

Setting the Remote Spawn Command Option to Scheduler

The Remote Spawn Command setting is only meaningful when running on the Linux Operating System. The value 'Scheduler' is valid if the job is a scheduler job running under an LSF, SGE or SLURM scheduler, and only if the MPI Vendor is 'Intel'.

When submitting a job using the AnsysEM job submission GUI, the Remote Spawn Command for an analysis may be specified using the batchoption with pathname 'DesignType/RemoteSpawnCommand', where DesignType is the Design Type to analyze. The Remote Spawn Command setting is only meaningful when running on the Linux Operating System. The value 'Scheduler' is valid if the job is a scheduler job running under an LSF, SGE or SLURM scheduler, and only if the MPI Vendor is 'Intel'. To specify the value 'Scheduler' for this option for a scheduler job, the Remote Spawn Command must be specified using the 'DesignType/RemoteSpawnCommand' batchoption in the product command line when the product is launched. In addition, the 'DesignType/MPIVendor' batchoption must be specified with value 'Intel' in the product command line when the product is launched. For interactive scheduler jobs, the Remote Spawn Command and the MPI Vendor may be specified with batchoptions or as design type options in the **HPC and Analysis Options** dialog box.

Setting MPI Version

When submitting a job using the Ansys Electromagnetics job submission GUI, the MPI Version may be specified using the batchoption with pathname DesignType/MPIVersion, where DesignType is the type of design to analyze (e.g. HFSS). It allows selection of which Intel MPI version to use, for both Windows and Linux. Valid values are "Default", "2018", and "2021".

For interactive solves, the MPI version may be specified with the batchoption in the command line used to launch the product, or as a design type option in the HPC and Analysis Options dialog box.

High Performance Computing (HPC) Integration

Ansys Electromagnetics products offer a direct integration with a number of High Performance Computing (HPC) software programs.

The list of currently supported schedulers includes:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Ansys Cloud Burst Computing
- Microsoft Windows HPC Server (Windows Only)
- Grid Engine (GE) (Linux Only)
- Platform's Load Sharing Facility (LSF) (Linux Only)
- PBS Pro or PBS Torque (Linux Only)

Electronics Desktop also supports custom integration.

A job scheduler may also be described as a batch system, a Distributed Resource Management System (DRMS) or Distributed Resource Manager (DRM). The features supported on each scheduler are included in the documents for each. For each job scheduler, the versions or revisions that have been tested are included.

A user may submit jobs using the command line tools or other tools provided by the scheduler. The Desktop includes a GUI to help the user submit jobs to a job scheduler. This generic Job Submission GUI is shared across the Ansys EM products. The general procedure is to specify the scheduler and head node, describe and submit the job, and monitor the results. Using Ansys Cloud Burst Compute will require a login through a browser instead.

Scheduler Terminology

This help uses some specific terminology when discussing HPC and schedulers. See definitions below.

Term	Definition
Core	Unit of processing.
Compute Cluster	Network of machines on which jobs run. Typically consists of head nodes and many compute nodes.
Distributed Processing	Multiple engines are launched simultaneously on the same machine *or* on different machines.
Engine	Electronics Desktop application (aka. executable) launched during analysis commands to generate analysis results.
Job	Application (aka. program, executable) with command line options that uses resources to produce results. For example, `hfss.exe -ng -BatchSolve`
Machine/Host/Node	Consists of one or more processors, memory, disk, etc.
Multiprocessing	A single engine uses multiple cores on the same machine.
Parallel Job	Job that runs on multiple cores belonging to the same or different machines.
Processor	Consists of one or more cores.
Resource	Machines, licenses, etc. that a job uses.
Serial Job	Job that runs on a single core.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Term	Definition
Service	Program that runs in the background (for example, RSM Service) and listens on a port. The OS provides the programming interface through which Applications communicate with services, once the machine and port number are known. Launching an executable on a remote machine requires a service to run on the remote machine.

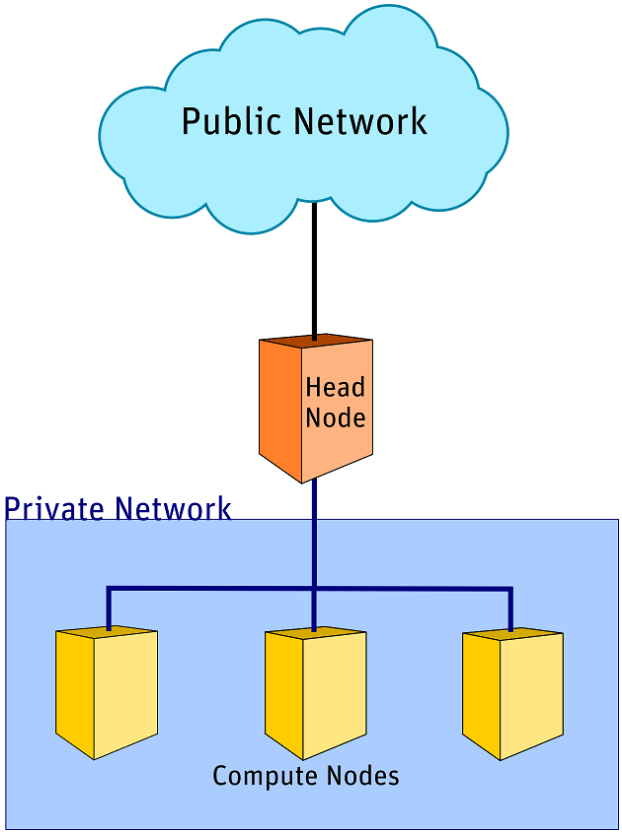
What a Scheduler Does

Schedules are responsible for the following:

- Enabling effective/efficient utilization of cluster's resources consistent with organization's goals
- Maintaining queue(s) of jobs
- Maximizing throughput of jobs by processing all jobs as quickly as possible (typically, one job per CPU)
- Allowing a choice of various scheduling policies (for example, First Come First Serve, Priority-Based, and Preemption)
- Providing a suite of tools or utilities (graphical or command line) for end users to submit, monitor, suspend, and abort jobs
- Managing a compute cluster by running various interacting services on head nodes and compute nodes
- Providing a programming interface to access services

Scheduler-Managed Compute Cluster

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Head node(s) typically maintain queues. Compute nodes are typically on a high speed network to improve scalability of parallel jobs. Services running on nodes interact with each other to manage resources. End user tools communicate with services to manage jobs.

Configuring Electronics Installation for HPC

In HPC applications, Ansys Electromagnetics Suite must be available on each cluster host where jobs may be run.

- On the Linux platform, Ansys EM may be installed on a shared drive that is accessible to all machines in the cluster.
- On the Windows platform, Ansys EM must be installed separately on each host of the cluster.

Ansys EM must be accessible using the same path on each host. All cluster users running Ansys EM jobs must have permission to read and execute the files in the installation directory and its subdirectories.

The temp directory selected during installation must be readable and writable by all user accounts used to run the Ansys Electromagnetics Suite. This temp directory path should be the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

same on all machines of the cluster and should be local to every machine. For example, C:\temp on Windows, /tmp on Linux.

Because HPC is offered as a direct integration, you need only install the Ansys EM software. No additional configuration is required.

Firewall Configuration

If firewall is turned *off* between the machines of the cluster, there is no need for any configuration.

If firewall is turned *on*:

- For a Windows cluster: Configure firewall by adding exceptions that allow Ansys EM programs and services to communicate with each other. If you are using a standard Windows Firewall, this is automatically done for you by the installation program. If you are using a third-party firewall software, it needs to be configured in a similar manner.
- For a Linux cluster: Open up the firewall for range of ports denoting ephemeral (or dynamic) ports. Check with your system administrator on how this can be done on each machine of cluster.

Installation Directory Examples

Microsoft Windows

Install the Ansys Electromagnetics Suite in directory "C:\Program Files\ANSYS Inc\v252\AnsysEM" on each node of the cluster. The same directory pathname must be used on all hosts.

Linux

Install the Ansys Electromagnetics Suite in a common directory that is accessible using the path /opt/ansys_inc/v252/AnsysEM on each execution node of the cluster.

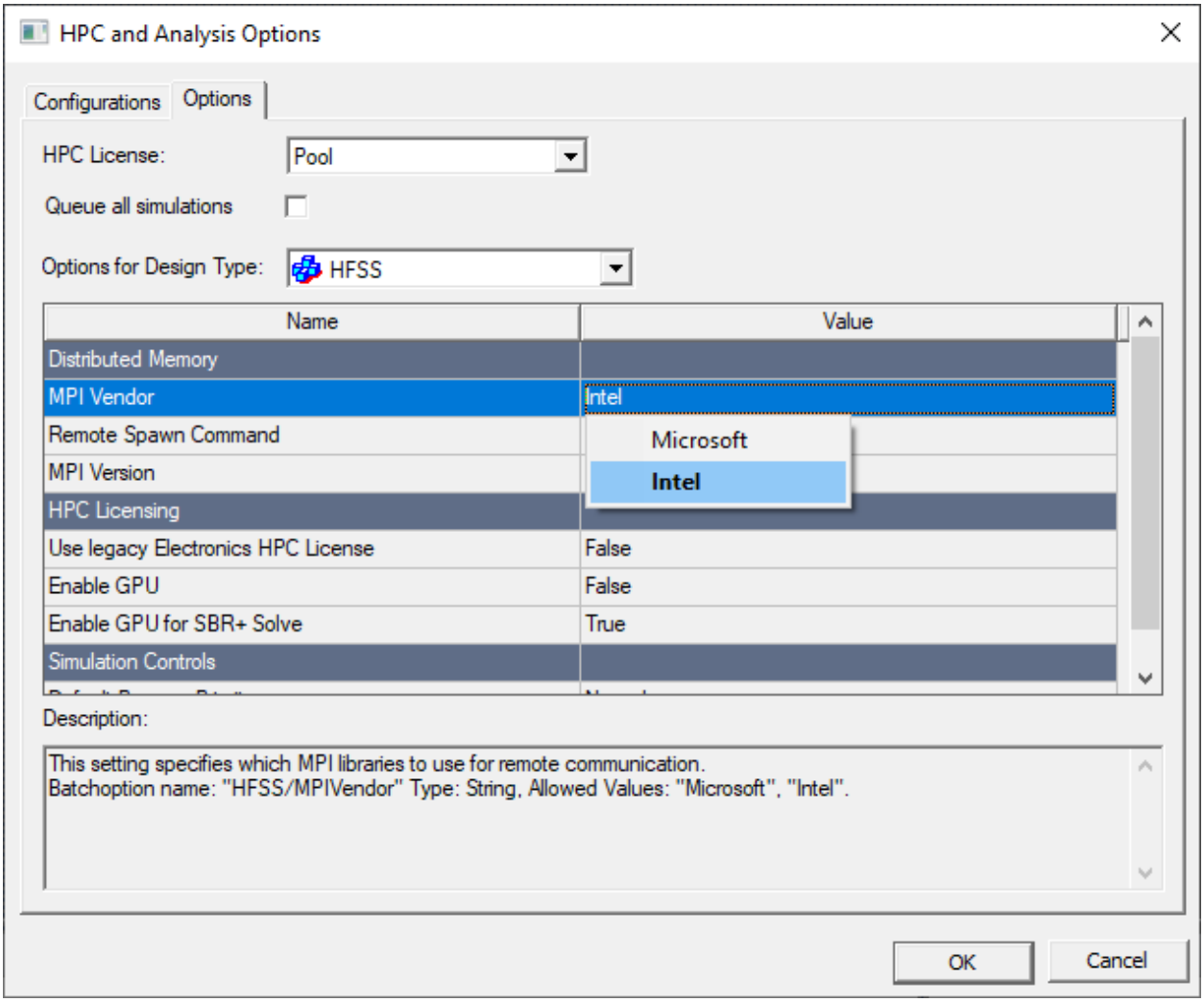
MPI Vendor and MPI Version

For HFSS, HFSS 3D Layout, Q3D Extractor, Maxwell, and Icepak design types:

These settings may be used to specify the MPI Vendor and Version for distribution via MPI.

Go to the **Options** tab of the **HPC and Analysis Options** dialog box, and select the design type. Then you can select the MPI Vendor and MPI version.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



MPI Vendor

The default MPI Vendor is "Intel". On Linux, "Intel" is the only supported MPI vendor. On Windows, both "Intel" and "Microsoft" are supported MPI vendor options. For jobs distributed to multiple hosts, the "Microsoft" MPI Vendor is only supported when running as a Windows HPC job.

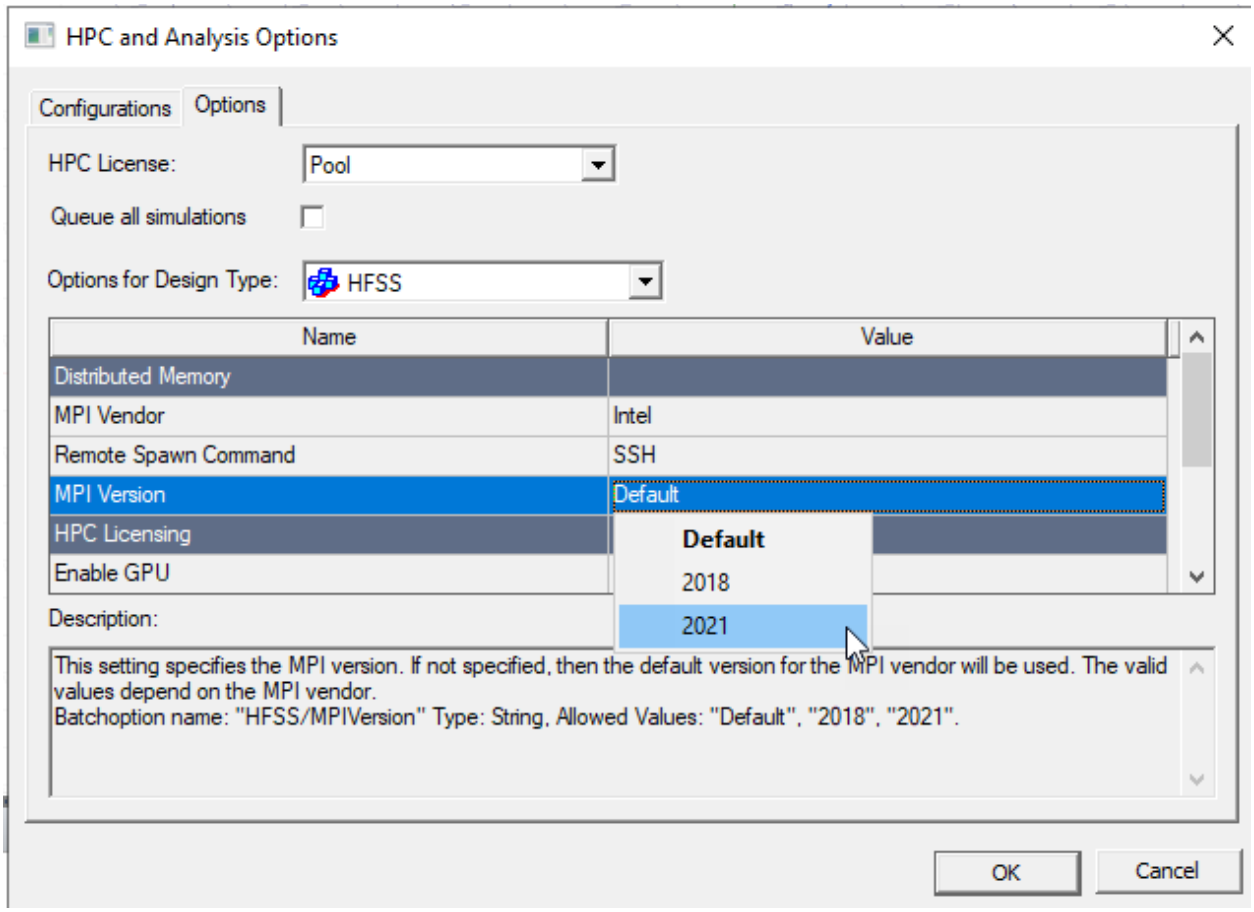
MPI Version

If only a single MPI Version is supported for the selected MPI Vendor, then the MPI Version option is ignored. Multiple versions are only supported for Intel MPI, as follows:

- The value "Default" indicates that the default Intel MPI version should be used. This is Intel MPI 2021 in most cases.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The value "2018" indicates that Intel MPI 2018 should be used.
- The value "2021" indicates that Intel MPI 2021 should be used.



Also see Setting Intel MPI Interconnect for more details.

Integration with Microsoft Windows® HPC Scheduler

The Windows HPC scheduler is only supported on Windows.

Jobs may be submitted in any of the following ways:

- Using Windows HPC GUIs from Microsoft: Job Manager or Cluster Manager
- Using Windows HPC command line tools
- Using Ansys Electronics Desktop UI commands for scheduler selection, job submission and job monitoring/control.

General Guidelines for Submitting Ansys EM Jobs

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A Job submitted to Windows HPC Cluster is defined by Job properties, Task List and Task properties. Priority, resource requirements, node preferences, etc. come from Job properties. In the case of Ansys Electromagnetics jobs, Task List consists of a single task. Properties of this task specify the command line that runs Ansys Electromagnetics desktop in non-graphical mode to perform analysis of a project.

Specifying the Number of Compute Resource Units for HPC Jobs

You can either select **Use automatic settings** on the *Compute Resources* tab, or you either enter the number of tasks and total cores per machine, or individual nodes. HFSS, HFSS 3D Layout, and Icepak have **Use automatic settings** selected by default. This release permits options in setting whether nodes are exclusive for the submission. See Windows HPC Non-exclusive Jobs.

Ansys EM Project File and Project Directory for use with Windows HPC Scheduler

Ansys Electromagnetics Suite tools write their results to a subdirectory of the directory containing the Ansys EM project file. The Project Directory (the directory containing the project file) must be accessible to all of the cluster hosts that may run Ansys EM jobs. The user account for the job must have permission to read the project directory, and to create and modify files and subdirectories of this directory. The pathname of the project file must be accessible to all cluster hosts using the same path name, which is generally expressed as a UNC pathname.

Example:

The project file is on the user's workstation (with hostname user1_PC) in directory C:\user1\projects\new\project1.aedt, and the directory C:\user1\projects is shared with sharename projects.

Correct

When submitting the job, you should use the following pathname to specify the project file:

```
\\user1_PC\projects\new\project1.aedt
```

Incorrect

If a local pathname is used, the cluster hosts will not be able to find the user's project on the workstation

```
user1_PC: ' C:\user1\projects\new\project1.aedt '
```

Submitting and Monitoring Ansys EM HPC Jobs

Jobs may be submitted to the Windows HPC Scheduler using any of the following methods:

- Using the *Submit HPC Job* dialog box
- Using the Windows HPC Job Manager GUI

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Using the Windows HPC Command Line Tools
- Using the Windows PowerShell

Client Utilities from the Microsoft HPC Pack, must be installed on the submit host to use any of these methods to submit a job to a cluster. The *Submit HPC Job* dialog box will be unable to contact the cluster head node if the client utilities are not installed.

This document covers the first method. See the Microsoft documentation for information on the other three methods.

- Submitting and Monitoring Jobs for Windows HPC
- Specifying the Number of Compute Resource Units for HPC Jobs

Jobs may be submitted from any Microsoft Windows host meeting the following requirements:

- For submitting jobs to the Windows HPC scheduler, the Desktop process must run on a node that is configured for submission of jobs to the Windows HPC cluster. That is, the Windows HPC Client Utilities must be installed on the node, and network communication from the Desktop node to the head node of the cluster must be allowed. For Ansys Electromagnetics Suite 2025 R2, Windows HPC Server 2008 R2 (or later) client utilities are required. Using a computer on the network is not supported for submission of jobs to the Windows HPC cluster.
- When submitting jobs to a Windows HPC cluster, the user must also specify the head node of the cluster to which the jobs will be submitted. When the user selects the "Windows HPC" scheduler in the "Choose scheduler" list, the Head Node edit control is enabled. The user may enter the Windows HPC cluster head node name into the edit box. Alternatively, the head node may be selected using a "Browse for Computer" browser by pressing the ellipsis [...] button.
- The Windows HPC Pack client utilities are installed on the submission host.
- Network communication between the submission host and the Windows HPC Cluster head node is permitted; there is a network connection between these hosts that is not blocked by any firewall or the like.
- The submission user is permitted to submit jobs to the Windows HPC Cluster.

Job Monitoring

1. Windows HPC Jobs may be monitored using the *Monitor Job* dialog box, which is brought up by the **Tools > Job Management > Monitor Jobs...** command. This dialog bos may also be brought up by checking the **Begin monitoring this job now** check box when a job is successfully submitting using the job submission dialog box. You can also select the **Simulation** tab of the ribbon and select the **Monitor** icon. You can monitor this job either automatically (by checking the option) or through the **Tools> Job Management> Monitor Jobs...** command. For more details, see Monitor Jobs window.

In addition to the above requirements to allow job monitoring the following is also necessary:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Network communication between the submission host and all Windows HPC Cluster nodes where the job may run is permitted; there is a network connection between these hosts that is not blocked by any firewall or the like.

Cluster Configuration

Any job running on a Windows HPC Cluster that is distributed over multiple compute hosts requires network communication between processes running on these hosts. The cluster must be configured to allow this communication. Any firewall or other security software must be disabled or configured to allow communication between any of the compute hosts were a job could run.

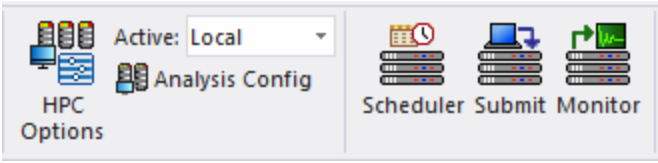
Job Submission User Profile on Cluster Compute Nodes

In order for a job to run correctly, the submission user's profile must be accessible and properly initialized on the cluster compute nodes where the job runs. If the Ansoft/temp subdirectory of the user's "My Documents" directory does not exist or is not accessible on the compute cluster nodes where a job runs, the batchoptions for the job will not be processed correctly, resulting in job failure. One way to ensure that this directory is created on each compute host is for the submission user to login to each compute host and run the product GUI one time.

Submitting and Monitoring Jobs for Windows HPC

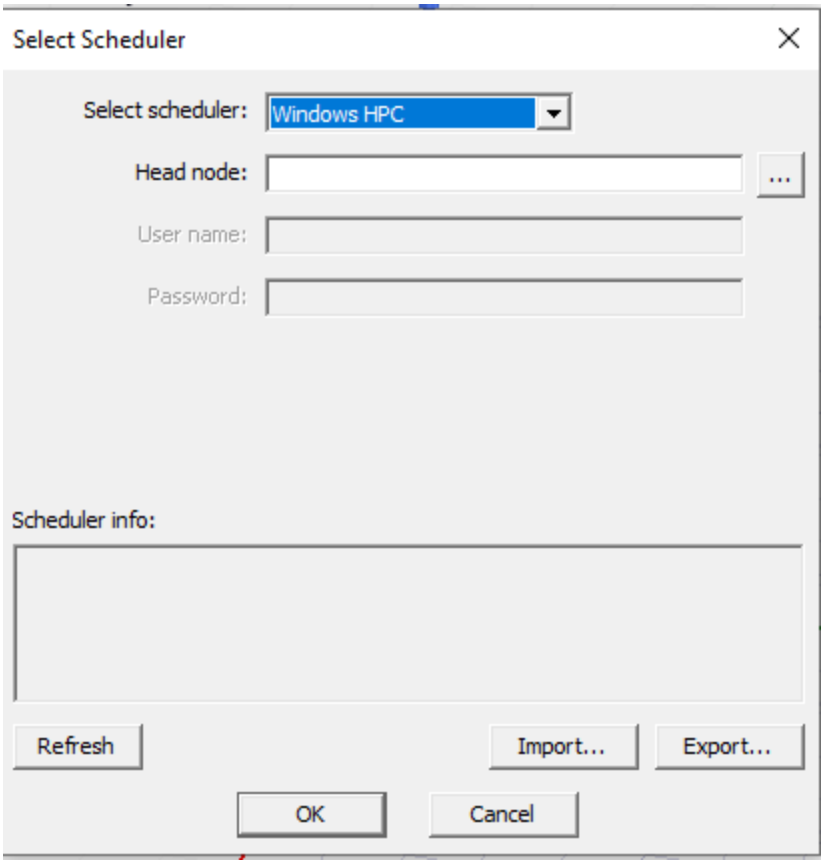
In order to submit jobs using **Windows HPC**:

- Click **Tools** > **Job Management** > **Select Scheduler**, or
- Select the **Simulation** tab of the ribbon, and click the **Scheduler** icon.



This opens the **Select Scheduler** dialog box. Specify **Windows HPC** as the scheduler.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For Windows HPC, specify the head node of the cluster.

After specifying the cluster head node, click **Refresh**. This verifies that the head node may be contacted, and displays the scheduler name, a brief description (including the head node name), and the version of the Windows HPC head node.

Pressing **Cancel** discards changes made in this dialog box. Pressing **OK** verifies that the head node can be contacted before accepting the changes. If no problem occurs, the dialog box will close. If there is a problem contacting the head node, the dialog box will not close and the changes will not be accepted.

After setting the job submission node, perform *one* of the following:

- Select **Tools** > **Job Management** > **Submit Job...**
- Click **Project** > **Submit Job...**
- Click **Maxwell** > **Submit Job...**
- Select the **Simulation** tab and click the **Submit** icon.
- Use a shortcut menu to select **Submit Job**.

This opens the **Submit Job To** dialog box.

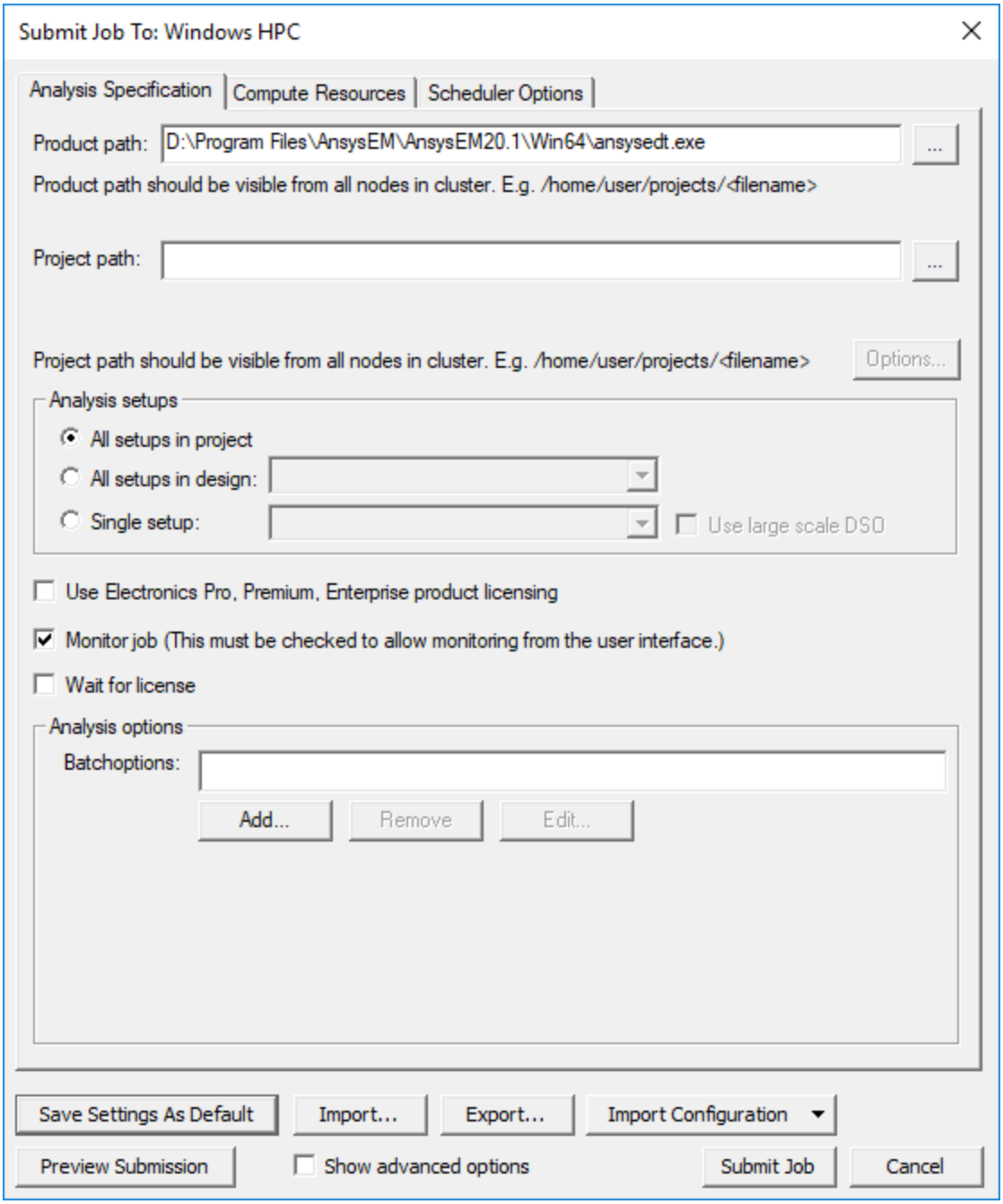
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Submit Job To** dialog box contains three tabs:

- **Analysis Specification**– specify the Product path, Project name, the setups, and analysis options such as batchoptions, or, for advanced users, Environment variables. If you select the Analysis or Optimetrics setup, the Analysis Specification is pre-populated.
- **Compute Resources**– this tab can be populated either by automatic settings, by predefined Analysis Configuration, or specifying parameters in the fields for resource selection, for job parallelization and enabled forms of parallelization.
- **Scheduler Options**– contains fields for Job name and priority. The customization options shown by checking advanced are not used for Windows HPC.

On the **Analysis Specification** tab, enter path names for the Product and Project. These must be UNC paths that are accessible from each compute host used for Ansys Electromagnetics jobs. The Project can be an archive. The submission user must have permission to write to the directory containing the project file.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can select which setups are analyzed in the **Analysis Setups** area of the dialog box. There are radio buttons to select:

- All setups in the project
- All setups in a specified design (you select the design from the drop-down menu)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

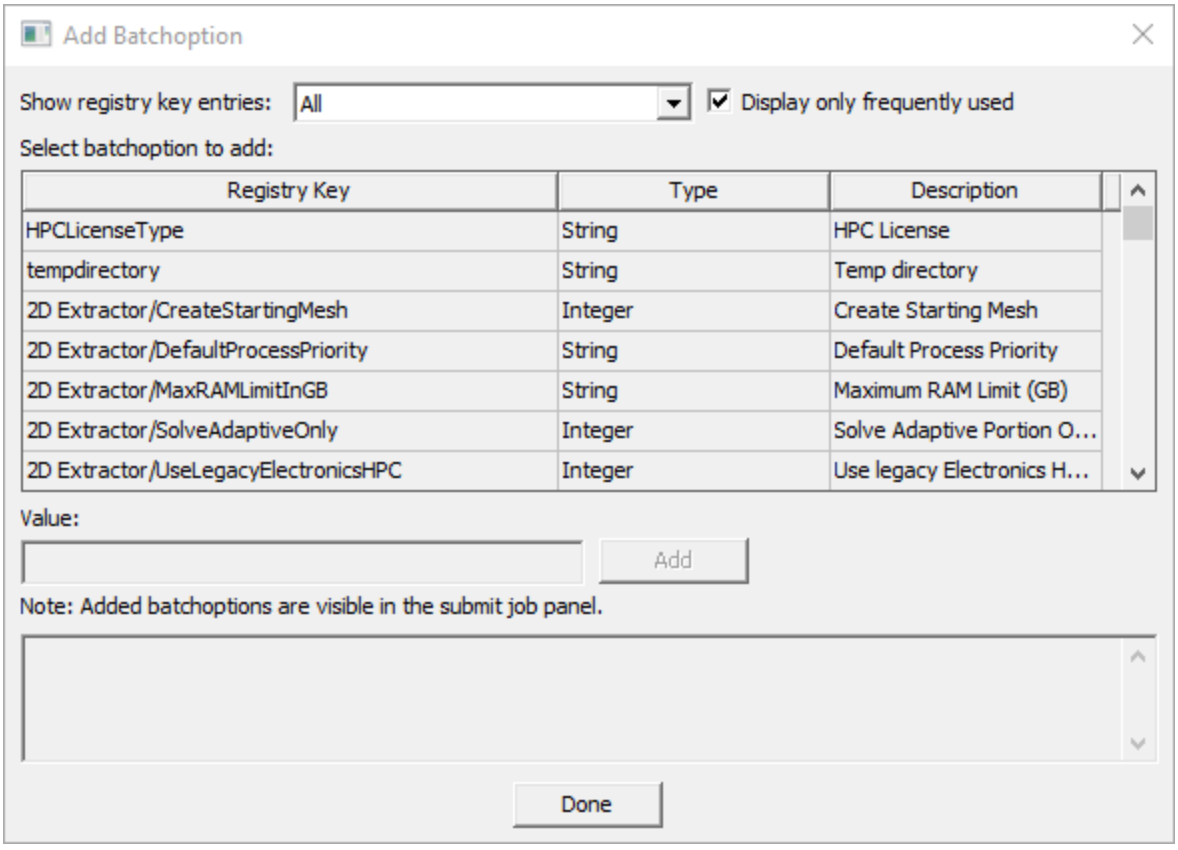
- Single setup (if you selected the Submit Job command from the shortcut menu, the setup name populates the field)

If you specify multiple setups, they will be processed sequentially in the order displayed in the edit box.

The Analysis options include:

- Monitor job – you must enable this option to monitor the job from the user interface.
- Wait for license – specify whether to wait until a license if available before starting a simulation.
- Batch options – optionally specify -batchoptions in the text field. See detailed discussion of -batchoptions beginning under Running Ansys Electronics Desktop from a Command Line.

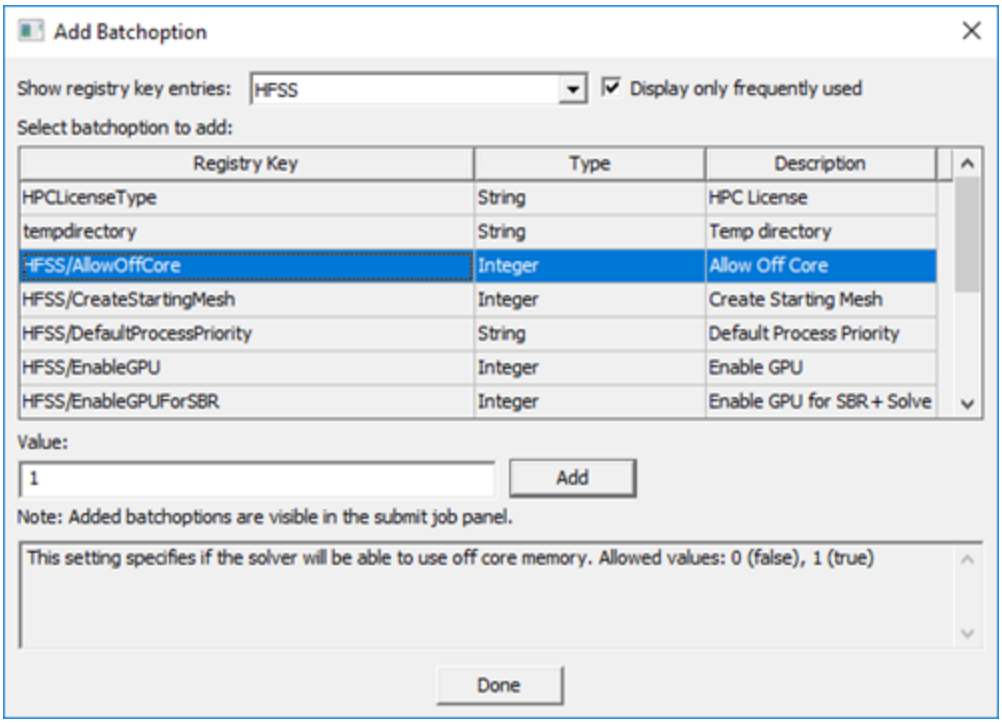
The **Add...** button opens the **Add Batchoption** dialog box.



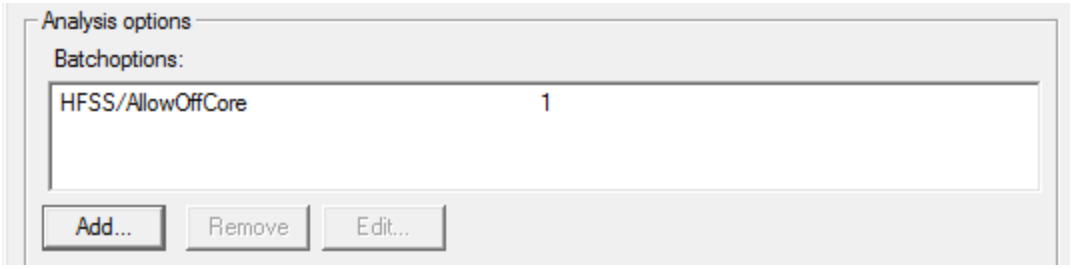
This dialog box provides access to all -batchoption commands. The drop-down menu lets you select specific categories, and you can choose to display only frequently used commands. You can edit and remove any batch options you specify.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select a Registry Key in order to show the current Value for the type. The lower field explains the meaning of the Type Value.

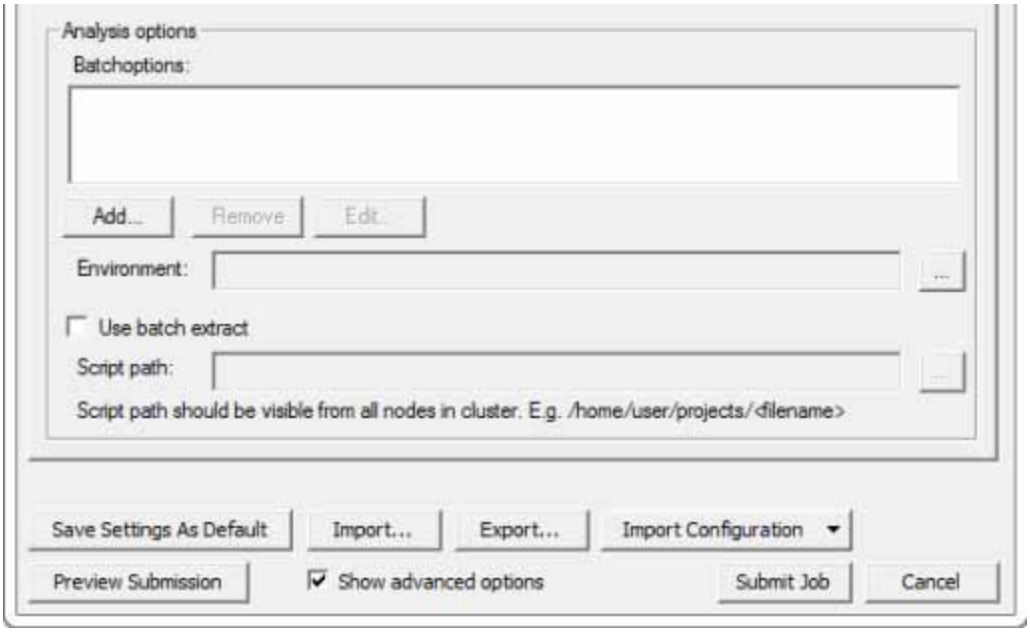


Any batchoptions for which you select **Add** will be visible in the **Submit Job** dialog box.

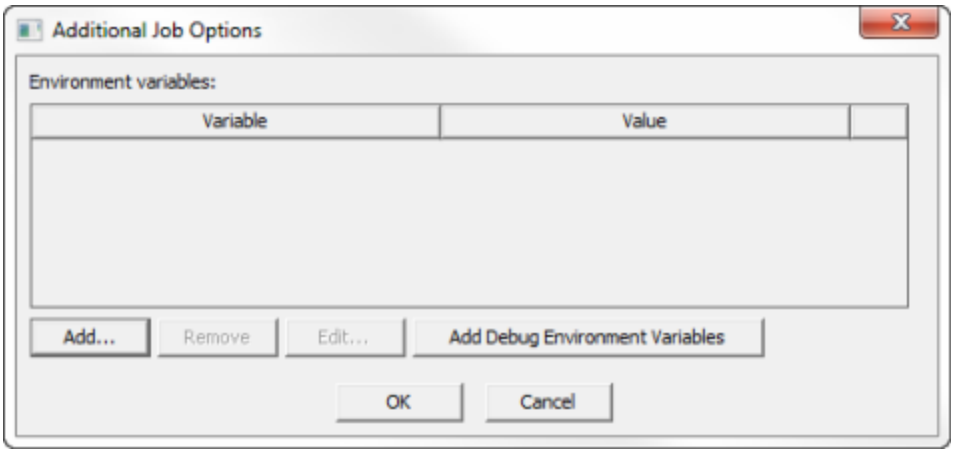


If you have the **Show advanced options** box checked in the **Submit Job** dialog box, the Environment field and the Use batch extract fields display.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

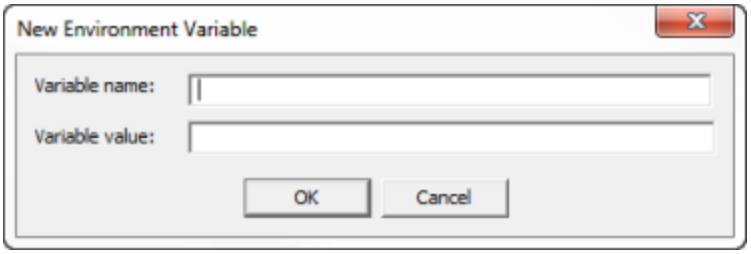


The **Environment** field lets you specify any environment variables. Click the ellipsis button [...] to display the **Additional Job Options** dialog box.



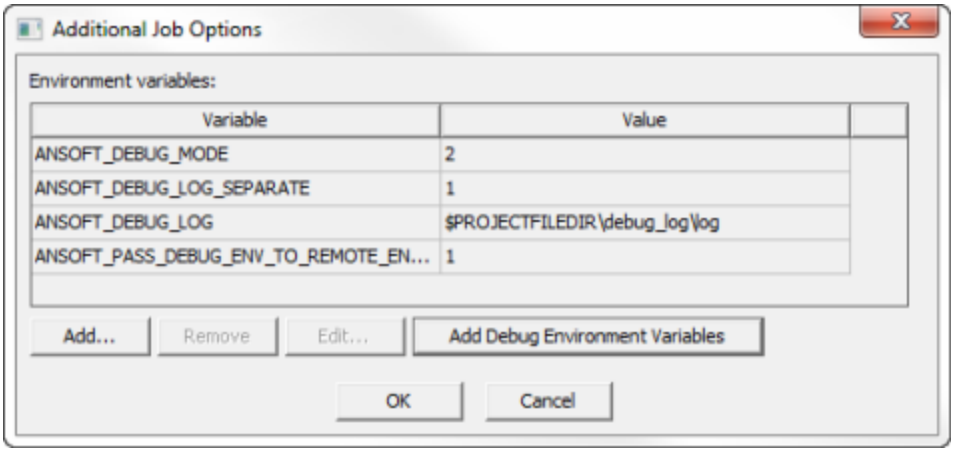
Click the **Add...** button to open the **New Environment Variable** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Here you can provide a Variable name and Variable value. Click **OK** to display the variable in the **Additional Job Options** dialog box.

Select a variable to enable the **Remove** and **Edit...** buttons. You can also click **Add Debug Environment Variables**.



> **Note:**
>
> For certain methods of resource selection for Windows HPC job submission, Ansys Electronics Desktop checks for nodes being online. In an auto-scaling cluster, nodes can fail the online check because they're not online until a job is running. An environment variable allows customers to bypass this check:
>
> ANSYSEM_SKIP_NODE_ONLINE_CHECK
>
> Set the variable value to 1 to disable the online check. Debug logging fully verifies what aspects of the cluster are causing this check to fail.

Job submission in Windows works without changing settings if environment variable ANSYSEM_SUBMIT_JOB_REQ_NODE_ONLINE is set with value 0. The installation doesn't set this environment variable. ANSYSEM_SUBMIT_JOB_REQ_NODE_ONLINE accepted

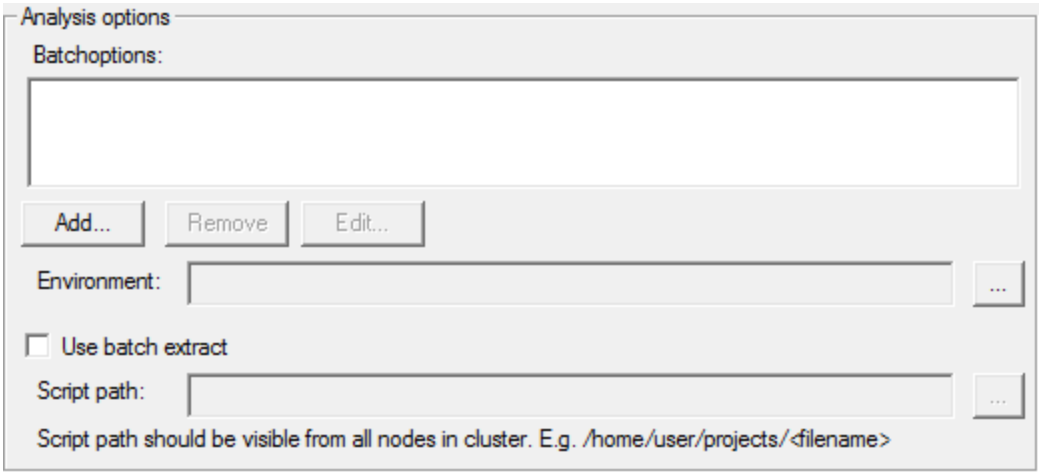
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

values: 0 or 1 (0 = disable; 1 = enable, i.e. require nodes to be online for submission). Default: 1 (if not set, Ansys Electronics Desktop requires nodes to be online for submission).

If you have enabled **Show Advanced Option**s, any variables that you add will be displayed in the **Environment** field of the **Submit Job** dialog box.

Use Batch Extract for Windows HPC

Enabling **Show Advanced Options** for Windows HPC also displays the **Use batch extract** check box and script path field.

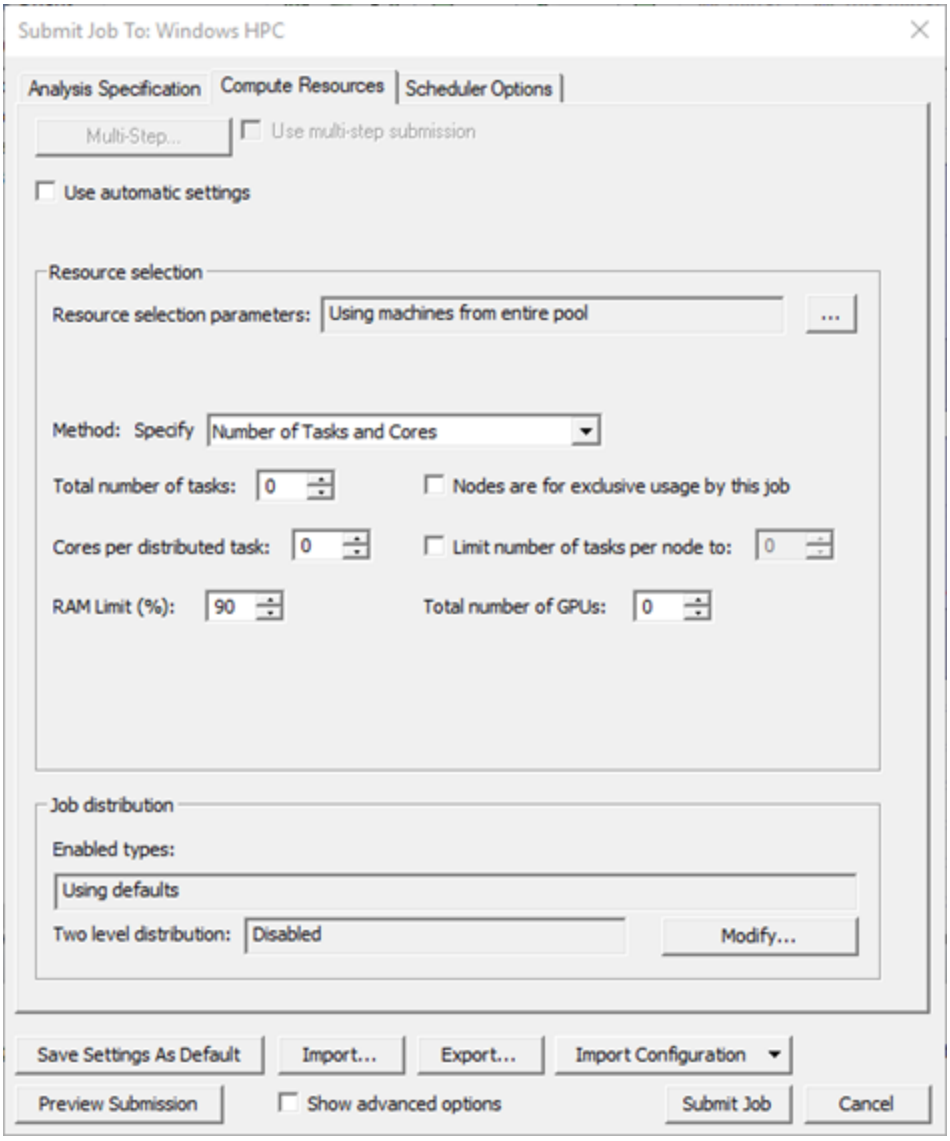


See Running Ansys Electronics Desktop from a Command line for an explanation of the solve information available through batch extract.

The **Preview Submission** button opens a window that shows the text commands that will be sent to the scheduler.

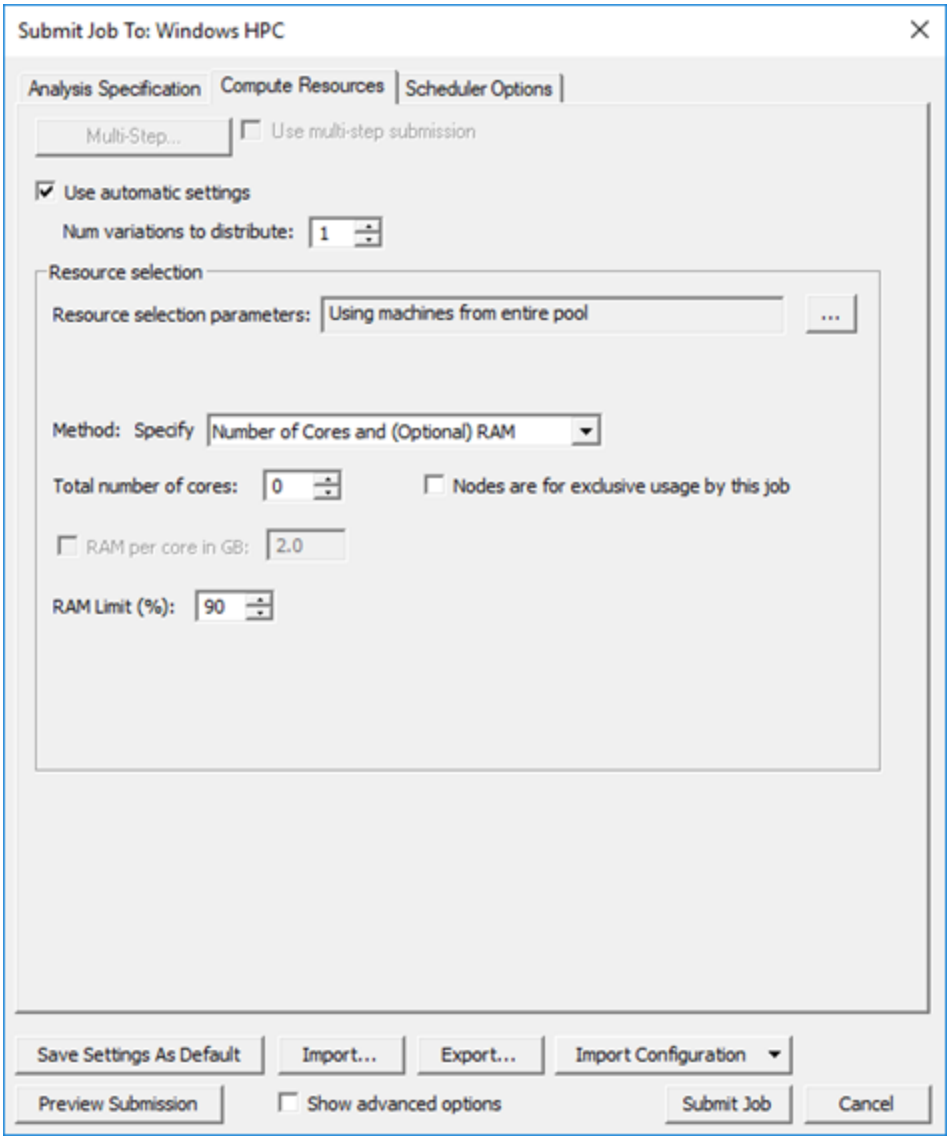
The following figure shows the **Compute Resources** tab of the **Submit Job To** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For Ansys Electronics Desktop configurations, the **Submit Job To** dialog box includes a **Use automatic settings** check box that simplifies the **Compute Resources** tab.

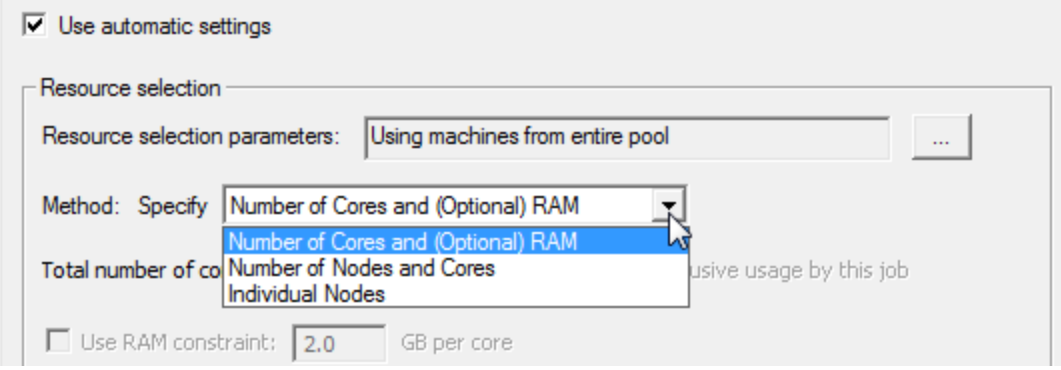
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



With **Use automatic settings** selected, the **Job distribution** field is removed. When using automatic settings, you can specify **Resource selection** parameters. The ellipsis button [...] opens the Compute Resource Selection Parameters dialog box. If you do not specify any parameters, the default is "Using machines from entire pool."

The Method field of the **Submit Job To** dialog box has a drop-down menu with two or three selections, depending on whether you select **Use automatic settings**.

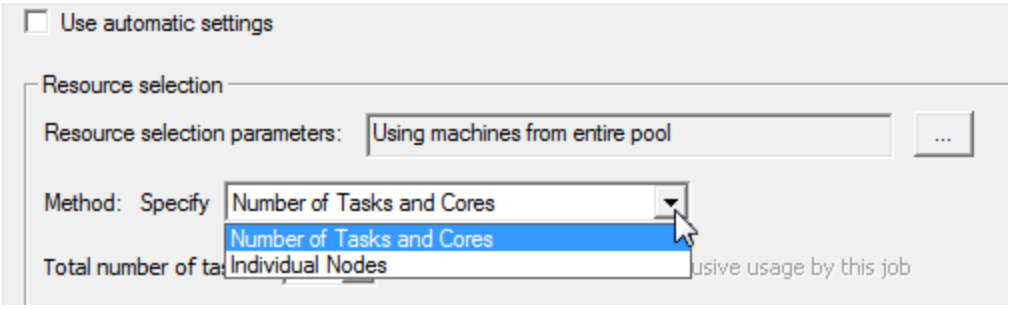
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

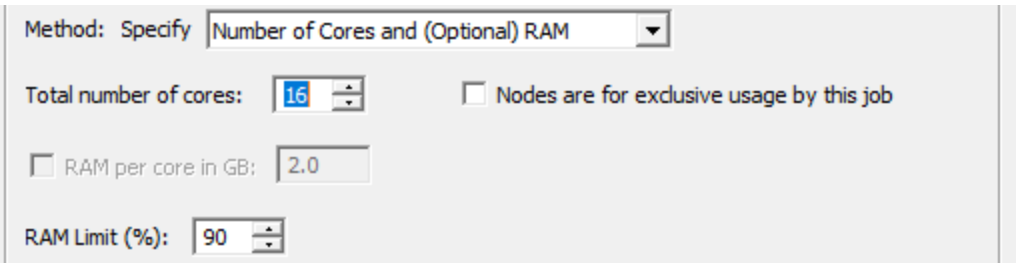
If you select Use automatic settings, Optimetrics variations will be solved sequentially. Other distribution types will be distributed automatically. It does distribute frequencies, domains, and use of multiple level domains.

If you uncheck or cannot access Use automatic settings, these two Methods are listed:



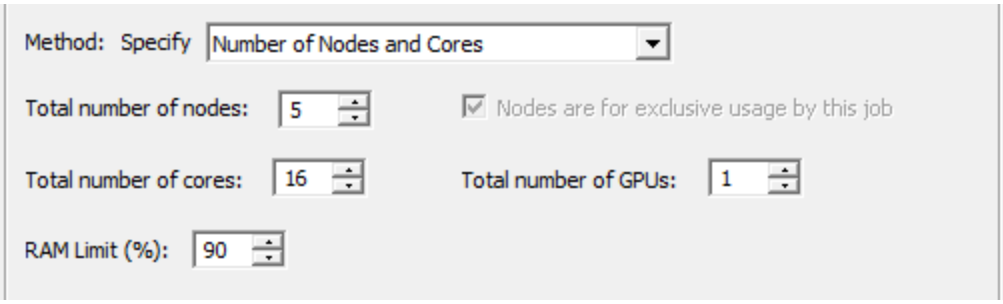
For Windows, there is a 64 core limit for a single machine. Each Method selection changes the available options listed:

- Specify Number of Cores and (Optional) RAM

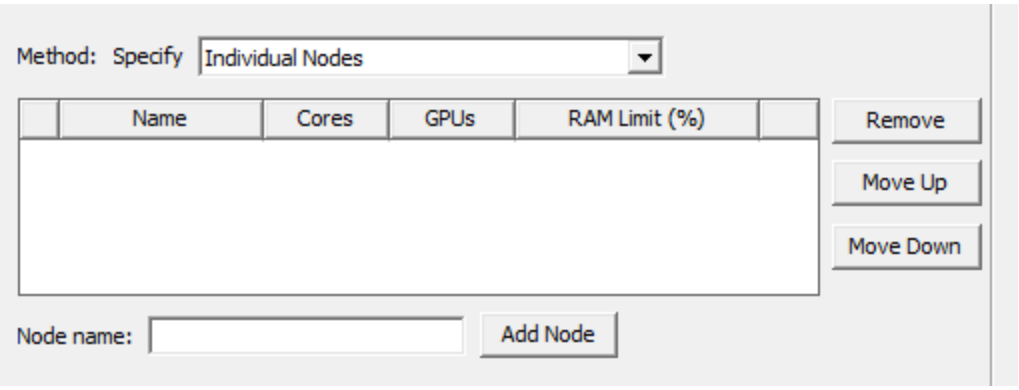


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Number of Nodes and Cores

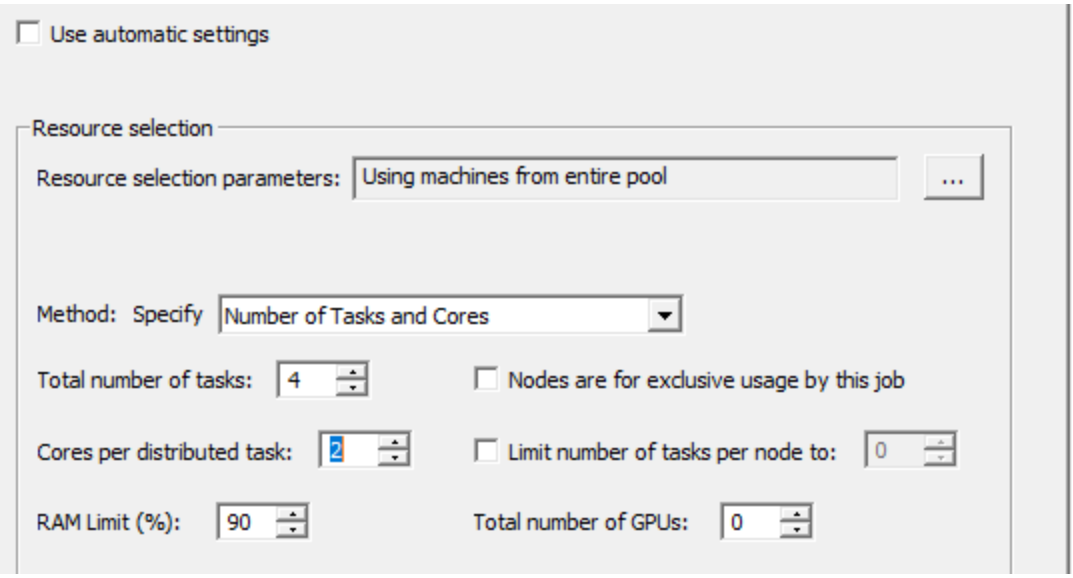


- Individual Nodes



- Number of Tasks and Cores ("Use automatic settings" is unchecked for this option. Checking "Use automatic settings" means that you do not have to specify tasks or core parameters):

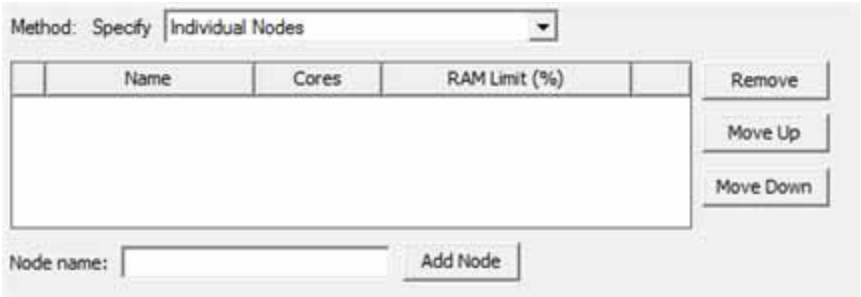
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Individual Node List

For Windows HPC jobs, you may either specify a node list, or specify job parallelization parameters, but not both.

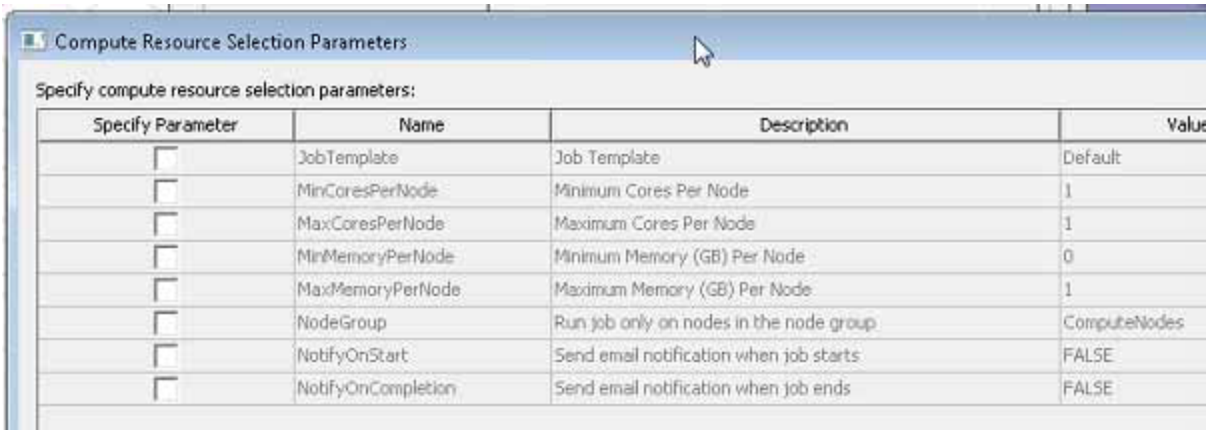
If you select the Individual Nodes Method, you may specify a node list, and the Job parallelization controls are disabled. In this case, the node list should only include cluster nodes that are valid for the job. For each node, you enter the node name and add the node. In the table, you can specify the number of cores and the RAM limit as a percentage. You can use the Remove, Move Up and Move Down buttons to edit and order the list.



Compute Resource Selection Dialog

By default, you can draw from the entire pool. You can also click the ellipsis button [...] to open a **Compute Resource Selection** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The resource selection parameters for Windows HPC jobs are:

- JobTemplate: Job Template - The JobTemplate may limit the job parameters or specify defaults values for job parameters
- MinCoresPerNode: Minimum Cores Per Node
- MaxCoresPerNode: Maximum Cores Per Node
- MinMemoryPerNode: Minimum Memory (GB) Per Node
- MaxMemoryPerNode: Maximum Memory (GB) Per Node
- NodeGroup: Run job only on nodes in the node group
- NotifyOnStart: If True, send email notification when job starts. Email notifications must be configured and enabled for the cluster by the administrator. (The cluster head node must run Windows HPC Server 2008 or above.)
- NotifyOnCompletion: If True, send email notification when job ends. Email notifications must be configured and enabled for the cluster by the administrator. (The cluster head node must run Windows HPC Server 2008 or above.)

Job Parallelization

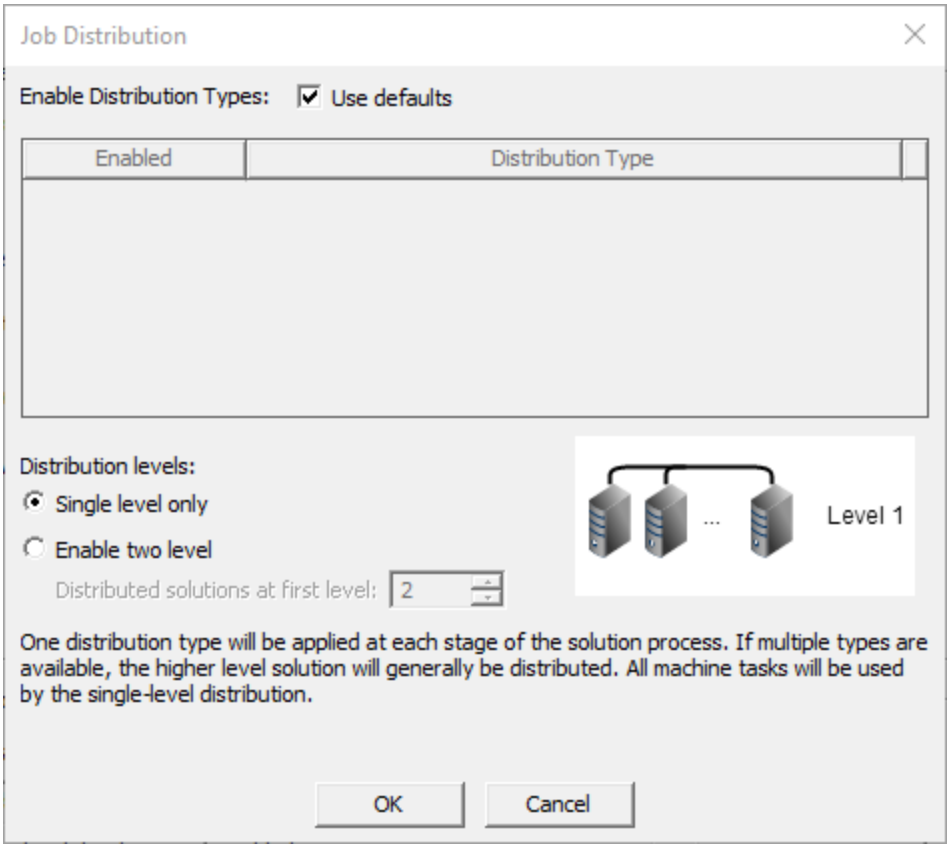
For Windows HPC jobs, you may either specify a node list, or specify the job parallelization parameters, but not both. The Job parallelization fields let you specify:

- Total number of tasks: The number of nodes requested for the job is the total number of tasks divided by limit on the number of tasks per node, rounded up if it is not an integer.
- Cores per distributed task. This determines the amount of multiprocessing per task.
- Whether nodes are for exclusive usage by this job
- Whether to limit the number of tasks per node to a value. If the "Limit number of tasks per node" check box is not checked, then the job is submitted with a job unit type of "Core".

Job Distribution

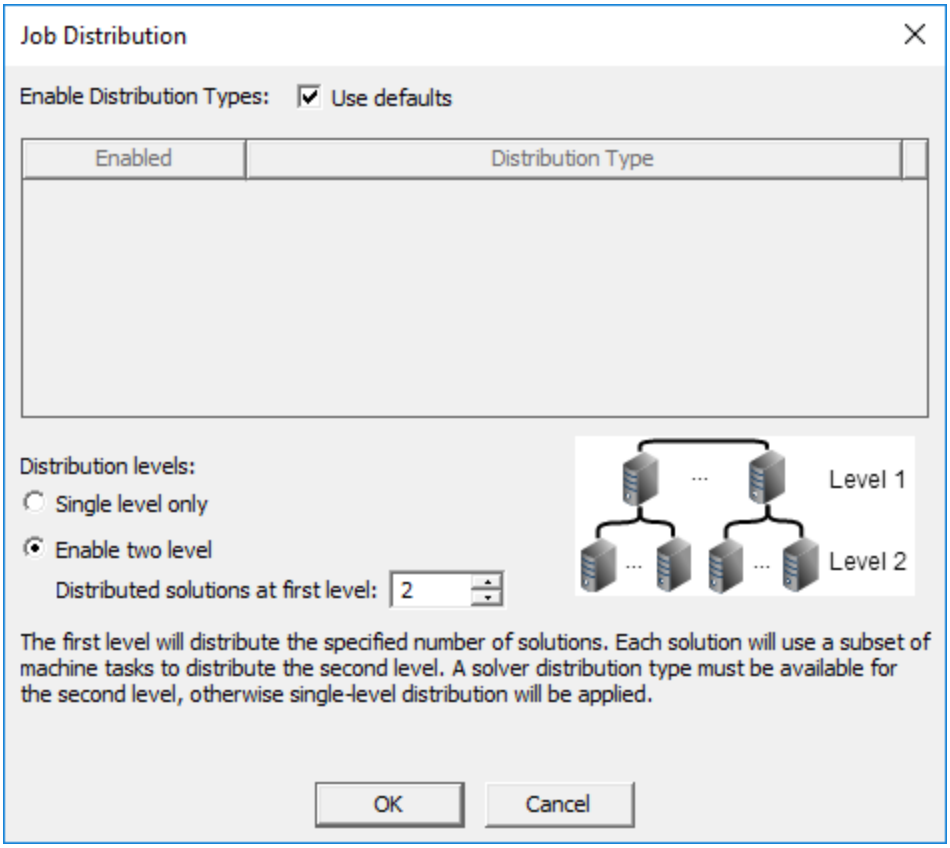
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Single level or two level distribution (*single level* is the default). Click **Modify** to display the **Job Distribution** dialog box and select the **Enable two level** option if applicable and desired.



- Second level distribution operates within DSO. If available and enabled you can specify the number distributed solutions for level 1.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

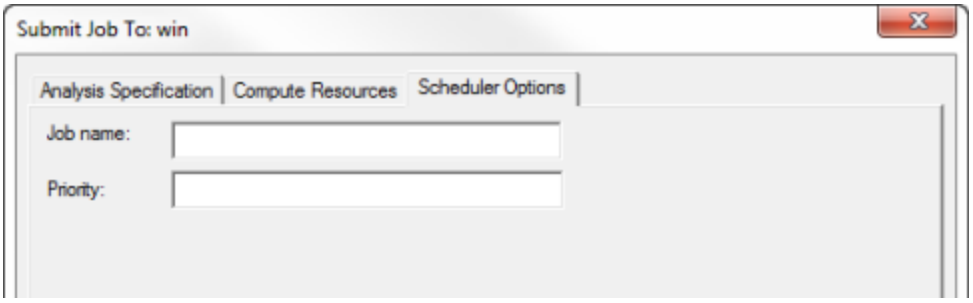


In response to a set of minimal constraints, the Scheduler may increase the resources assigned beyond the minimal values in order to meet the full set of requirements. For example, if you specify 7 distributed engines, with two processors per engine, and also limit the number of engines per node to 4, the scheduler may increase the number of cores used in order to meet the limit specified for engines per node. Notice that a preview of the Submit Job Results shows the number of resources assigned, and that the scheduler generated code includes an MPI specification.

Scheduler Options

The **Scheduler Options** tab provides for specifying the job name and/or the job priority. While the **Show advanced options** check box enables the display of Job submission options, no job submission options should be specified for Windows HPC.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Preview Submission

The Preview Submission button opens a window that shows a text description of the job to be submitted and the task used to start the product on one of the nodes.

The JOB PARAMETERS section contains information on parameter that apply to the job as a whole.

- The "Job resource parameters" section indicates whether the job has exclusive use of nodes, the job unit type, and the minimum and maximum number of units requested for the job, node group, and email notifications.
- The "Job attributes" section displays the job name and job priority.
- The "User Specified Compute Resource Attributes" displays the Resource selection settings.

The TASK PARAMETERS section contains information on parameters that apply to the Desktop task, which is the main task of the job.

- The "Desktop task resource parameters" section indicates the job unit type (which is the same as in the JOB PARAMETERS), and the minimum and maximum number of units requested for the Desktop task.
- The "Command Line section" displays the desktop task command line, including all arguments.
- The "Environment variables" section displays the environment variables that are set for the Desktop task; the same environment variables will also apply to all other tasks of the job.
- The "Working directory" section indicates the working directory in which the Desktop task will run.

Monitor Job

If you have checked the Monitor Job option on the **Submit Job To** dialog box, **AnalysisSpecification** tab, you can invoke the **Monitor Job** window by clicking **Tools** > **Job Management** > **Monitor Jobs...** This dialog box may also be brought up by checking the **Begin**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

monitoring this job now check box when a job is successfully submitting using the job submission dialog box. For more details, see: Monitor Jobs Window.

Windows® HPC Job Templates

The job templates are managed by the Windows HPC cluster administrator. Every cluster has at least one job template, the "Default" job template. Every job has an associated job template. If no job template is specified, then the "Default" job template is used. The job template controls two related aspects of the job submission process. When a job is submitted, there are a number of job parameters which may be specified. Each parameter has a set of valid values. For example, the Priority parameter has five valid values, Highest, AboveNormal, Normal, BelowNormal, and Lowest. The job template controls the default value of each parameter; this is the value that the parameter has if it is not specifically overridden by the submitter. For example, in the Default job template, the default value of the Priority parameter is Normal. The job template may also limit the allowed values of each parameter to a subset of the valid values. For example, a job template for privileged users could allow all five Priority values, which a job template for unprivileged users could limit the allowed Priority values to Normal, BelowNormal and Lowest.

Each job template is a Windows object with access controlled by an ACL (access control list). Instead of the usual "Read" or "Read & Execute" permissions, there is a "Submit Job" permission which corresponds to the right to submit a job with this job template. The cluster administrator may create job templates to limit or control access to cluster resources. For example, a job template with limited allowed job run times, or access to a limited set of compute nodes could be created by the cluster administrator. Specific users or user groups could be forced to use this limited job template by omitting access to the other job templates or by adding a deny access entry for the specified user or group to the other job templates. See the *Microsoft HPC Pack 2019: Job Templates* white paper from Microsoft for additional details:

https://learn.microsoft.com/en-us/powershell/high-performance-computing/job-templates?view=hpc19-ps

Job templates may also be created to allow users to run jobs with limited knowledge of the appropriate job parameters. The cluster administrator creates a job template which has reasonable default values for the type of job to be run, and informs users which job template to use for each type of job. The template could also limit some parameters to only the subset of all values that are useful for the type of job associated with the template.

Selecting Computation Resource Units (Job Unit Type)

The Job Unit Type is the smallest unit of processing resources used to schedule the job. This is one of the most important job properties. There are three options for the Job Unit Type: cores, nodes or sockets.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Cores**: Jobs are scheduled in units of cores, which may be also described as a CPU cores, logical processors, or CPUs. This is the smallest unit of granularity available. This selection allows the scheduler to start multiple tasks on a processor, if the total number of cores needed by the tasks is less than or equal to the number of cores on the processor. This selection may also allow the scheduler to distribute more of the computational load to processors with more cores than to processors with fewer cores. For Windows, there is a 64 core limit for a single machine.
- **Nodes**: Jobs are scheduled in units of nodes, hosts or machines. This is the coarsest level of granularity that may be selected. When this option is selected, only one task will run on any give node at any given time. This is useful in cases where it is not desirable to run multiple tasks on a single host. For example, if each task is multi-threaded, running multiple tasks on the same node may not be needed to fully utilize the computing resources on the node. This may also be preferred if the tasks are memory intensive, and multiple tasks would be competing for the limited memory resources.
- **Sockets**: A socket (which may also be called a NUMA node) is a collection of cores sharing a direct connection to memory. A socket will contain at least one core, and it may contain several cores. The socket concept may not necessarily correspond to a physical socket. Scheduling at the socket level may be useful in cases in which each task requires extensive use of the memory bus, and scheduling multiple tasks on the same socket would result in excessive bus contention.

Windows® HPC Non-Exclusive Jobs

Windows HPC allows three choices for the Cluster AffinityType setting:

- No Jobs: Affinity is not set for any jobs
- Non Exclusive Jobs (default): Affinity is not set for exclusive jobs
- All Jobs: Affinity is set for all jobs

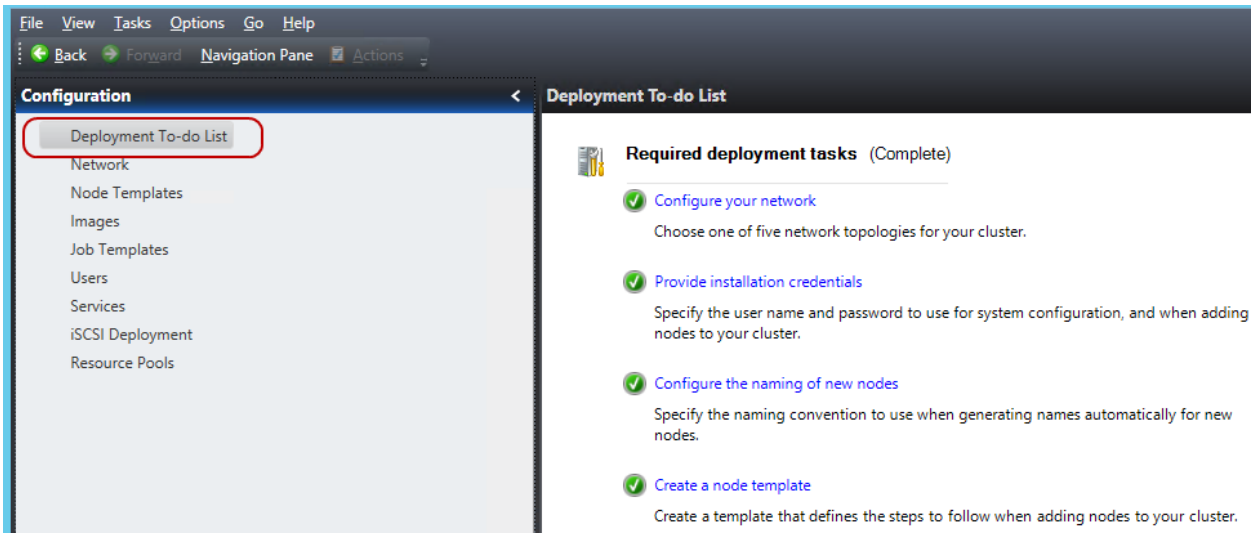
If the AffinityType is set to “No Jobs,” then we can allow non-exclusive jobs in many cases, but non-exclusive jobs will be disallowed if you specify an explicit host list (because all resources on all specified hosts are used), if you specify nodes and cores (to ensure that the job has exclusive access to the specified number of nodes), or if you specify multi-threading (to prevent a single task from being split across hosts). If the AffinityType is set to “Non Exclusive Jobs,” then exclusive jobs will be disallowed for all cases, the same as the restrictions for the previous release.

The previous job submission dialog always has the “exclusive” check box disabled for Windows HPC jobs, so that “exclusive” jobs cannot be submitted to a Windows HPC cluster. The “exclusive” check box is now enabled for Windows HPC jobs. This allows you to submit exclusive jobs to a Windows HPC cluster. If this check box is checked, but the submission parameters and Windows HPC cluster AffinityType do not allow submission of an exclusive job, then an error message is displayed when you submit or preview the job.

Setting the Windows HPC Cluster Affinity Type

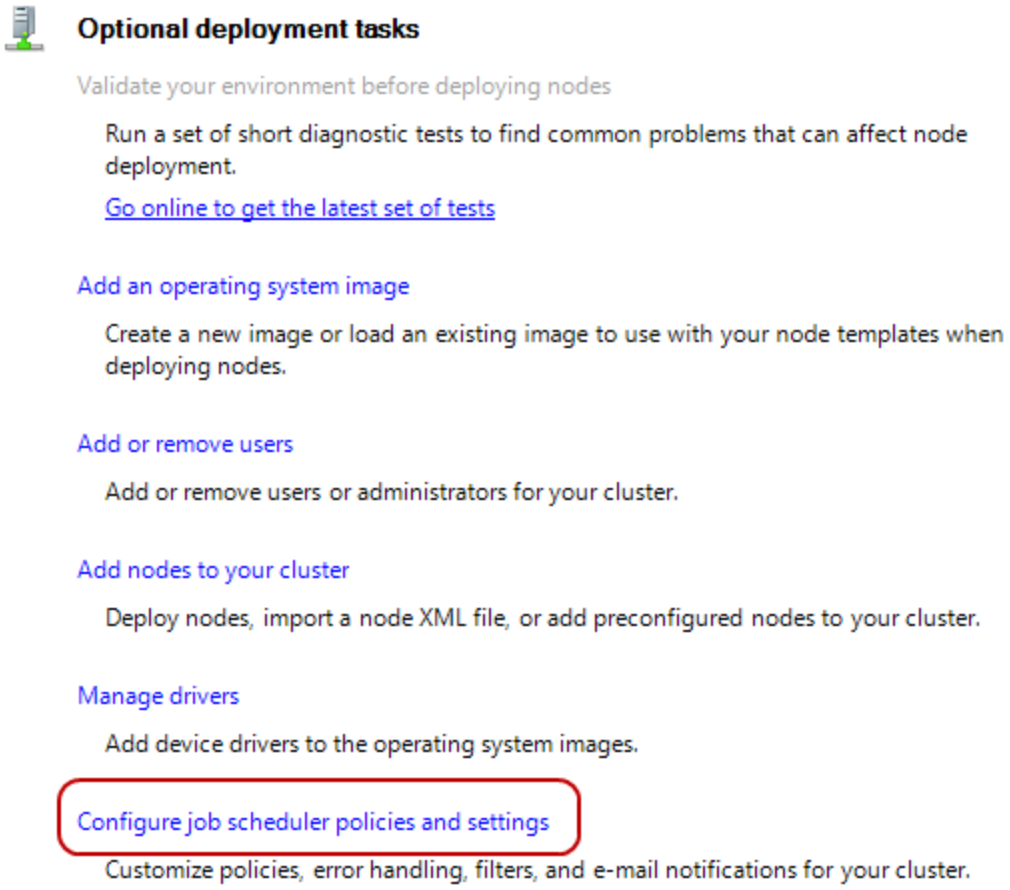
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A Windows HPC Cluster administrator is able to modify the Affinity Type setting for the cluster. The administrator should start the Windows HPC Cluster Manager. The first step is to select the Configuration button on the lower left pane, then select the Deployment To-do List in the upper left pane. Then, the upper right pane displays the Deployment To-do List, as shown below.



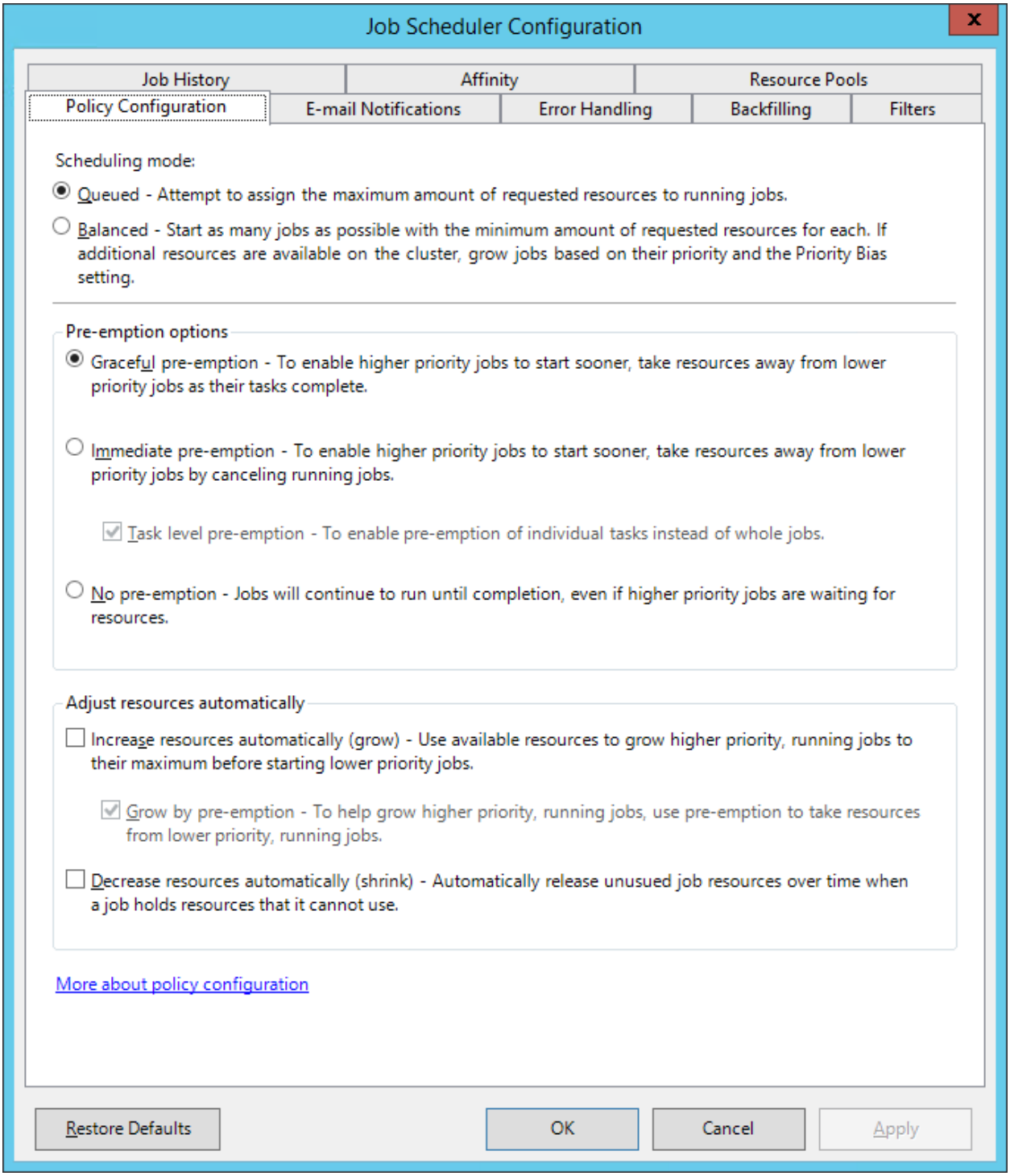
Next, the "Configure job scheduler policies and settings" should be selected. This is under the heading "Optional deployment tasks."

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



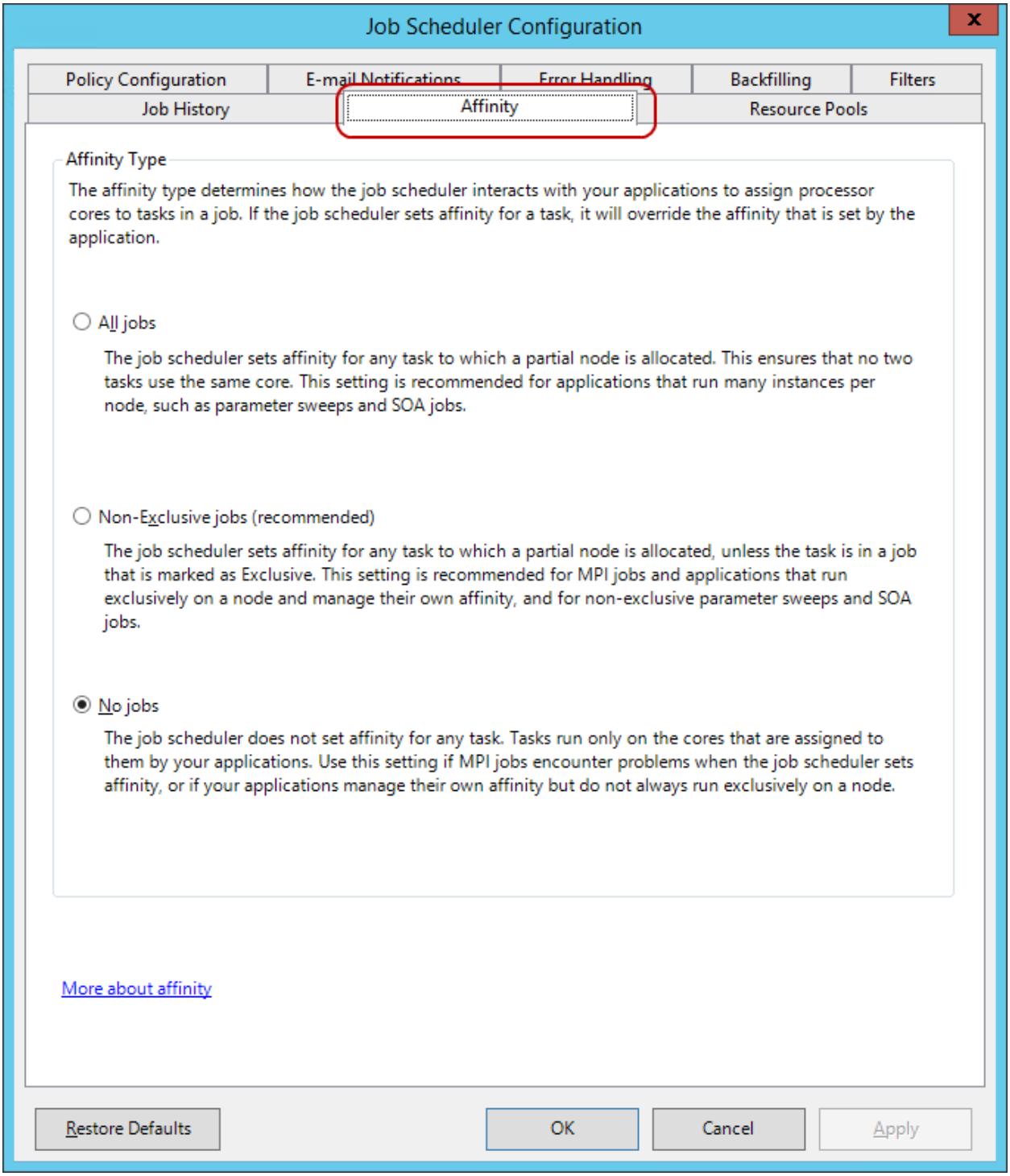
The *Job Scheduler Configuration* dialog box pops up, as shown below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Next, select the "Affinity" tab should be selected.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The cluster administrator may now select a radio button to select the desired Affinity Type:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- All jobs
- Non-Exclusive jobs
- No jobs

In order to submit non-exclusive jobs, the "No jobs" option should be selected. The "Non-Exclusive jobs" option may be selected, but this will not allow non-exclusive Ansys Electromagnetics jobs to be submitted to the cluster. The "All jobs" option is not supported for Ansys Electromagnetics jobs. The OK or Apply buttons may be used to apply the selected option to the cluster.

Examples with "Submissions with Nodes are for exclusive usage by this job" Unchecked and Checked

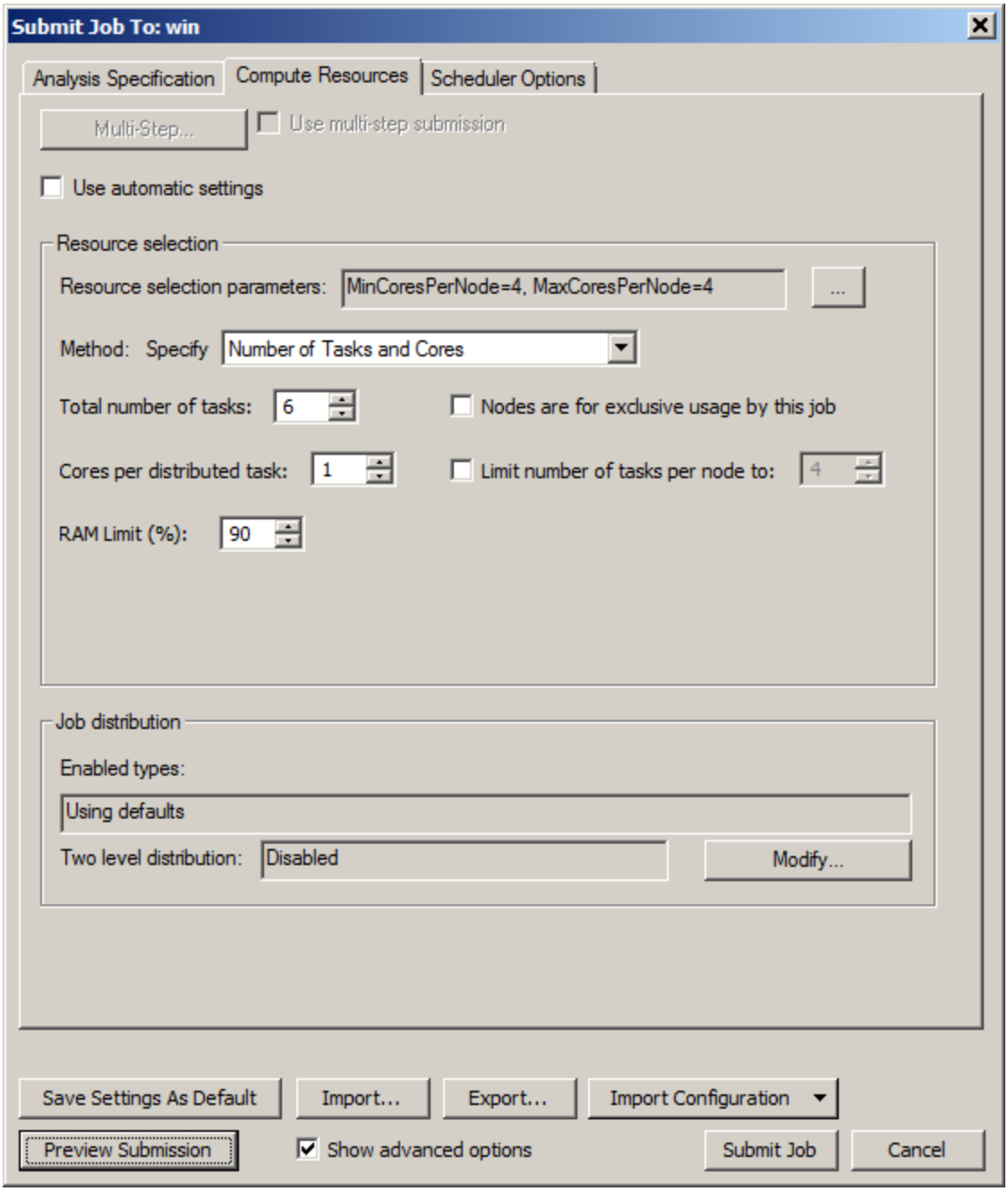
Most of the following screen shots are for the same project file submitted to a Windows HPC cluster with the AffinityType set to "No Jobs". The scheduler resource selection parameters are the same for all of these screen shots:

- MinCoresPerNode=4
- MaxCoresPerNode=4

The first set of screen shots show submission of a job to a Windows HPC cluster in which the "Use automatic settings" check box is not checked, and the resource selection method is "Specify Number of Tasks and Cores". The exclusive setting for the job is controlled by the "Nodes are for exclusive usage by this job" check box. In previous releases, this check box was always disabled and checked for Windows HPC jobs. Now, it is enabled, and the user may select whether it is checked or unchecked. If it is checked, then the behavior is the same as previous releases. If it is unchecked, then the job is submitted with the exclusive parameter for the job set to false.

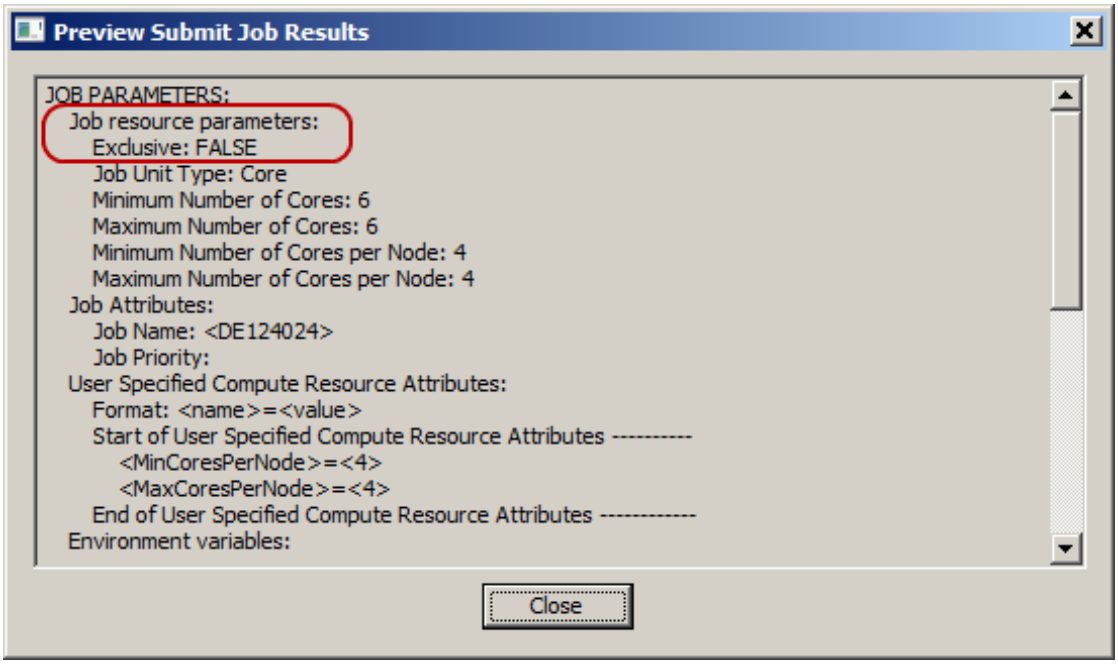
The following screen shot shows this configuration with a Submit Job with Task and Cores specified and nodes as non-exclusive.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The next figure shows Preview for the job submission indicating that the exclusive parameter for the job is false.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

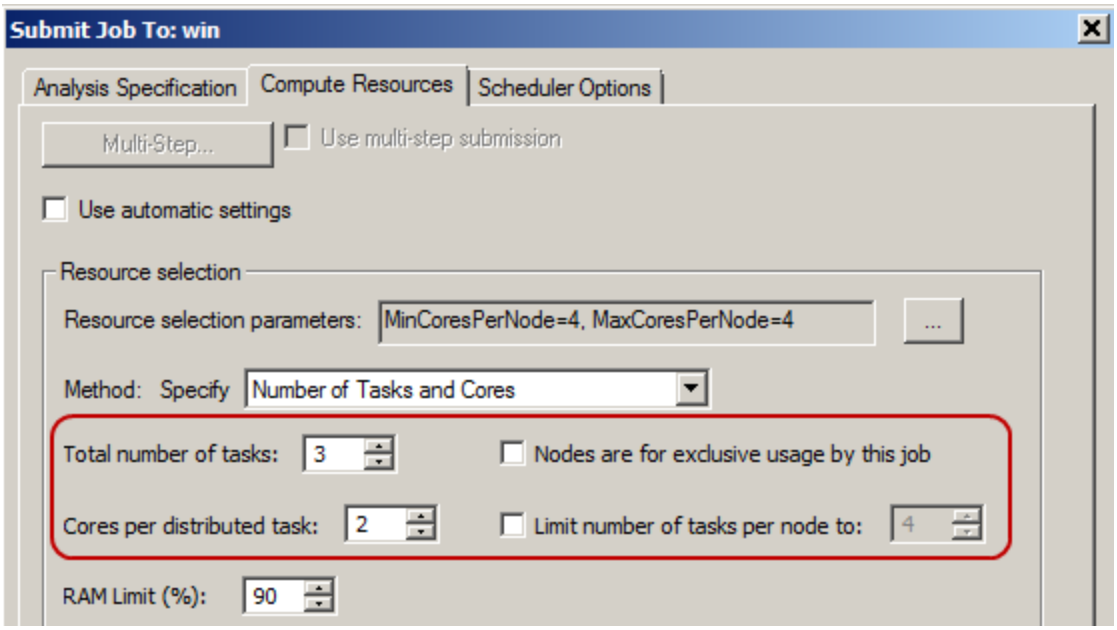


Jobs may only be submitted using this resource selection method with the exclusive parameter set to false if all three of the following conditions are met:

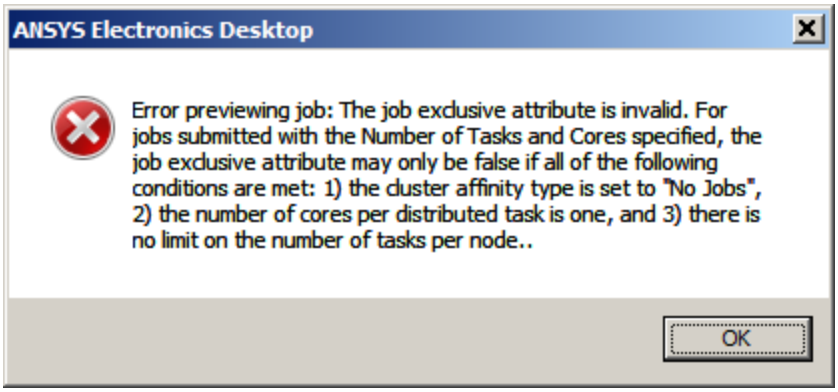
1. The cluster AffinityType is "No Jobs",
2. There is no multiprocessing, and
3. There is no limit on the number of tasks per node.

If any of these requirements are not met, then the job is not submitted or previewed, and an error message appears. The following figure shows this configuration.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



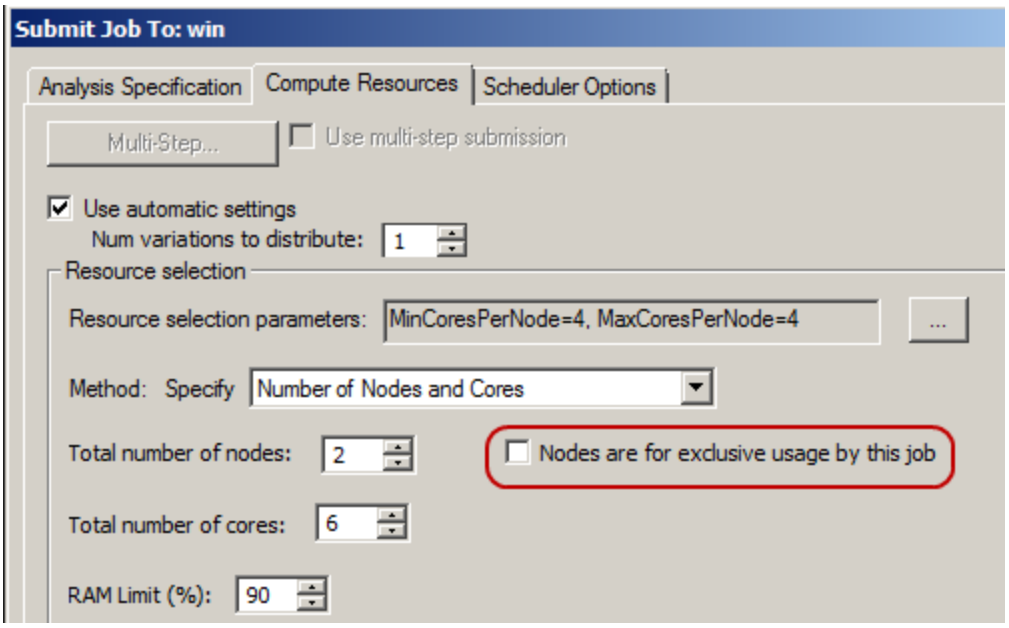
The following figure and screen shot ErrorMsgTasksAndCoresNonExMP.png shows the error message box shown to the user.



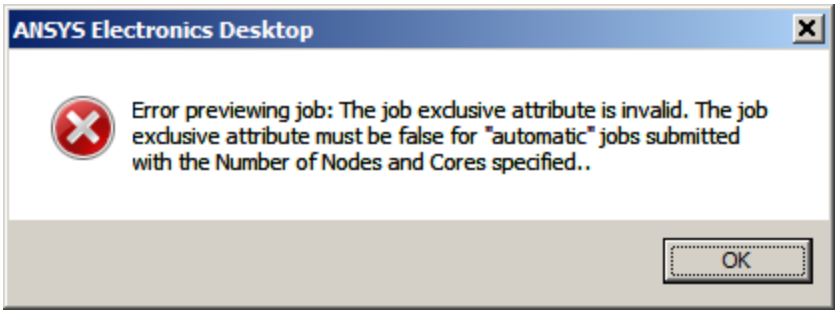
The next example shows submission of a job to a Windows HPC cluster in which the "Use automatic settings" check box is checked, and the resource selection method is "Specify Number of Nodes and Cores". If you select *Use automatic settings* with *Num variations to distribute* set to 1, Optimetrics variations will be solved sequentially. Other distribution types will be distributed automatically. It does distribute frequencies, domains, and use of multiple level domains. If you set Num variations to distribute to 2 or more, Optimetrics variations will be solved in parallel. Other distribution types will be distributed automatically. The exclusive setting for the job is controlled by the "Nodes are for exclusive usage by this job" check box. In previous releases, this check box was always disabled and checked for Windows HPC jobs. Now, it is enabled, and the user may select whether it is checked or unchecked. If it is checked, then the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

behavior is the same as previous releases. If it is unchecked, then the job is not submitted or previewed because only exclusive jobs are allowed when using this resource selection method, and an error message box is shown to the user in this case. The following figure shows this configuration:

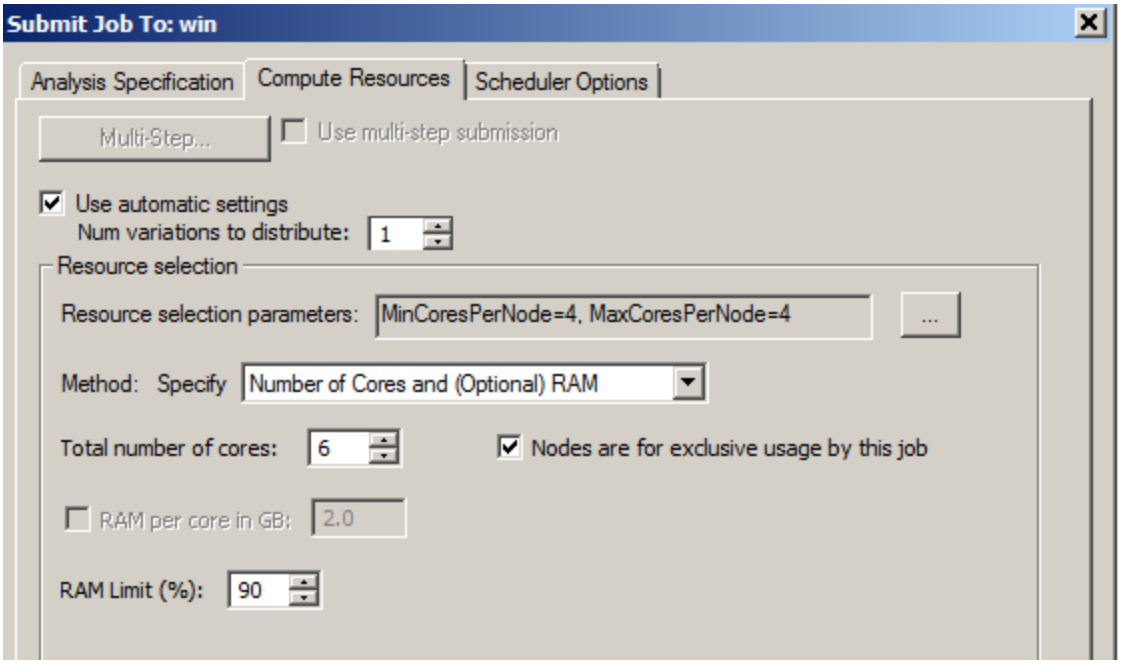


The following figure shows the error message.

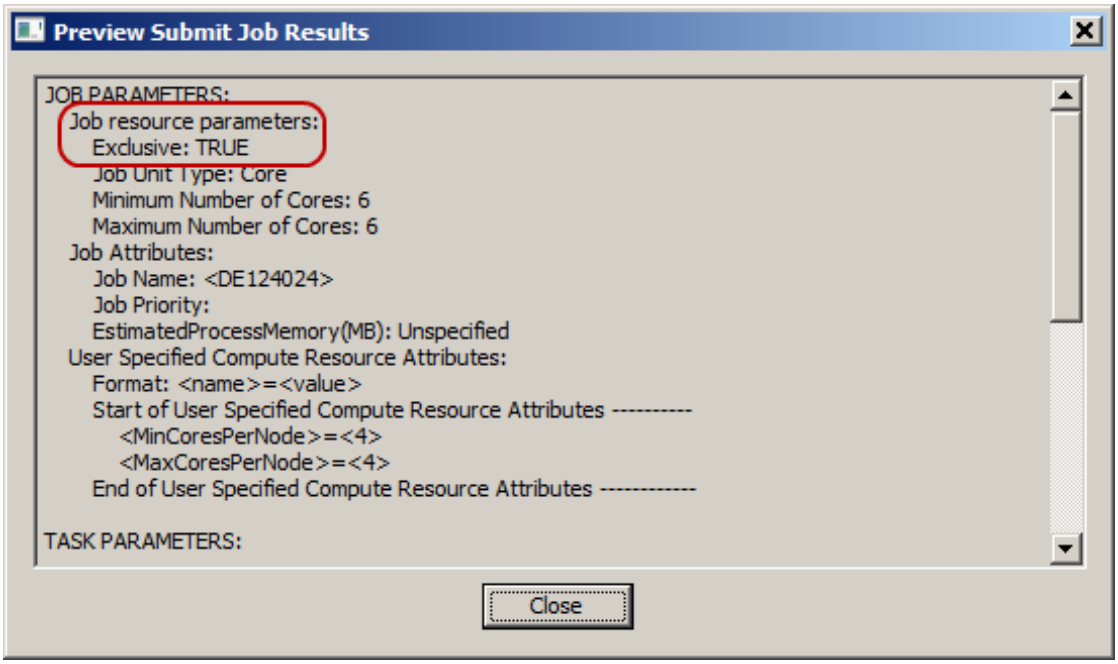


The next example show submission of a job to a Windows HPC cluster in which the "Use automatic settings" check box is checked, and the resource selection method is "Specify Number of Cores and (Optional) RAM". The exclusive setting for the job is controlled by the "Nodes are for exclusive usage by this job" check box. In previous releases, this check box was always disabled and checked for Windows HPC jobs. Now, it is enabled, and you may select whether it is checked or unchecked. If it is checked, then the behavior is the same as previous releases. The following figure shows this configuration.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

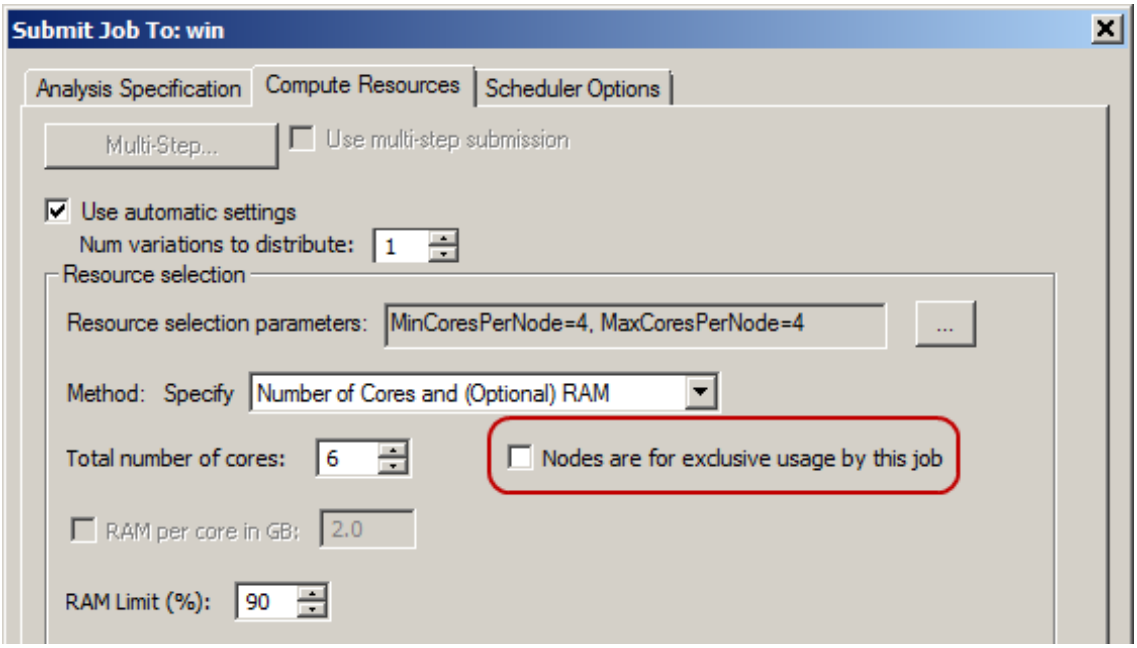


The following figure shows the Job Preview for this case, indicating that the exclusive parameter for the job is true.

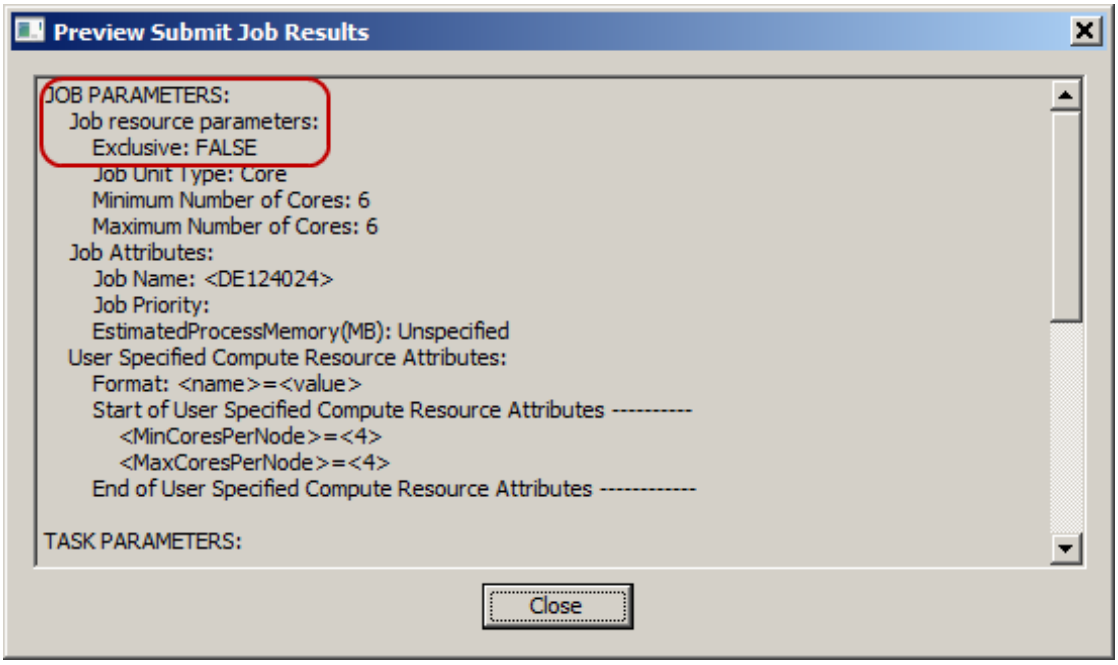


If the exclusive check box is unchecked, then the job is submitted with the exclusive parameter set to false. The following figure shows this configuration.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



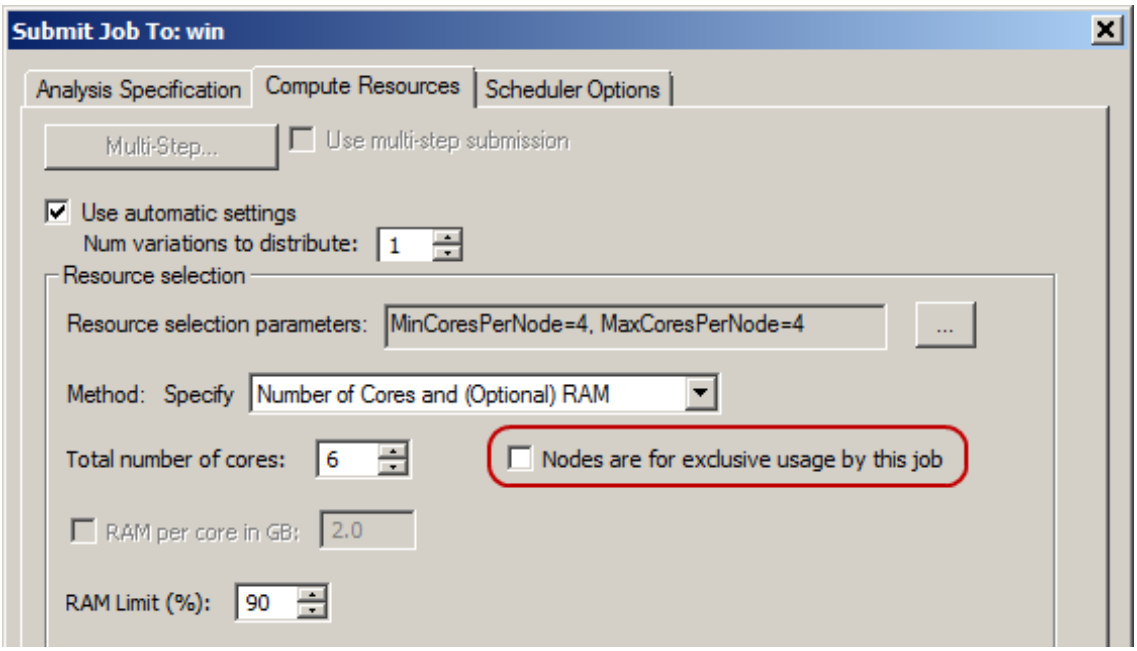
The following figurescreen shot PreviewCoresAndRAMNonEx.png shows the Job Preview for this case, indicating that the exclusive parameter for the job is false.



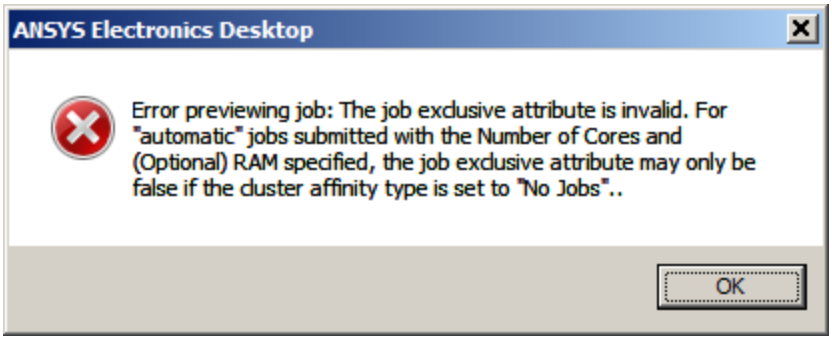
The last example resembles the previous one, except that the Windows HPC cluster AffinityType is set to "Non Exclusive Jobs" in this case. The job is submitted with the "Use

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

automatic settings" check box in the checked state, and the resource selection method set to "Specify Number of Cores and (Optional) RAM". Because the AffinityType is not "No Jobs", only exclusive jobs may be submitted to this cluster. If the exclusive check box is not checked, then the job is not submitted or previewed, and an error message box is displayed to the user. The following figure shows this configuration.



The following figure shows the error message.



Windows® HPC Job Credentials

Normally, a user will be prompted for the credentials used to submit a job. One way to simplify this process is to use the "cluscfg setcreds" command to set the user's credentials in the credentials cache. If this is done, then no password needs to be supplied for a job submitted for

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the specified user. Here is a cluscfg command that may be used to set the user credentials in the credentials cache:

```
cluscfg setcreds /password:* /scheduler:cluster_name
/user:domain\user_name
```

Here:

- cluster_name = the name of the cluster (hostname of the head node)
- domain = optional domain name; if omitted, the following \ should also be omitted
- user = user name

When this form of the command is used, the user is prompted for the password and also asked if the password should be remembered (cached).

See the following web page for more information on the cluscfg setcreds command:

http://technet.microsoft.com/en-us/library/cc947669(WS.10).aspx

Integration with Grid Engine (GE)

The Grid Engine (GE) scheduler (also known as OGE and SGE) is only supported on Linux. With GE, jobs may be submitted in any of the following ways:

- Using GE commands (qsub, etc.) or the SGE gui (qmon)
- Using the generic scheduler GUI in local mode
- Using the generic scheduler GUI in service mode

For additional information on supported schedulers, visit the Platform Support page. Then, locate and click the following link:

Ansys 2025 R2 - Job Schedulers and Queuing Systems Support

Ansys Electromagnetics products support Grid Engine (GE) for Serial analysis, Multi Processing and Distributed Analysis. Models with parametric sweeps can use Large Scale DSO. With GE, the Ansys EM job doesn't require graphics. Ansys EM job's progress can be monitored through GE commands or through Electronics Desktop's Job Management interface.

GE Job Management

You can use Ansys Electronics Desktop to submit batch jobs to GE and monitor those jobs.

This involves the following steps:

1. Use **Tools** > **Job Management** > **Select Scheduler** to select SGE/OGE/GE as the scheduler.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Use **Tools** > **Job Management** > **Submit Job** to submit a batch job to GE.

   GE-specific Settings:

   - On the **Compute Resources** tab, click the **Resource Selection Parameters** ellipses button (**...**) to specify the **ParallelEnvironment** parameter. If you do not specify anything, the scheduler will select a parameter.
   - On the **Compute Resources** tab, select **Specify node list** if you wish to specify the nodes. In a computing environment where the available cores are not uniform, you can use this to have control over which resources your job will use. If your Analysis configuration contains a node list, you can use Populate this Page from Analysis Configuration.
   - Memory resource behavior is dependent on the GE version as well as the particular scheduler settings. The output of the `qconf -sc` command shows all of the complexes available to GE schedulers. For a complex to be considered valid for memory resource selection, it must be of type "MEMORY", have relop "<=", be requestable (could be "forced" as well, instead of "YES"), and be consumable. At least one valid memory complex must be available to submit jobs using the Automatic Cores and RAM method.

     For Univa GE 8.3 and later, there is an additional column for whether resources are available to a preempting job after preemption of a running job. It is up to the cluster administrator to determine the appropriate Available After Pre-Emption (aapre) setting for memory complexes. This setting is ignored for memory complex validation.

     Determining the correct memory complex by default is error-prone. Because the correct choice of memory complex can vary from cluster to cluster, the memory complex selection is now exposed by default under the compute resource selection parameters, allowing the user to make the selection without having to set an environment variable. Only complexes that have been validated (meeting the requirements specified above) can be selected. The cluster administrator (or someone who has knowledge of the specifics of the cluster in use) should be able to determine the correct memory complex to use for Ansys Electronics jobs. To disable exposure of this selection option, the following environment variable can be set to "0": ANSOFT_SGE_ENABLE_MEM_RES_ATTRIB
   - On the **Scheduler Options** tab, you can **Customize Job Submission**. When the **Override job submission** radio button is selected, user-specified options replace most of the job submission options. When the **Additional job submission options** radio button is selected, user-specified options are appended to the bsub command.

   Click **Preview Submission** to view the qsub commands to be used to submit the job.

3. Use **Tools** > **Job Management** > **Monitor Job** to monitor the job's progress.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Installation of Ansys EM Tools on GE

Windows:

Install on every node of cluster

Setup 'temp directory' to a path that is same on all nodes. For example, c:\temp

LINUX:

Install on a single node, on a shared drive.

Setup 'temp directory' to a path that is same on all nodes. For example, /tmp

Ensure that the product is available using the same path on all nodes

Permissions:

All users of the cluster should have read/write permissions to temp directory

All users should have read/execute permissions to installation directory

When a desktop scheduler GUI is run the same node as the job submission node, no other configuration is necessary: installation is sufficient. You select the scheduler through the desktop GUI. You need to ensure that scheduler commands are available in the path before you launch desktop.

> **Note:**
>
> There is no need to install RSM unless the you are using the scheduler GUI on a post processing node that is different from the job submission node. In this case, RSM must be configured with the scheduler type and path.
>
> A post processing nodes is a node in the cluster that can run the Ansys Electromagnetics desktop in graphical mode. A job submission node is a node in the cluster in which job submission commands are available.

Turn OFF firewall between cluster nodes.

Scenario 1: The post-processing node and job-submission node roles are served by distinct machines.

In this case, perform the following configuration:

The job-submission node should be configured to run RSM service, which serves as a proxy to scheduler. The RSM Service should be running as 'root' in order to facilitate jobs running using the credentials of the job's owner. A **configuration file in the RSM installation folder should be edited** to specify information regarding the scheduler that manages jobs on this cluster. A

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

block labeled 'Scheduler' must be included within the 'AnsoftCOMDaemon' block. This block contains two string entries:

- SchedulerName: this contains the unique part of the scheduler proxy library name
- ConfigString: this contains a scheduler specific configuration string

The case of the SchedulerName string is significant on Linux because Linux file names are case sensitive. The case of the SchedulerName string is not significant on Microsoft Windows. In Ansys Electromagnetics Suite 2025 R2, the possible scheduler names are: lsf and sge. The ConfigString entry is a scheduler specific configuration string, described below.

In addition, the AnsoftRSMService must be started with appropriate environment variables set. Generally, the environment variables must be set the same as they would be set for using the scheduler via command lines.

SGE Details

For SGE, the ConfigString entry must contain the search path for the SGE commands. It may contain a single directory, the directory containing the SGE commands. Alternatively, it may be a path, with directories separated by the colon character ":", where the SGE command directory appears before any other directory containing files with the same name as any SGE commands.

Example ansoftrsmservice.cfg configuration file:

```
$begin 'AnsoftCOMDaemon'
$begin 'Managed COM Servers'
$end 'Managed COM Servers'
$begin 'Scheduler'
'SchedulerName'='sge'
'ConfigString'='/opt/sge6.2u4/bin/lx24-amd64'
$end 'Scheduler'
$end 'AnsoftCOMDaemon'
```

Scenario 2: The post-processing node and job-submission node roles are served by the same machine.

The **Select Scheduler...** command is used to gather details about the scheduler. In this case, the Desktop process should be started in an environment suitable for submitting jobs to the scheduler. See: Selecting a Scheduler.

The environment should be configured so that all SGE commands are found using the standard search path. In particular, search for the following commands in the search path should result in the SGE command being found: "qsub", "qdel", "qstat", and "qconf". No other command with the same name should appear before the SGE command in the search path.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

GE Commands for Information about Jobs and Cluster Configuration

The following GE commands are especially useful for getting information about the cluster configuration or for getting information about running or completed jobs. This list only contains a few of the most common commands. Consult Grid Engine help for more information.

qconf -help: The first line displays the SGE version

qacct -j *job-id*: Displays a log of the completed job with the specified job ID (if accounting is enabled)

qstat -j *job-id*: Displays a log of the running job with the specified job ID

qconf -sc: Show all complex attributes

qconf -spl: Show a list of all parallel environments

qconf -sp *pe-name*: Show details of the parallel environment with the specified name

qconf -sql: Show a list of all queues

qconf -sq *queue-name*: Show details of the queue with the specified name

qconf -sconf: Show configurations

Submitting Ansys EM SGE Batch Jobs via the Command Line

The SGE qsub command may be used to submit Ansys EM jobs. Typical command formats are:

```
qsub qsub_args ansysEM_exe ansys_args
qsub qsub_args job_script
qsub qsub_args[ -]
```

where:

- *qsub_args* are the options of the qsub command,
- *ansysEM_exe* is the pathname of the Ansys EM tool executable to launch,
- *ansys_args* are the arguments to the Ansoft tool command, and
- *job_script* is a shell script containing the Ansys Electromagnetics desktop command to run.

In the first format, the Ansys EM desktop command and its arguments are specified on the **qsub** command line. In the second format, the pathname of a shell script containing the Ansys EM desktop command and its arguments is specified on the **qsub** command line. In the third format, the command is omitted or replaced with a hyphen; this indicates that the command or script will be taken from stdin.

Quoting Ansys EM Command or Arguments for SGE

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the Ansys EM tool executable pathname (*ansysEM_exe*) or any of the arguments of the Ansys EM tool command (*ansysEM_args*) contain characters which are interpreted by the command shell, then these special characters must be properly quoted to ensure that the correct command is launched by SGE. This is especially important when using the first form of the **qsub** command, as the Ansys EM desktop command is processed by the shell twice in this case. It is processed by the shell when the **qsub** command is processed, and again when the job is started.

Serial SGE Batch Jobs

In general, Ansys EM batch jobs may be submitted as SGE serial jobs without any special considerations.

See Monitoring Ansys EM SGE Batch Jobs for options for monitoring Ansys EM batch jobs.

Parallel SGE Batch Jobs

When an Ansys EM batch job is run as an SGE parallel job, the SGE scheduler will select the hosts for the distributed analysis job, and start the desktop process on one of these hosts. The desktop process will obtain the list of hosts from the SGE scheduler, and start analysis processes, as needed, using the SGE scheduler facilities. To run an SGE parallel job, the job must be submitted to an SGE parallel environment (PE).

If the qmaster tcp port is not configured as a service, but rather via the environment variable SGE_QMASTER_PORT, this variable must be set in the Ansys EM batch job environment. This is needed because the ANSOFT EM desktop uses the "qrsh -inherit" command to launch engine processes.

See Monitoring Ansys EM SGE Batch Jobs for options for monitoring Ansoft batch jobs.

Setting Up an SGE Parallel Environment (PE)

To allow Ansys EM batch jobs to distribute analysis engines to multiple hosts, the job must be run in a parallel environment (PE) in which the control_slaves parameter is set to TRUE. This setting is required to allow the Ansys EM desktop to start analysis engines on hosts other than the local host.

Here is a sample parallel environment configuration:

```
pe_name ans_test1
slots 999
user_lists NONE
xuser_lists NONE
start_proc_args /bin/true
stop_proc_args /bin/true
allocation_rule $round_robin
control_slaves TRUE
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
job_is_first_task FALSE
urgency_slots min
accounting_summary TRUE
```

The user_lists and xuser_lists parameters are ACLs (access control lists) used to control which users have permission to use the parallel environment. The user_lists setting gives permission to use the PE. The xuser_lists setting denies permission to use the parallel environment. The xuser_lists settings override the user_lists settings.

The start_proc_args and stop_proc_args parameters contain the pathname and arguments for the parallel environment startup and shutdown scripts. No startup or shutdown scripts are needed for parallel Ansys Electromagnetics batch jobs. The setting /bin/true may be used as the value for these scripts; this utility does nothing and returns an exit code indicating success (0).

The parallel environment allocation_rule parameter will affect how the analysis engine tasks are distributed across the hosts allocated to the job. The $round_robin setting distributes the tasks across the hosts in a round robin fashion, resulting in the load being relatively evenly distributed over all of the hosts. The $fill_up setting allocates all slots on a host before distributing the tasks to another host; the result is that most hosts are either fully utilized or completely unused. See the sge_pe man page for other settings for this parameter.

The control_slaves parameter must be set to TRUE, as described above.

The job_is_first_task parameter also affects how tasks are allocated. When submitting a job to run in a parallel environment, the number of parallel tasks, n, is specified on the command line. If this setting is TRUE, then the job process is considered one of the tasks, and only (n-1) additional tasks are allocated to the job. If the setting is FALSE, then the job process is not considered to be one of the tasks, and n additional tasks are allocated for the job.

See the sge_pe man page for more information about these and other PE parameters.

A parallel environment does not run tasks directly. Instead, the tasks are distributed to queues associated with the parallel environment. In order to complete the setup of a parallel environment, one or more queues need to be associated with the parallel environment. The queue pe_list parameter is used to specify the parallel environments (PEs) supported by the queue. This is an important step; **if no queues support a given PE, then jobs submitted to that PE will not run**.

Parallel Batch Job Command Line Considerations

The number of engines run on a host will depend on the total number of distributed engines, and the number of hosts allocated to the job. The memory required on a host depends on the number of engines running on the host and on the memory needed for each engine. The **qsub** command **-l resource=** *value*,... or **-q** *queue_list* > command line options specify that the parallel batch job run on machines with sufficient memory and other resources.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Monitoring GE Serial and Parallel Batch Jobs

You can monitor jobs through the Electronics Desktop user interface, or through the command line.

The suggestions below are for GE serial jobs and GE parallel jobs.

GE qstat Command

The SGE **qstat** command may be used to display information on jobs and queues. If the **-j** option is included, then information on jobs is displayed. If the option includes a job list (**-j [** *job_list* **]**), then the displayed information is limited to the jobs in the job list.

The **-u***user*,... option limits the output to jobs associated with users in the user list. If the **-u***user*,... option is not specified, then information on queues or jobs of the current user are displayed.

The **-t** option displays extended information about the subtasks of each displayed job. This is equivalent to the **-g t** option. The **-r** option displays extended information about the resource requirements of the displayed jobs.

See the SGE manual pages for more information.

Ansys EM Desktop -monitor Command Line Option for SGE

The **-monitor** command line option enables batch job output to the standard output and standard error streams. The warning, info, and progress messages are sent to the standard output stream. The error and fatal messages are sent to the standard error stream.

The SGE scheduler redirects the standard output and standard error streams of batch jobs to files specified in the **qsub -o [[** *hostname* **]:]** *path*,... and the **-e [[** *hostname* **]:]** *path*,... command line options, respectively. If either option is not specified, then the associated stream is redirected to the default file pathname.

The **qsub -j y[es] | n[o]** controls whether the standard output and standard error streams are merged. If the y or yes value is specified, then the standard error stream is merged into the standard output stream. If the **-e** *host_and_path* option is also specified in this case, the *host_and_path* setting is ignored. If the n or no value is specified, or if this option is not specified, then the standard error stream and standard output stream are not merged.

You can monitor the progress of a job by checking the standard output file for progress, info and warning messages, and checking the standard error file for error and fatal messages.

Example SGE qsub Command Lines

All of the following examples show how to submit Linux HFSS jobs on SGE, but similar command lines will work for all Ansys Electromagnetics products.

Serial job using command line:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
qsub -b y /opt/ansys_inc/v252/AnsysEM/ansysedt -ng -BatchSolve
 ~/projects/OptimTee.aedt
```

- The **-b y** option indicates that hfss is launched directly from the command line, instead of using a script.
- No queue is specified, so the default queue will be used.

Serial job with a hard runtime limit of 15 minutes:

```
qsub -b y -l h_rt=00:15:00 /opt/ansys_inc/v252/AnsysEM/ansysedt
 -ng -BatchSolve ~/projects/OptimTee.aedt
```

- The -l h_rt=00:15:00 option indicates that this job has a "hard" runtime limit of 15 minutes.

Serial job using a script, with a runtime limit specified in the script:

```
qsub ~/sge/scripts/OptimTee.csh
```

- The **-b y** option is absent, so the script ~/sge/scripts/OptimTee.csh will be run when the job starts.
- The script file OptimTee.csh may contain SGE directives in addition to the command(s) to run. In this example, a directive with a hard runtime limit if 15 minutes is included in the script.

Script file contents:

```
#!/bin/csh
#$ -l h_rt=00:15:00
/opt/ansys_inc/v252/AnsysEM/ansysedt -ng -BatchSolve
 ~/projects/OptimTee.aedt
```

- The SGE directive #$ -l h_rt=00:15:00 is equivalent to including -l h_rt=00:15:00 on the qsub command line.

Distributed processing job using 4 engines:

```
qsub -b y -pe pe1 4 /opt/ansys_inc/v252/AnsysEM/ansysedt
 -ng -BatchSolve -Distributed -machinelist num=4
~/projects/OptimTee.aedt
```

- The **-b y** option indicates that hfss is launched directly from the command line, instead of using a script.
- The **-pe pe1 4** command_line option indicates that this is a parallel job running under the pe1 parallel environment, and that 4 cores or processors are allocated to this parallel job.
- The "-machinelist num=n" option is now required for batch jobs.
- The **-Distributed** option indicates that this is a DSO job, so that multiple engines will be started. Because 4 cores are allocated to the job, the job will run 4 engines. The -Distributed option may now have additional options, such as includetypes=xxx, excludetypes=xxx, maxlevels=n, and numlevel1=n, where n indicates and integer, and xxx indicates a list of distribution types or "default".

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Recommended Practices for GE Clusters

The following subsections contain recommendations on how to set up an GE cluster for efficiently running Ansys Electromagnetics Suite serial and parallel jobs. These recommendations require the cluster administrator to make configuration changes.

Submitting Exclusive Jobs

Consumable Memory Limits

Serial Jobs in SGE

Parallel Jobs in SGE

Using Multithreading with Parallel Jobs

Submitting Exclusive Jobs

In many cases, clusters are used to run "large" Ansys Electromagnetics Suite batch jobs. That is, these are jobs that may require a large quantity of resources, such as processors, memory, disk space, or run time. One way to ensure that the resources needed by the batch job are available to the job is to run the job in an "exclusive" mode. That is, any host running the job is not available for use by any other jobs. There is no GE built in mechanism for specifying that a job is "exclusive". GE is extensible, and it is not difficult to configure the cluster to allow exclusive jobs. The steps below show one way to do this. This example requires GE 6.2u3 or later. Note that specifying a job as "exclusive" may delay the start of the job if there are not enough hosts available to run the job exclusively.

1. Use the command qconf -mc to add a new complex to the table of complexes. Recommended attributes are:
    - name : exclusive
    - shortcut : excl
    - type : BOOL
    - relop : EXCL
    - requestable : YES
    - consumable : YES
    - default : 0
    - urgency : 0
2. Set the value of "exclusive" to TRUE for each execution host using the command qconf -me hostname, where hostname is the name of the host. The values of all host configuration parameters may be displayed using the command qconf -se hostname. The "complex_values" line should look similar to:

    complex_values exclusive=TRUE, but other values may also be included.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. When submitting a job, the job will be "exclusive" if the value "excl" is included in the resource list specified by the qsub -l option. If the resource list does not include "excl" then the job will not be exclusive, and other jobs may run on the same host or hosts as this job.
4. Example qsub command line for exclusive serial job:

   ```
   qsub -b y -l excl /opt/ansys_inc/v252/AnsysEM/ansysedt -ng -
   BatchSolve -machinelist num=1 ~/projects/OptimTee.aedt.
   ```

   Although serial jobs use only one slot, no other jobs will run on the host where this job is running, even if additional slots are present.

5. Example qsub command line for exclusive parallel job using eight engines, each using a single thread of execution:

   ```
   qsub -b y -l excl -pe pe1 8 /opt/ansys_inc/v252/AnsysEM/ansysedt
   -ng -BatchSolve -Distributed -machinelist num=8
   ~/projects/OptimTee.aedt
   ```

   None of the hosts used for this job will be allowed to run other jobs while this job is running.

Consumable Memory Limits

GE contains several built-in complexes related to memory, including mem_total, for example, but none of these are "consumable". If a job is submitted with resource list including one of these non-consumable memory complexes (such as mem_total), then the job will run on a host or hosts only if sufficient memory is available. If a second job is submitted, the memory request for the second job is compared to the original total when determining if the job may run on a host. This may result in both jobs running out of memory. For example, if host A has mem_total=16G of memory, and two jobs are submitting with option "-l mt=16G", then both jobs could run on host A, if sufficient slots are available on host A.

GE allows complexes to be "consumable" to avoid this type of problem. If a complex is consumable and a job requests x amount of the complex in the -l resource list, then the available amount of the resource is decreased by x for subsequent jobs. For the same example as above, if the mem_total complex was consumable, then the first job would run on host A. This would decrease the available mem_total from 16G to 16G-16G = 0. The second job could not run on host A because there is no memory available for this job.

We do not recommend changing the behavior of the built-in complexes (such as mem_total) because other scripts may expect normal behavior of the built-in complexes.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Recent versions of UGE (Univa Grid Engine) come with "m_mem_free" and "mem_free" complexes already configured, and if so then there is no more configuration required. You can just use mem_free when per-host memory request is desired, and m_mem_free when per-core (per-slot really) memory request is desired. SGE may already have "mem_free" which can be used for per-host memory request.

Below shows how to configure the mem_free consumable resource.

Recommended attributes are:

- name : mem_free
- shortcut : mf
- type : MEMORY
- relop : <=
- requestable : YES
- consumable : YES
- default : 0
- urgency : 0

Note:

Ansys Electronics Desktop has the capability in auto cores and RAM to automatically select the memory complex (if one is available) and create this command line option for the user, if you check the "Use RAM constraint" and enter a non-zero amount of GB to use per core.

Serial Jobs in GE

If a serial job is submitted with the option -l phys_mem=mem_neededincluded, then the job may only run on a host in which the remaining physical_memory is equal to or greater than the mem_neededvalue.

Example 1: Host A has physical_memory=16G, and host B has physical_memory=8G. If mem_neededis 8G, the job may run on either host A or host B. If mem_neededis 16G, then the job may only run on host A.

Example 2: Host A has physical_memory=16G, and host B has physical_memory=8G. Job 1 is already running on host A, and it was submitted with option -l phys_mem=8G. If job 2 is submitted with option -l phys_mem=16G, then job 2 cannot start until job 1 finishes, because only host A has 16GB of physical_memory. If job 2 is submitted with option -l phys_mem=8G,

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

then job 2 may start immediately, and run on either host A or host B, because both hosts have 8G of physical_memory remaining.

Parallel Jobs in GE

Because the consumable setting for physical_memory is YES (and not JOB), each slot of the job requires a physical_memory of mem_needed. The number of slots on a host assigned to the job is limited by the number of available slots on the host. It is also limited by the physical_memory available on the host; the number of slots assigned to the job cannot exceed the available physical_memory on the host divided by the mem_neededspecification.

Example 1: Execution host A and execution host B both have 4 slots per host (configured in the queue associated with the parallel environment). Host A has physical_memory=16G and host B has physical_memory=8G (shown by commands qconf -se Aand qconf -se B). If a job is submitted that requires 6 slots and 4G per slot, it will be able to run, with 4 slots on host A and 2 slots on host B. The qsub command might look like: qsub -l phys_mem=4G -pe pe_name 6 command args

Example 2: Same as example 1, except that 7 slots are requested. In this case, the job will never run. Although there are 8 slots available on hosts A and B, only two of the slots on host B are usable by this job because it only has physical_memory of 8G. With only 6 slots total available to this job (4 on host A and 2 on host B), the job can not start. In this case the command might look like: qsub -l phys_mem=4G -pe pe_name 7 command args

Using Multithreading with Parallel Jobs

For large jobs it may be useful to combine multiprocessing with distributed processing. Distributed processing refers to starting multiple processes, in which each process performs a portion of the analysis. These processes may run on the same host or on different hosts. The number of processes running at the same time is known as the number of "analysis engines". Multiprocessing refers to using multiple threads within a single process to decrease the run time of the process. Multiprocessing may also be called multi-threaded processing.

As a concrete example of combining multiprocessing with distributed processing, an analysis could run with four engines, where each engine uses two threads. In order to distribute the processing load so that no processor is overloaded, one slot is generally allocated per thread, so 8 slots would be needed for this example (4 engines * 2 threads per engine = 8 threads). The four engines could all run on a single host, or they could be distributed across 2, 3 or 4 hosts, depending on available slots. Each engine represents a single process, so the two slots for each engine must be allocated on the same host.

This section describes how to set up a GE cluster so that a specified number of slots per host may be requested when a job is submitted. This procedure will require the cluster administrator privileges. This capability may be used to submit parallel jobs in which one engine runs on each host, and the number of slots per host matches the number of threads used by each engine.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Let n be the largest number of slots available on any host used for the jobs. Create a separate parallel environment for each value of the number of slots per host from 1 to n. For example, pe_sph1 is a parallel environment in which one slot is allocated to the job per host, and pe_sph2 is a parallel environment in which two slots are allocated to the job per host, etc. The command qconf -ap pe_name may be used to create each new parallel environment. The allocation_rule parameter should be set to the number of slots per host, an integer from 1 to n. The control_slaves parameter should be set to TRUE, as described above. The slots parameter should be set to the maximum number of slots managed by this parallel_environment, which is typically set to a large number, such as 999. The other parameters should be set to values appropriate for the cluster. For example, the pe_sph2 parallel environment might have the following parameters:
    - pe_name : pe_sph2
    - slots : 999
    - user_lists : NONE
    - xuser_lists : NONE
    - start_proc_args : /bin/true
    - stop_proc_args : /bin/true
    - allocation_rule : 2
    - control_slaves : TRUE
    - job_is_first_task : FALSE
    - urgency_slots : min
    - accounting_summary : TRUE
2. When submitting a job, use the parallel environment where the slots per host matches the number of threads per engine.

The batchoptions setting -machinelist num=n is required. This should be set to match the number of slots per host. With any analysis, a portion of the analysis may not be distributed across multiple engines.

Example qsub command line for running distributed processing with four engines and multiprocessing with two threads per engine:

```
qsub -V -b y -pe pe_sph2 8 /opt/ansys_inc/v252/AnsysEM/ansysedt -ng -
BatchSolve -Distributed -machinelist num=4 -batchoptions
"projects/OptimTee.aedt"
```

The -Voption indicates that the all environment variables in the submission environment should be copied to the job environment.

- The -b y option indicates that hfss is launched directly from the command line, instead of using a script.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The -pe sph2 8 command_line option indicates that this is a parallel job running under the pe_sph2parallel environment so that two slots are allocated to this job from each host, and that 8 slots in total are allocated to this parallel job.
- The -Distributed option indicates that this is a DSO job, so that multiple engines will be started. The -Distributed option may now have additional options, such as includetypes=xxx, excludetypes=xxx, maxlevels=n, and numlevel1=n, where n indicates and integer, and xxx indicates a list of distribution types or "default".
- The -machinelist num=4option indicates that a total of four engines will be started.
- The entire hfss command is in double quotes, and the double quotes enclosing the -batchoptions value are escaped. Each of these double quotes is replaced by the sequence "\"".

Issue with qrsh (SGE)

Ansys EM parallel batch jobs use the SGE **qrsh** command to launch engine processes on remote hosts. If the **qrsh** command is not working correctly, then the parallel job is unable to launch engine processes on remote hosts. If this problem occurs, the batch log for the job typically includes one or more error messages indicating that a COM engine was unable to be started on a remote host. If this occurs, the user or cluster administrator should verify that the SGE qrsh command is working correctly, and correct the problem if the SGE **qrsh** command is not working correctly.

The **qrsh** command may be tested by running a simple command on a specified host, such as **qrsh -l hostname=host1 hostname** or **qrsh -l hostname=host1 ls /tmp**, where host1 is the remote host name. The first test should simply echo back the hostname of the remote machine. The second test should list the contents of the /tmp directory on the remote machine.

The failures of the SGE qrsh command are associated with the following global sge configuration parameters, listed below with values that may cause the failures:

```
qrsh_command /usr/bin/ssh -t
rsh_command /usr/bin/ssh -t
rlogin_command /usr/bin/ssh -t
```

If these parameter settings are removed, then the SGE built-in mechanisms are used for qrsh, rsh, and rlogin. No problems with the built-in versions have been reported. The SGE qconf -sconf global command may be used to view these parameter settings. The SGE qconf -mconf global command may be used to modify or remove these parameter settings.

Issue with MainWin Core Services for SGE

By default, SGE creates a temporary directory for each SGE batch job, and deletes this temporary directory and its contents when the job finishes. SGE sets the TMP and TMPDIR environment variables of the job environment to point to this temporary directory. Ansys EM desktop software starts the MainWin Core Services on startup, if they are not already running.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After the Ansys EM desktop software finishes, the MainWin Core Services time out and automatically shut down. The MainWin Core Services use the TMP and/or TMPDIR directories to store temporary data. If this temporary data is removed before the services shut down, then the services do not shut down automatically. Normally, SGE will remove the temporary directory and its contents before the services time out. The result is that these extraneous service processes run forever. If this problem occurs, each Ansoft batch job starts an additional set of these services that never shut down. This can result in an excessive number of processes running on the host where the Ansys EM desktop is started. The names of the service processes are:

- watchdog
- regss
- mwrpcss

Workaround for Issue with MainWin Core Services

One way to avoid this problem is to modify the environment in which the Ansys EM desktop runs so that the TMP and TMPDIR environment variables do not point to the directory which will be immediately removed by SGE when the job finishes. This can be done by copying the value of the TMPDIR environment variable to the ANS_SGE_TMPDIR environment variable, and unsetting the TMPDIR and TMP environment variables. The services ignore the ANS_SGE_ TMPDIR environment variable, but if this variable is set, then it will be used as the temporary directory for the rest of the Ansys EM software.

Here is an example bash wrapper script that may be used to work around this issue. In this example, the product is hfss, but the same approach will work for any Ansys EM product. In this example, the script is named sge_hfss and is in the AnsysEM software installation directory. When an Ansys Electromagnetics desktop job is submitted to the SGE scheduler, the script (sge_hfss, in this example) should be submitted instead of hfss. The script will modify the environment, as needed, then start hfss. When the analysis finishes, the script returns the exit status of hfss.

An alternative is to place the script in an arbitrary directory, and modify the script to include an absolute path to the product (hfss in this example).

Script contents:

```
#! /bin/bash

# This script will not correctly process arguments containing
# spaces or other characters special to the shell.

# Create hfss command line
# In this example, sge_hfss and hfss are in the same directory
# An alternative is to use an absolute path for the hfss command
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
cmd0=$0
cmd="${cmd0/%sge_hfss/hfss} $@"

# Fix environment variables
export ANS_SGE_TMPDIR=${TMPDIR}
unset TMPDIR
unset TMP

# Run the hfss command and return the exit status
${cmd}
exit $?
```

Integration with Platform's Load Sharing Facility (LSF)

The Load Sharing Facility (LSF) scheduler is only supported on Linux. Jobs may be submitted in any of the following ways:

- Job Submission GUI
- Using LSF commands (qsub, etc.)

For additional information on supported schedulers, visit the Platform Support page. Then, locate and click the following link:

Ansys 2025 R2 - Job Schedulers and Queuing Systems Support

> **Note:**
>
> If a temp directory is setup by the LSF cluster administrator, analysis engines use this temp directory, overriding the setting in the Ansys EM product.

LSF Job Management

You can use Ansys Electronics Desktop to submit batch jobs to LSF and monitor those jobs.

This involves the following steps:

1. Use **Tools** > **Job Management** > **Select Scheduler** to select LSF as the scheduler.
2. Use **Tools** > **Job Management** > **Submit Job** to submit a batch job to LSF.

   LSF-specific Settings:

   - On the **Compute Resources** tab, click the **Resource Selection Parameters** ellipses button (**...**) to specify the following:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Queue** – a drop-down menu lets you select Normal, chkpn_rerun_queue, idle, license, night, normal_allow_excl, owners, priorityuy, or short.
- **MinCoresPerNode** – the minimum number of cores allowed on a node to be eligible for selection; translates to bsub -R select[ncpus>=N]
- **MaxCoresPerNode** – the maximum number of cores allowed on a node to be eligible for selection; translates to bsub -R select[ncpus<=N]
- **MinMemoryPerNode** – the minimum amount of physical memory (specified in integer GigaBytes) allowed on a node to be eligible for selection; translates to bsub -R select[maxmem>=M]
- **MaxMemoryPerNode** – the maximum amount of physical memory (specified in integer GigaBytes) allowed on a node to be eligible for selection; translates to bsub -R select[maxmem<=M]

If you do not specify parameters, the scheduler does so.

- For LSF, the only non-automatic method of **Resource Selection** is **Number of Tasks and Cores**. You can specify the number of tasks, whether they are for exclusive use by the job, cores per distributed task, a limit number of tasks per node, and RAM limit as percentand a limit number of tasks per node.

3. Use **Tools** > **Job Management** > **Monitor Job** to monitor the job's progress.

LSF Job Submission Guidelines

Before submitting an LSF job, ensure the following are true:

- The project is available in a shared drive that is accessible to all machines in the cluster.
- The project is available using the same path on all machines of cluster.
- There is sufficient space in the project directory and temp directories.
- There is sufficient memory per engine.
- The number of compute resources (Distributed Analysis machines and Multi Processing cores) will achieve the desired scale factor and effective resource utilization.

Integration of Ansys EM Products with LSF

With LSF you do not need to set 'Distributed Analysis Machines' or 'Remote Machine' options. Instead, you submit a job to LSF, requesting the appropriate resources for the job (number of processors, memory per processor, etc.).

For example:

```
bsub -n 1 ansysedt.exe -Batchsolve -ng -local -machinelist num=1
OptimTee.aedt
bsub -n 4 ansysedt.exe -Batchsolve -ng -Distributed -machinelist
num=4 OptimTee.adsn
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The job is queued by LSF until the requested resources are available. Upon resource availability, LSF starts Electronics Desktop with the specified command line on one of the allocated machines. During analysis, Electronics Desktop dynamically obtains the allocated 'Distributed Analysis Machines' from LSF. Electronics Desktop interfaces with LSF to launch engines on remote machines without going through Ansys RSM.

Installing Ansys EM Tools on LSF Cluster

The LSF scheduler is supported on Linux only.

lsf.conf should contain this line:

```
LSF_UNIT_FOR_LIMITS=MB
```

The administrator should have this line in the lsb.params file to ensure that memory reservations are per-slot (per-core):

```
RESOURCE_RESERVE_PER_SLOT=Y
```

Jobs may be submitted in any of the following ways:

- Using LSF commands (bsub, etc.)
- Using the generic scheduler GUI in local mode
- Using the generic scheduler GUI in service mode

For additional information on supported schedulers, visit the Platform Support page. Then, locate and click the following link:

Ansys 2025 R2 - Job Schedulers and Queuing Systems Support

Setup:

1. Install on a single node, on a shared drive.
2. Setup 'temp directory' to a path that is same on all nodes (for example, /tmp).
3. Ensure that the product is available using the same path on all nodes.

Permissions:

- All users of the cluster should have read/write permissions to temp directory.
- All users should have read/execute permissions in the installation directory.
- Turn OFF any firewall between cluster nodes.

When a desktop scheduler GUI is run the same node as the job submission node, no other configuration is necessary; installation is sufficient. Select the scheduler through the Electronics Desktop GUI. Ensure that scheduler commands are available in the path before you launch Electronics Desktop.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> There is no need to install RSM unless you are using the scheduler GUI on a post-processing node that is different from the job submission node. In this case, RSM must be configured with the scheduler type and path.
>
> A post-processing node is a node in the cluster that can run Electronics Desktop in graphical mode. A job submission node is a node in the cluster in which job submission commands are available.

Per-slot Resource Reservation

Set the cluster for per-slot resource allocation if the automatic cores and RAM resource selection method is to be used. You can check the cluster to see if per-slot resource allocation is configured by using the "bparams -a" command. Search the output for "RESOURCE_ RESERVE_PER_SLOT" to determine the setting (either "Y" or "N"). If set to "N" then consult the LSF administration guide on how to change this to "Y".

Scenario 1: The post-processing node and job-submission node roles are served by distinct machines.

In this case, perform the following configuration:

The job-submission node should be configured to run RSM service, which serves as a proxy to scheduler. The RSM Service should be running as 'root' in order to facilitate jobs running using the credentials of the job's owner. A **configuration file in the RSM installation folder should be edited** to specify information regarding the scheduler that manages jobs on this cluster. A block labeled 'Scheduler' must be included within the 'AnsoftCOMDaemon' block. This block contains two string entries:

- SchedulerName: this contains the unique part of the scheduler proxy library name
- ConfigString: this contains a scheduler specific configuration string

The case of the SchedulerName string is significant on Linux because Linux file names are case sensitive. In Ansys Electromagnetics Suite 2025 R2, possible scheduler names are: lsf and sge. The ConfigString entry is a scheduler specific configuration string, described below.

In addition, the AnsoftRSMService must be started with appropriate environment variables set. Generally, the environment variables must be set the same as they would be set for using the scheduler via command lines.

LSF Details

For the LSF scheduler proxy library, the ConfigString entry in the ansoftrsmservice.cfg configuration file is ignored. It may be empty or omitted entirely.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The AnsoftRSMService must be started with the environment set as it would be set for submitting jobs to the LSF cluster.

- For Linux, the cshrc.lsf or the profile.lsf file may be sourced to set up the environment, depending on the shell.

Example ansoftrsmservice.cfg configuration file:

```
$begin 'AnsoftCOMDaemon'
$begin 'Managed COM Servers'
$end 'Managed COM Servers'
$begin 'Scheduler'
'SchedulerName'='lsf'
'ConfigString'=''
$end 'Scheduler'
$end 'AnsoftCOMDaemon'
```

Scenario 2: The post-processing node and job-submission node roles are served by the same machine.

The **Select Scheduler...** command (as described in the Job Management User Interface for LSF section) is used to gather details about the scheduler. In this case, the Desktop process should be started in an environment suitable for submitting jobs to the scheduler. See below for details.

The environment should be configured so that the following LSF environment variables are set appropriately for the LSF cluster in use: LSF_BINDIR, LSF_SERVERDIR, LSF_LIBDIR, and LSF_ENVDIR. In addition, the following LSF commands should be found in the LSF_BINDIR directory: "bsub", "bjobs", "bkill", "lsid", "lsrun", "lshosts", "bmgroup", "bparams" and "bqueues".

Using the bsub Command to Submit Batch Jobs

The LSF bsub command may be used to submit jobs. The typical command format is:

`bsub` *`bsub_args ansys_exe ansys_args`*

where:

- *bsub_args* are the options of the bsub command,
- *ansysEM_exe* is the pathname of the Ansys Electromagnetics desktop executable to launch, and
- *ansys_args* are the arguments to the Ansys Electromagnetics desktop executable.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

bsub Arguments

The **bsub** command has a large number of options that may be used to control the submission process. Only a few options that are often used with Ansys Electromagnetics jobs are listed here. The following options may be used to submit serial or parallel LSF jobs.

-n*min_proc*, *max_proc* or **-n***min_proc*

Submits a parallel job, specifying the number of processors (or slots) required for the job. Here, *min_proc* is the minimum number of processors, and *max_proc* is the maximum number of processors. If no maximum is specified, then exactly *min_proc* processors are requested. If PARALLEL_SCHED_BY_SLOT=Y in lsb.params, this option specifies the number of slots required to run the job, not the number of processors. If the **-n** command line option is not specified, then the job is submitted as a serial batch job.

-R "span[ptile=n]"

There are many ways to use the -R "res_req" option to the bsub command. We only cover -R "span[ptile=n]" here, because this option is very useful for Ansys Electromagnetics jobs. When this option is specified, the LSF scheduler will allocate n processors (or slots) on each host to this job, even if more processors are available on the host.

-x

All hosts running this job operate in exclusive execution mode. The job will only run on a host having no other jobs running on that host. No other batch jobs will be started on a host while this job is running on that host.

See LSF documentation for a complete list of options for the **bsub** command.

> **Important:**
>
> If the Ansys EM tool executable pathname (*ansys_exe*) or any of the arguments of the Ansys tool command (*anssys_args*) contain characters which are interpreted by the command shell, these special characters must be properly quoted to ensure that the correct command is launched by LSF. A similar problem may occur if any of the *ansoft_args* require single quote, double quote or space characters. Note that the Ansys Electronics Desktop command is processed by the shell twice: once when the **bsub** command is processed, and again when the job starts. See: Example Command Lines.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example LSF bsub Command Lines (Linux Only)

> **Note:**
>
> The following examples use HFSS as the product, but similar command lines will work for all Ansys EM products.

Serial Job

```
bsub -n 1 "C:\Program Files\ANSYS Inc\v252\AnsysEM\ansysedt" -ng
-BatchSolve -machinelist num=4 ~/projects/OptimTee.aedt
```

The -n 1 option indicates that this job runs on one core.

Serial Job Requiring a Minimum of 4GB

```
bsub -n 1 -R "select[mem>4000]"
"C:\Program Files\ANSYS Inc\v252\AnsysEM\ansysedt" -ng
-BatchSolve -machinelist num=4 ~/projects/OptimTee.aedt
```

The -R "select[mem>4000]" option indicates that this needs a minimum of 4 GB memory.

Multi-processing Job using 4 Cores

```
bsub -n 4 -R "span[ptile=4]"
"C:\Program Files\ANSYS Inc\v252\AnsysEM\ansysedt" -ng -BatchSolve
-batchoptions -machinelist num=4 ~/projects/OptimTee.aedt"
```

- The -R "span[ptile=4]" option indicates that the four cores need to be on the same machine.
- The -batchoptions option indicates that HFSS should use four cores for multi-processing.
- The entire hfss command is in double quotes, and the double quotes enclosing the -batchoptions value are escaped. Each of these double quotes is replaced by the sequence "\"".

Distributed Processing Job using 4 Engines

```
bsub -n 4 "C:\Program Files\ANSYS Inc\v252\AnsysEM\ansysedt" -ng -
BatchSolve -Distributed ~/projects/OptimTee.aedt
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The -n 4 option indicates that the four cores are needed for the job.
- The -Distributed option indicates that this is a DSO job, so that multiple engines will be started. Because 4 cores are allocated to the job, the job will run 4 engines. The -Distributed option can have additional options, such as includetypes=xxx, excludetyeps=xxx, maxlevels=n, and numlevel1=n, where n indicates and integer, and xxx indicates a list of distribution types or "default".

Distributed Processing and Multi-processing Job using 4 Cores, with 2 Cores for Multi-processing

```
bsub -n 4 -R "span[ptile=2]" ~/projects/OptimTee.csh
```

Shell Script (~/projects/OptimTee.csh):

If a command is included in the **bsub** command line, the entire command will be processed by the command shell two times. The command is processed when the **bsub** command is processed by the shell and is processed again when the command is started by the scheduler. This example shows how to use a shell script so that the command line will be processed only once. The command is placed in the shell script, and then the shell script pathname is placed in the **bsub** command line. Then, the command is only processed by the command processor when the job is started. When using this approach, the shell script should be accessible from all of the cluster hosts.

```
#!/bin/csh
"C:\Program Files\ANSYS Inc\v252\AnsysEM\ansysedt" -ng -BatchSolve
-Distributed -machinelist num=2 -batchoptions
~/projects/OptimTee.aedt
```

- The **-n 4** option indicates that the four cores are needed for the job.
- The **-R "span[ptile=2]"** option indicates that the cores must be allocated in groups of two cores on the same machine.
- The **-machinelist num=2** option indicates that this is a DSO job and that a total of two engines will be started.
- The **hfss** command is placed in the shell script (~/projects/OptimTee.csh). In the **bsub** command line, the **hfss** command is replaced by the shell script pathname.

Monitoring LSF Batch Jobs

You can monitor jobs through the Electronics Desktop user interface, or through the command line.

The suggestions below are for batch jobs run under LSF.

Ansys Electronics Desktop -monitor Command Line Option

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The -monitor command line option enables batch job output to the standard output and standard error streams. The warning, info, and progress messages are sent to the standard output stream. The error and fatal messages are sent to the standard error stream.

LSF bpeek Command

The LSF bpeek command may be used to monitor job progress. The command **pbeek [-f]** *job_id* displays the standard output and standard error produced by the job with id *job_id* from the job start to the current time (the time when the command is executed). This command is only valid for jobs that have not yet finished. When used with the **-f** option on Linux, the output of the job is displayed using the command tail **-f**, so that ongoing progress may be monitored.

In order to display messages to standard output and standard error, specify the **-monitor** command line option on the Ansys EM tool command line. Then, these messages can be seen using the LSF bpeek command.

Terminating LSF Batch Jobs

To cancel or terminate an Ansys EM LSF batch job, we recommend using the job monitoring UI to terminate jobs cleanly, rather than using the bkill commands. Using this approach will allow the batch job to shut down in an orderly fashion.

Using the LSF **bkill** command without the **-s SIGTERM** option or simply terminating the job processes may cause some of the following problems:

- Some engine processes are not shut down and continue to run.
- LSF job is not fully removed.
- Project .lock file is not removed.
- MainWin core service processes (watchdog, mwrpcss and/or regss) are not stopped.

Some of these may interfere with submission of additional LSF batch jobs. For example, it may be necessary to manually remove the project lock file to submit another batch job for the same project. MainWin core service processes may also interfere with starting subsequent Ansoft batch jobs. Normally, these processes should timeout and end 15 seconds after the Ansys Electromagnetics product shuts down. Any MainWin core service processes (watchdog, mwrpcss and/or regss) that continue to run for more than 15 seconds after the product has stopped may be hung. The hung processes may need to be manually killed, after ensuring that these processes are associated with an Ansys EM job that has finished or terminated.

Stop a job cleanly - ensures that the results obtained until now are preserved:

```
bkill -s TERM <jobid>.
```

Stop an job abruptly - results are most likely lost. You have to manually remove the project lock file:

```
bkill <jobid>
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LSF Known Issues and Workarounds

The following are known issues. Workarounds are noted when available:

- Desktop or remote machine cannot have multiple IP addresses. This is unsupported.
- Core dump files may appear when a job has finished running. Results are still computed correctly. Workaround: Limit size of core dumps to 0 through the following job submit option: `bsub -C 0 -n <number-of-cores> -q <queue-name>`
- Firewall should be turned off on all machines in the cluster.
- Sometimes LSF ends a job (for example, a job may be preempted due to a high priority job). This may result in the presence of a .lock file in the project directory. You must manually delete the .lock file before continuing with further analysis.
- When an LSF job is ended, MainWin services (watchdog, regss, and mwrcpss) could keep running. The result is that later jobs cannot start on the machine. The fix is to end these processes before starting a new job.
- Analysis fails abruptly when running out of resources (cpu/memory/disk). Ensure sufficient resources are provided.

LSF Troubleshooting

The following are general troubleshooting steps:

1. Ensure the LSF `lsrun` command is enabled.
2. Look for user errors.

   For example:

   - Are the executable path and project path correct and complete?
   - Are there sufficient resources (CPU/Memory/Disk) allocated to the job?
   - Is the project available on the execution host?
   - Does the job submitter have read/writer permissions on the project directory and read/execute permissions on the installation directory?
   - Is the project locked?
3. Determine whether this is a standalone product issue.
   - Run Electronics Desktop on the machine outside of the scheduler and see if it opens and analyzes.
4. Examine outputs and logs.
   - Output of the LSF batch job. Obtain this using LSF commands: `"bacct -l <jobid>"`
   - Batch log (typically <projectname>.log, located in the project directory.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Enable additional debug logs using the steps below.

   In the job submission window, set the following environment variables:

   - ANSOFT_DEBUG_MODE = 1
   - ANSOFT_DEBUG_LOG = <*path to directory accessible by all machines in the cluster*>
   - ANSOFT_DEBUG_LOG_SEPARATE = 1
   - ANSOFT_LSF_LOG = <*path to a specific .log file in the directory set under ANSOFT_DEBUG_LOG*>
6. For each pair of machines between which remote analysis fails, run `ping remote-machine` and note the output.
7. For each machine in the network, dump network interfaces (for example, run `ifconfig -a`) and note the output.

Integration with PBS (Portable Batch System)

The PBSPro and PBS/Torque schedulers are only supported on Linux. Jobs may be submitted in any of the following ways:

- Job Submission GUI
- Using PBS commands (qsub, etc.) or the PBS gui (xpbs)

For information about supported schedulers, visit the Platform Support page. Then locate and click the following link:

Ansys 2025 R2 – Job Schedulers and Queuing Systems Support

PBS Job Management

You can use Ansys Electronics Desktop to submit batch jobs to PBS and monitor those jobs.

This involves the following steps:

1. Use **Tools** > **Job Management** > **Select Scheduler** to select PBS as the scheduler.
2. Use **Tools** > **Job Management** > **Submit Job** to submit a batch job to PBS.

   PBS-specific Settings:

   - On the **Compute Resources** tab, click the **Resource Selection Parameters** ellipses button (**...**) to specify either the **Queue** or **QueueAtServer** parameter.

     For **Queue**, you may select a queue for the job from the list of queues configured for the default server. Only queues that are enabled and that do not have the from_ route_only attribute set to true are listed.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For **QueueAtServer**, you may specify a queue at the default server or at another server by entering text into this field in one of three formats:

- queue_name
- @server_name
- queue_name@server_name

The queue_name format submits with the name queue_name. The other formats submit with the name server_name. The *destination* value of the -q *destination* option on the qsub command line is the user-specified string. *This string will not be validated by the scheduler proxy library*.

Either **Queue** or **QueueAtServer** may be specified, but both may not be specified. If neither **Queue** nor **QueueAtServer** is specified, the job is submitted to the default queue at the default server.

3. Use **Tools** > **Job Management** > **Monitor Job** to monitor the job's progress.

Non Standard Installations for PBS

PBSPro

If the environment variable PBS_DEFAULT is set, then the value of this environment variable will be used as the name of the default server, instead of obtaining the default server name from the PBSPro configuration file. The default pathname of the PBSPro configuration file is /etc/pbs.conf. The environment variable PBS_CONF may be used to specify a different pathname for the PBSPro configuration file.

PBS/Torque

If the environment variable PBS_DEFAULT is set, then the value of this environment variable will be used as the name of the default server, instead of obtaining the default server name from the PBS/Torque server file. The name of the PBS/Torque server file is server_name, and it is installed in the TORQUEHOME directory. By default, TORQUEHOME is /var/spool/torque. To specify a different TORQUEHOME directory, the environment variable ANSOFT_TORQUEHOME should be set to the pathname of the desired directory.

PBS Limitations

General Limitations

- There is no support for GPUs when submitting jobs via the GUI.
- Support for PBSPro and PBS/Torque is only available on Linux; Windows is not supported.
- Staging of input or output files is not supported for jobs submitted using the GUI.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- All jobs submitted via the GUI are independent jobs. Neither job dependencies nor job arrays are supported.
- If the user specified server is not the default server, then there is no check for sufficient resources before submitting the job.
- If the user specified server is not the default server, then the limit on the number of tasks per node is ignored for both PBSPro and PBS/Torque. For PBS/Torque, only one task will be allocated for each node. For PBSPro, the scheduler may allocate any number of tasks to a node, provided that the node has sufficient cores for all of the tasks.
- For jobs submitted to a routing queue, the check for sufficient nodes and cores only verifies that there are sufficient nodes and cores associated with the server.
- The queue attributes “resources_max” and “resources_min” are not checked when determining whether there are adequate resources to run the job.

PBSPro Limitations

- The PATH in the submission user’s default environment must include the directory containing the PBSPro commands.
- Failover is not supported.
- HPC Basic Profile Jobs are not supported.
- Globus vnodes are not supported. When checking for sufficient nodes and cores for the job, only nodes of type PBS are considered.
- For jobs submitted to an execution queue, the only vnode attribute used to determine if a vnode is available to the job is the “queue” attribute.

PBS/Torque Limitations

- When checking for sufficient nodes and cores for the job, only nodes of type cluster are considered.
- The “exclusive” check box has no effect for PBS/Torque.
- For PBS/Torque, even if a job is submitted to an execution queue at the default server, there is no check for sufficient nodes and cores available to the queue. All of the server’s execution nodes are assumed to be available for the job.
- For PBS/Torque, there are significant limitations when submitting a job in which the number of tasks and number of cores per task are specified. Unlike PBSPro, there is no capability to specify that the cores should be allocated in “chunks”. Instead, the submission command includes the number of groups of nodes and the number of processors per node (ppn) for each node in the group. To determine the size of each group and the ppn setting for each group, the server nodes are examined from largest number of cores to smallest. This may not be optimal because some of these nodes may not be usable by the queue specified for the job, or because the nodes with the largest number of cores may be busy. Similar issues could occur for PBSPro, but they should be less likely, because only the nodes usable by the queue are considered.

Submitting Jobs Via Ansoftrsmservice on a Different Host

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Before starting the ansoftrsmsevice as a daemon on a job submission host, the 'Scheduler' section of the ansoftrsmservice.cfg must be specified.

This section contains two settings, 'SchedulerName', which must be set the string 'pbs', and 'ConfigString', which must be set to the pathname of the directory containing the PBSPro or PBS/Torque commands.

Here is an example ansoftrsmservice.cfg file, showing the format of this file and an example 'ConfigString' setting:

```
$begin 'AnsoftCOMDaemon'
$begin 'Managed COM Servers'
$end 'Managed COM Servers'
$begin 'Scheduler'
'SchedulerName'='pbs'
'ConfigString'='/share/pbs/default/bin'
$end 'Scheduler'
$end 'AnsoftCOMDaemon'
```

Submitting Ansys EM PBS Batch Jobs

The PBS qsub command may be used to submit Ansys EM batch jobs. The typical command format is:

qsub *qsub_args script*

where:

- *qsub_args* are the options of the **qsub** command,
- *script* is the pathname of the job script.

The job script is a shell script containing the Ansys batch command or commands to be run. If a batch command line contains any characters that are special to the shell running the script, then these special characters should be quoted, as needed. The job script may also contain PBS directives on lines before the first executable line of the script. Any **qsub** options on the command line will take precedence over the PBS directives in the job script.

When a PBS batch job is started, the job script runs as the job user in a new shell. In this shell environment, the path must include the directory containing the PBS commands.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

You should ensure that the PATH variable set in the shell startup script (e.g., .cshrc, .profile, .bashrc) includes the directory containing the PBS commands. For example:

```
export PATH=/opt/pbs/default/bin:$PATH
```

If the PATH variable is not set correctly, the job runs only locally, the batch log file shows the list of allocated hosts as empty, and the error file shows an error (sh: qstat: command not found.)

Further PBS directives need to be on top of the job script file. This is discussed in the PBS documentation.

Serial PBS Batch Jobs

In the PBS documentation, serial batch jobs are also called single-node jobs. In general, any job submitted without specifying the -l nodes=value command line argument, will run as a serial or single-node job.

See Monitoring PBS Batch Jobs for options that can facilitate monitoring of Ansys Electromagnetics batch jobs.

Parallel PBS Batch Jobs

In the PBS documentation, parallel batch jobs are also called multi-node jobs. When an Ansys Electromagnetics batch job is run as an PBS parallel job, the PBS scheduler will select the hosts for the distributed analysis job based on the qsub command line arguments, the PBS resource directives from the job script, and the status of the hosts when the job is run. The desktop process will be started on one of these hosts. The desktop process will obtain the list of hosts allocated to the job from the PBS scheduler, and start analysis processes on these hosts, as needed, using the PBS scheduler facilities. To run a PBS parallel job, the job must be submitted with a -l nodes=value **qsub** command line argument or with a -l nodes=value PBS directive in the job script.

Monitoring PBS Batch Jobs

You can monitor jobs through the Electronics Desktop user interface, or through the command line.

The suggestions below are for batch jobs run under PBS.

PBS qstat Command

The PBS **qstat** command may be used to display information on jobs and queues. In this section, several qstat command line options that may be used to monitor job progress are described.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **qstat -a** command displays information about all jobs in the system.

The **qstat -r** command displays information about all running jobs in the system.

The **qstat -s** command resembles the **qstat -r** command; the only difference is that a comment from the scheduler or batch administrator is also shown for each job.

The **qstat -au** *userid* command displays information about all jobs owned by user userid.

The **qstat -f** *jobid* command displays all available information about the job with id *jobid*.

See the PBS manual pages for more information.

Ansys EM -monitor Command Line Option for PBS

The Ansys EM **-monitor** command line option enables batch job output to the standard output and standard error streams. The warning, info, and progress messages are sent to the standard output stream. The error and fatal messages are sent to the standard error stream.

The PBS scheduler redirects the standard output and standard error streams of batch jobs to files specified in the **qsub -o** [*hostname*:]*pathname* and the **-e** [*hostname*:]*pathname* command line options, respectively. If either option is not specified, then the associated stream is redirected to the default file pathname for that stream.

The **qsub -j join** option controls whether the standard error stream for the job will be merged with the standard output stream for the job. A join value of **oe** indicates that the interleaved standard output and standard error will be sent to the standard output file or stream. A join value of **oe** indicates that the interleaved standard output and standard error will be sent to the standard error file or stream. A join value of n indicates that the standard output and standard error streams will not be joined. If the **qsub -j join** option is not specified, then the standard error and standard output streams will not be joined.

A user can monitor the progress of a job by checking the standard output file for progress, info and warning messages, and checking the standard error file for error and fatal messages.

qsub Arguments

The PBS **qsub** command has a large number of options for control of the submission process. In this section, we review the -l nodes=*value* command line option with Ansoft parallel batch jobs.

This option or directive has the following format:

`-l nodes=`*`node_spec`* `[+`*`node_spec...`*`][#`*`suffix`*`]`

where *node_spec* is one of the following

`nodename[:`*`pc_spec`*`[:`*`pc_spec...`*`]]`

Host name of the specified node, followed by optional **ppn** or **cpp** specifiers.

`[`*`N`*`][:`*`property`*`[:`*`property...`*`]][:`*`pc_spec`*`[:`*`pc_spec...`*`]]`

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Optional number of nodes, followed by optional node properties, followed by optional **ppn** or **cpp** specifiers. If the number *N* is omitted, then the default value of 1 host is used.

Here, the optional **ppn** or **cpp** specifiers *pc_spec* are of form:

`ppn=X`

Number of processes (tasks) per node. Default is 1 if not specified.

`cpp=Y`

Number of CPUs (threads) per process. Default is 1 if not specified.

The optional global suffix, #suffix, which applies to all hosts has one of the following values:

`#excl`

This suffix requests exclusive access to the allocated nodes.

`#shared`

This suffix requests shared access to the allocated nodes.

The total number of requested processes is determined by adding up the product of the number of nodes and the number of processes per node for each *node_spec*. In general, this should match the number of distributed engines specified in the Ansys Electromagnetics desktop - Machinelist num=*num_distributed_engines* command line option.

The number of CPUs per process (**cpp**) specified in the PBS **qsub** command line or in the PBS directives in the script file should generally match the number of processors per engine specified in the Desktop **-batchoptions** value.

See the PBS documentation for a complete list of options for the **bsub** command, and further information on running multi-node jobs.

Example PBS qsub Command Lines

All of the following examples show how to submit Linux hfss jobs on PBS, but similar command lines and job scripts will work for all Ansys EM products. Most of the following examples are PBS "Single-node jobs." The last example is a PBS "multi-node jobs"; this example demonstrates how to specify the allocation of threads, tasks and nodes to a job.

Serial job:

```
qsub ~/pbs_scripts/OptimTee.sh
```

Job Script File:

```
#!/bin/sh
/opt/ansys_inc/v252/AnsysEM/ansysedt -ng -BatchSolve
~/projects/OptimTee.aedt
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Serial job that needs a minimum of 4GB memory and two hours of real (wallclock) time:

```
qsub ~/pbs_scripts/OptimTee.sh
```

Job Script File:

```
#!/bin/sh
#PBS -l walltime=2:00:00
#PBS -l mem=4gb
/opt/ansys_inc/v252/AnsysEM/ansysedt -ng -BatchSolve
~/projects/OptimTee.aedt
```

Multi-processing job using 4 cores:

```
qsub ~/pbs_scripts/OptimTee.sh
```

Job Script File:

```
#!/bin/sh
#PBS -l ncpus=4
/opt/ansys_inc/v252/AnsysEM/ansysedt -ng -BatchSolve -batchoptions
-machinelist num=4
~/projects/OptimTee.aedt
```

- The #PBS -l ncpus=4 directive indicates that four cores or CPUs are allocated to this job.
- The -batchoptions option indicates that Ansys Electronics Desktop should use four cores for multi-processing.

Distributed processing job using 4 engines on a single host:

```
qsub ~/pbs_scripts/OptimTee.sh
```

Job Script File:

```
#!/bin/sh
#PBS -l ncpus=4
/opt/ansys_inc/v252/AnsysEM/ansysedt -ng -BatchSolve -Distributed -
machinelist num=4
~/projects/OptimTee.aedt
```

- The #PBS -l ncpus=4 directive indicates that four cores or CPUs are allocated to this job.
- The -Distributed option indicates that this is a DSO job, so that multiple engines will be started. Because 4 cores are allocated to the job, the job will run 4 engines. The -Distributed option may now have additional options, such as includetypes=xxx, excludetyeps=xxx, maxlevels=n, and numlevel1=n, where n indicates and integer, and xxx indicates a list of distribution types or "default".

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Distributed processing and multi-processing job using 8 cores on two nodes, running 4 engines (two per node) with 2 cores for multi-processing:

```
qsub ~/pbs_scripts/OptimTee.sh
```

Job Script File:

```
#!/bin/sh
#PBS -l nodes=2:ppn=2:cpp=2#excl
/opt/ansys_inc/v252/AnsysEM/ansysedt -ng -BatchSolve -Distributed
-machinelist num=4 -batchoptions ~/projects/OptimTee.aedt
```

- The PBS directive **#PBS -l nodes=2:ppn=1:cpp=2#shared** indicates that two nodes are requested [2], two processes (engines) run on each node [ppn=2], and each process will use two cores [cpp=2]. The hosts allocated to this job may not be used for any other jobs while this job is running [#excl].
- The **-machinelist num=4** option indicates that this is a DSO job and that a total of four engines will be started. This option is required for all batch jobs.

Integration with SLURM Scheduler (Linux Utility)

For Linux customers, Electronics Desktop supports the SLURM (Simple Linux Utility for Resource Management) scheduler for job submission, monitoring, and control (abort). In addition, this provides Windows-to-Linux support for the same functionality. Using SLURM as a starting point, customers should be able to develop their own integrations for other schedulers. Electronics Desktop can check job states in SLURM scheduler within the current job monitoring GUI. Persistence mode provides more efficient management between Desktop and scheduler proxies. Windows-to-Linux submission also provides convenient access to generic scheduler in Linux environments through Remote RSM Service.

When you submit a job to SLURM scheduler or Remote RSM to a SLURM cluster, you can use their own job options to override job options in the **Scheduler Options** tab. If you want to do that, the override job options must have "--export=None".

For additional information on supported schedulers, visit the Platform Support page. Then, locate and click the following link:

Ansys 2025 R2 - Job Schedulers and Queuing Systems Support

See also: Integrating Ansys EM Tools with Third Party Schedulers.

Configuration of the generic scheduler with SLURM

Requirements

Files must be under ${ANSYS_EM_INSTALL_DIR}/schedulers directory. Desktop and RSM service use the same required file list below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The required files are listed here:

- ${ANSYS_EM_INSTALL_DIR}/schedulers/libgeneric_scheduler.so
- ${ANSYS_EM_INSTALL_DIR}/schedulers/proxies.cfg
- ${ANSYS_EM_INSTALL_DIR}/schedulers/scripts/persistence.py
- ${ANSYS_EM_INSTALL_DIR}/schedulers/scripts/SLURM_GetInfo_linux.py
- ${ANSYS_EM_INSTALL_DIR}/schedulers/scripts/SLURM_GetRunInfo_linux.py
- ${ANSYS_EM_INSTALL_DIR}/schedulers/scripts/SLURM_SubmitJob_linux.py
- ${ANSYS_EM_INSTALL_DIR}/schedulers/scripts/SLURM_ControlJob_linux.py
- ${ANSYS_EM_INSTALL_DIR}/schedulers/scripts/SLURM_GetJobState_linux.py
- ${ANSYS_EM_INSTALL_DIR}/schedulers/scripts/SLURM_LaunchProcess_linux.py
- ${ANSYS_EM_INSTALL_DIR}/schedulers/scripts/utils/ComputeResourceList.py
- ${ANSYS_EM_INSTALL_DIR}/schedulers/scripts/utils/ErrorAndDebug.py
- ${ANSYS_EM_INSTALL_DIR}/schedulers/scripts/utils/__init__.py
- ${ANSYS_EM_INSTALL_DIR}/schedulers/scripts/utils/RunCommand.py
- ${ANSYS_EM_INSTALL_DIR}/schedulers/scripts/utils/SiteCustomize.py
- ${ANSYS_EM_INSTALL_DIR}/schedulers/scripts/utils/slurm_srun_wrapper.sh

Environment Variables

There are some environment variables for the Generic Scheduler proxy. All environment variables apply on both Desktop and RSM Service.

`ANSYSEM_GENERIC_SCRIPT_DIR`

This variable adds a customized directory to be searched for proxy scripts. The path can be any form of following addresses. Persistence script path will not change.

- absolute address to any folder with read access
- relative address to folder under the Electronics Desktop installation directory
- relative address to folder under the schedulers directory
- relative address to folder under the schedulers/scripts directory

`ANSYSEM_GENERIC_EXEC_PATH`

This variable adds customized directories to be searched for proxy executables. Mono executable and IronPython executable should be specified when they are not in the Ansys Electronics Desktop installation. This environment variable must be set for RSM Service.

`ANSYSEM_GENERIC_PROXY_PERSISTENCE`

If set, this variable redirects the persistence script location to be used; otherwise, the default persistence script path is used.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`ANSYS_EM_EXEC_DIR`

Set this variable to the same value with ANSYS_EM_INSTALL_DIR (not the RSM service path). To ensure the RSM service can accurately locate the essential script files required for MPI integration, it is necessary to set this variable within the RSM service environment.

Configuration File

The configuration file "proxies.cfg" is located at ${ANSYS_EM_INSTALL_DIR}/schedulers/. The file can have multiple scheduler configurations. The file content and explanation are below.

[slurm] ***# Unique scheduler name***

DisplayName=SLURM ***# Display name in GUI***

 # Proxy Scripts

SubmitJob=scripts/SLURM_SubmitJob_linux.py

ControlJob=scripts/SLURM_ControlJob_linux.py

GetSchedInfo=scripts/SLURM_GetInfo_linux.py

GetJobState=scripts/SLURM_GetJobState_linux.py

GetRunInfo=scripts/SLURM_GetRunInfo_linux.py

LaunchProcess=scripts/SLURM_LaunchProcess_linux.py

 # Persistence Mode

Persistence=Yes

Persistence Mode

Persistence mode uses a persistent process to communicate with Ansys Electronics Desktop and proxy scripts. If persistence mode is disabled, Ansys Electronics Desktop uses proxy scripts directly. Enabling persistence mode may help speed up communication with proxy scripts. **Note**: disabling persistence will cause large delays in GUI functionality. The primary use case of disabling persistence is to support troubleshooting.

- To enable persistence mode, make sure "Persistence=Yes" is in configuration file before launching Ansys Electronics Desktop.
- To disable persistence mode, make sure "Persistence=No" is in configuration file before launching Ansys Electronics Desktop.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Remote RSM Service Support

In ansoftrsmservice.cfg, fill your scheduler information from configuration file proxies.cfg as below. `SchedulerName` must be generic. Change `"Proxy"` as needed.

```
$begin 'Scheduler'
  'SchedulerName'='generic'
  'ConfigString'='{"Proxy":"slurm"}'
$end 'Scheduler'
```

Then, set the environment variable “ANSYSEM_GENERIC_EXEC_PATH” with Mono and IronPython path. Multiple paths can be filled with delimiter “:”. Ansys Electronics Desktop has built-in components in AnsysEM installation path, and you can replace the installation path like:

```
ANSYSEM_GENERIC_EXEC_PATH= <INSTALL_DIR>/ansys_
inc/v252/AnsysEM/common/mono/Linux64/bin:<INSTALL_DIR>/ansys_
inc/v252/AnsysEM/common/IronPython
```

Also set environment variable “ANSYS_EM_GENERIC_COMMON_TEMP” to a directory widely accessible to users using Remote RSM Service on the cluster. This environment variable specifies a temporary folder to use. Thus, it must be accessible for most users.

```
ANSYS_EM_GENERIC_COMMON_TEMP=/tmp
```

Then, launch **ansoftrsmservice** in the AnsysEM RSM installation folder. (Be sure not to confuse AnsysEM (Ansoft) RSM with Ansys RSM since these are two very different things.) It is recommended that the user for **ansoftrsmservice** should be a dedicated service user account for only running **ansoftrsmservice**. This account should have limited permissions for security reasons. The service user should not have a login. It is recommended to disable login, so that the only way to run as the user is to “su” to that user from the root (or other accounts with sudo permissions). Other users can use any account credentials in the Desktop Select Scheduler GUI to submit jobs as the requested user.

Using the generic scheduler with SLURM

To select the scheduler and get scheduler information:

1. Click **Tools** > **Job Management** > **Select Scheduler...**, or select the **Simulation** tab on the ribbon, and click the **Scheduler** icon.

   This opens the **Select Scheduler** dialog box.

2. Select **SLURM** from the drop-down menu; or, if the Remote RSM service is used, fill in scheduler master node address, and use valid account credentials.
3. Click **Refresh**.

   Scheduler Information should show.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **OK** to select the scheduler.

   Submit Jobs and Monitor Jobs with SLURM

   Job submission and monitoring follow the same process and method as using other schedulers. Currently, submission using a node-list is not supported.

5. Use **Tools** > **Job Management** > **Monitor Job** to monitor the job's progress.

Using MPI Tight Integration

By default, jobs would be submitted with SSH as remote spawn command method. If MPI tight integration preferred, you should submit the job with an additional batch option setting in the Submit Job Browser:

Enable MPI Tight Integration:

```
<design-type>/RemoteSpawnCommand = "Scheduler"
```

Enable SSH:

```
<design-type>/RemoteSpawnCommand = "SSH"
```

Using the Command Line to Submit HPC Jobs

Ansys Electronics Desktop can be run from the command line. When using the command line to perform HPC jobs, take the following into consideration.

Distributed Jobs

An Ansys EM batch job which distributes the analysis over several hosts may also be called a distributed job. To submit a distributed job, the following Ansys EM desktop command line options should be used:

- The -Distributed option should be present, and the -Local option should be absent. When running as a batch job under one of the schedulers with direct integration, this option is a directive to the job to 1) obtain the list of hosts allocated to the job, directly from the scheduler, and to 2) use the scheduler to launch the analysis engines on the hosts allocated to the job. The -Distributed option may now have additional options, such as includetypes=xxx, excludetypes=xxx, maxlevels=n, and numlevel1=n, where n indicates and integer, and xxx indicates a list of distribution types or "default".
- The -Machinelist num=*num_distributed_engines* option must be included, where *num_distributed_engines* is the total number of analysis engines to be started on the hosts assigned to the job.

Other examples:

- "Serial Job on a Single Processor" on the next page
- "Distributed Job Using Four Processors" on the next page

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- "Multiprocessing Job Using Four Cores" below
- "Distributed Analysis and Multi-Processing in the Same Job" on the facing page

Serial Job on a Single Processor

Suppose Ansys Electronics Desktop is installed at "C:\Program Files\ANSYS Inc\v252\AnsysEM\" and you are using RSM for DSO:

```
"C:\Program Files\ANSYS Inc\v252\AnsysEM\ansysedt.exe" -ng -
BatchSolve -machinelist num=2
-monitor \\shared_drive\projs\OptimTee.aedt
```

User is using LSF for remote-analysis/DSO

```
bsub -n 1 "C:\Program Files\ANSYS Inc\v252\AnsysEM\ansysedt.exe" -ng
-BatchSolve -machinelist num=3 -monitor -local \\shared_
drive\projs\OptimTee.aedt
```

Distributed Job Using Four Processors

Ansoft RSM

```
"C:\Program Files\ANSYS Inc\v252\AnsysEM\ansysedt.exe" -ng -
Batchsolve -monitor -Distributed
-machinelist list="10.1.1.221, 10.1.1.222, 10.1.1.223, 10.1.1.224"
\\shared_drive\projs\OptimTee.aedt
```

LSF

```
bsub -n 4 "C:\Program Files\ANSYS Inc\v252\AnsysEM\ansysedt.exe" -ng
-Batchsolve -monitor
-Distributed -machinelist num=4
\\shared_drive\projs\OptimTee.aedt
```

Multiprocessing Job Using Four Cores

Multi-processing job using 4 cores

```
bsub -n 4 -R span[ptile=4] "C:\Program Files\ANSYS
Inc\v252\AnsysEM\ansysedt.exe" -ng -monitor
-Local -BatchSolve -machinelist num=4 -batchoptions \\shared_
drive\registry.txt \\shared_drive\projs\OptimTee.aedt
```

This requests 4 cores to come from the same machine, as multi-processing needs cores to be on the same machine

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Distributed Analysis and Multi-Processing in the Same Job

Distributed-processing using 4 engines and multi-processing using 4 cores, using a total of 16 cores

```
bsub -n 16 -R "span[ptile=4]" "C:Program Files\ANSYS
Inc\v252\AnsysEM\ansysedt.exe" -ng
-BatchSolve -Distributed -machinelist num=4
-batchoptions \\shared_drive\registry.txt
\\shared_drive\projs\OptimTee.aedt
```

Integrating Ansys EM Tools with Third-Party Schedulers

This document indicates how to create a dynamically linked library to allow integration of Ansys EM tools with an arbitrary scheduler environment. Each scheduler proxy library is used for a single specific scheduler environment. If the library is installed with a valid name and in the correct location, it is then automatically loaded and used by Ansys EM tools.

- Introduction
- Common Requirements for Running Jobs
- Using a Shared Library (Linux) or a DLL (Microsoft Windows)
- Scheduler Proxy Interfaces
- Using an IronPython Program for Scheduler Integration

Introduction

Ansys EM Software Tools may be run as serial or parallel jobs on a cluster under control of a scheduler. Serial jobs are run using a single analysis engine at any one time on a single host. If the tool performs multiple analyses (for a frequency sweep or a parametric analysis, for example), the analyses are performed one after the other. Parallel jobs are run using multiple analysis engines running in parallel on the same host or on separate hosts. For parts of the analysis (such as meshing), the parallel job may use only a single analysis engine on a single host. Other parts of the analysis (such as a frequency sweep, parametric analysis or DDM) may be distributed to multiple analysis engines running in parallel.

- Serial Jobs
- Parallel Jobs

Serial Jobs

When an Ansys EM batch analysis runs as a serial job, the analysis engines run on the same host as the desktop process. The desktop process does not need to interact with the scheduler to get the names of hosts allocated to the job or to start processes on other hosts.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parallel Jobs

For a parallel job, the desktop process starts multiple analysis engines that run in parallel. These engines may be started on the host where the desktop process is running, or on other hosts allocated to the job. The desktop process interacts with the scheduler to obtain information on the hosts that are allocated to the job, and to start engines on the local host or on other hosts allocated to the job. This document provides information on how to facilitate this interaction between the desktop process and the scheduler controlling the cluster.

For some popular job schedulers in a standard configuration, Ansys EM provides an "out of the box" integrated solution that will work with the scheduler. In this case, the Ansys EM installation includes code that will determine if the analysis is running as a scheduler job and communicate with the scheduler when needed. For other schedulers, the code to obtain information about the hosts allocated to a job and to distribute portions of the job to hosts assigned to the job is not provided in the installation. In order to facilitate using Ansys EM Software Tools with other schedulers, the user may provide a way for Ansys EM Tools to interact with the scheduler. Currently, two general approaches are available to users.

In the first approach, the user creates a shared library (on Linux) or a dynamically linked library (on Microsoft Windows) to provide communication between the Ansys EM Tool and the scheduler. This library is loaded by the Ansys EM Tool at runtime, and if the Ansys EM Tool is running as part of a scheduler job, the Ansys EM Tool interacts with the library to get information from the scheduler, and to start additional processes on specified hosts. Each such library implements the same set of extern "C" functions needed to mediate the interactions between the Ansys EM Tool and the scheduler.

In the second approach, the user creates an IronPython program to provide communication between the Ansys EM Tool and the scheduler. This program is loaded by the Ansys EM Tool at runtime, and if the Ansys EM Tool is running as part of a scheduler job, the Ansys EMs Tool uses the IronPython program to get information from the scheduler, and to start additional processes on specified hosts. Each python script contains a class implementing a specified interface, which contains functions needed to mediate the interactions between the Ansys EM Tool and the scheduler. The details of the interface are described below. The IronPython interface is equivalent to the extern "C" functions used in the first approach.

Common Requirements for Running Jobs

The following requirements must be met for serial and parallel jobs to run successfully. They apply whether using "out of the box" scheduler integration or scheduler integration using a library or using an IronPython program. *Host requirements* apply to all hosts that may be allocated to an Ansys EM serial or parallel batch job.

Installation Requirements

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Ansys EM installation directory must be accessible from all cluster hosts using the same path. One way to achieve this is to place the Ansys EM installation on a shared drive that is accessible to the cluster hosts using the same pathname. On Windows, this may require the use of UNC names to refer to the installation directory. Another option is to install the Ansys EM tool locally on each cluster host using the same local directory path.

Project File and Directory Requirements

The directory containing the project file must also be available from all cluster hosts using the same path. The project file and the containing directory must be readable and writable by the user account used to run the job. The controlling process for a distributed job is called the Desktop process, and it reads from and writes to the project file and other files in the same directory and its subdirectories. Although only the Desktop process reads from and writes to this directory, the Desktop process may be started on any of the hosts allocated to the job, so all hosts should have access to this directory using the same path.

For cloud submission such as Ansys HPC Platform Services, and Ansys Cloud Burst Compute there are no requirements for shared drives or shared directories (installation or project).

Using a Shared Library (Linux) or a DLL (Microsoft Windows)

This section describes how to create a dynamically linked library to allow integration of Ansys Electromagnetics Suite 2025 R2 with an arbitrary scheduler environment. Each scheduler proxy library is used for a single specific scheduler environment. If the library is installed with a valid name and in the correct location, then it will automatically be loaded and used by Ansys Electromagnetics Suite 2025 R2.

Installation Details

The scheduler proxy library must be installed in the schedulers subdirectory of the Ansoft installation directory. For example, if the Ansys EM installation directory is "C:\Program Files\v252\AnsysEM", then the scheduler proxy library must be installed in directory "C:\Program Files\ANSYS Inc\v252\AnsysEM\schedulers.

The scheduler proxy library base name must match "libprefix_scheduler" on Windows and "liblibprefix_scheduler" on Linux. The extension must be a valid extension for a dynamically loaded library on the platform where it is used. The scheduler proxy library name prefix libprefix shall be unique, so it does not conflict with other scheduler proxy libraries in the same directory. To avoid confusion, the scheduler proxy library name should be all lower case on OSs where file names are case sensitive.

Build Information for Scheduler Proxy Library

This section contains the recommended compiler and linker settings for building a scheduler proxy library.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 64-Bit Microsoft Windows
- Linux

64-Bit Microsoft Windows

The proxy library should be compiled and linked as a 64-bit DLL, using the following recommended compiler and linker options:

Compiler Options

- Use of MFC: Use Standard Windows Libraries
- Character Set: Use Multi-Byte Character Set [/D "_MBCS"]
- Runtime Library: Multi-threaded DLL [/MD]
- Calling Convention: __cdecl [/Gd (default)]

Linker Options:

- Create a DLL [/DLL]
- 64-bit code [MACHINE:X64]

Linux

The proxy library should be compiled and linked as shared library (*.so) file. The following compiler and linker options are recommended when building using gcc/g++:

Compiler Options

- Generate position-independent code, suitable for use in a shared library: [-fpic]
- Generate code compatible with pthreads library: [-pthread]

Linker Options:

- Create a shared object file: [-shared]
- Generate position-independent code, suitable for use in a shared library: [-fpic]
- Generate code compatible with pthreads library: [-pthread]

Implementation Details for Custom Scheduler Integration

Function Name Prefix

Each exported function will have a scheduler specific function name prefix. The function name prefix will be the same as the library name prefix, except that it is converted to upper case. For example, if the library name prefix is "lsf", then the function name prefix is "LSF". In the examples below, we use FN_PREFIX to denote the function name prefix.

The scheduler proxy library must provide implementations of the following extern "C" functions:

- IsProductLaunchedInYourEnvironment
- GetTempDirectory

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- GetMachineListAvailableForDistribution
- GetMessageStringToRegisterForSigTerm
- LaunchProcess
- GetUseRsmForEngineLaunch
- GetThisJobID
- GetSchedulerDisplayName

IsProductLaunchedInYourEnvironment

Purpose

Determine if the program is running in the context of the scheduler for which this library was written.

Signature

```
extern "C" bool FN_PREFIX_IsProductLaunchedInYourEnvironment();
```

Arguments

None.

Return Value

Returns true if the current process is running as a job of the scheduler. Otherwise, false is returned.

> **Note:**
>
> For many schedulers, the presence of certain environment variables or their values may be checked to determine if the current process is running as a job of the scheduler.

GetTempDirectory

Purpose

Get the pathname of the temporary directory provided by the scheduler for the current job. The pathname is an empty string if the scheduler does not provide a temporary directory for the current job.

Signature

```
extern "C" bool FN_PREFIX_GetTempDirectory(char * buffer,
unsigned int* length);
```

Arguments

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

buffer: Pointer to a character buffer to contain the temporary directory path name or NULL.

length: Pointer to a location to contain the length of the buffer. Must be a valid pointer to an unsigned int.

Return Value

If argument buffer is NULL, then then required length of the buffer is stored in the location to which argument length points, and true is returned.

If argument buffer is not NULL, then the value to which argument length points (the buffer length) is checked. If it is large enough to contain the pathname of the temporary directory, including the terminal null byte, then the pathname is copied to the buffer and true is returned. If the buffer length is insufficient for the pathname of the temporary directory, then the buffer is unchanged, and false is returned.

> **Note:**
>
> To get the pathname of the temporary directory, the infrastructure first calls this function with a NULL buffer, and obtains the required length of the buffer for the pathname. After creating a buffer of the appropriate size, the infrastructure calls this function again, passing the pointer to the buffer in the buffer argument and a pointer to the size of the buffer in the length argument.

GetMachineListAvailableForDistribution

Purpose

Get the list of hosts allocated to the current job. A host will appear in the list multiple times if the scheduler has allocated multiple processors or cores on the host to the job. The number of times the host appears in the list is equal to the number of processors or cores of the host that are allocated to the current job. The list is a text string containing a space separated list of hostnames.

Signature

```
extern "C" bool FN_PREFIX_GetMachineListAvailableForDistribution
(char * buffer, unsigned int* length);
```

Arguments

buffer: Pointer to a character buffer to contain the list of machines available for distribution or NULL.

length: Pointer to a location to contain the length of the buffer. Must be a valid pointer to an unsigned int.

Return Value

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If argument buffer is NULL, then then required length of the buffer is stored in the location to which argument length points, and true is returned.

If argument buffer is not NULL, then the value to which argument length points (the buffer length) is checked. If it is large enough to contain the lists of hosts, including the terminal null byte, then the list is copied to the buffer and true is returned. If the buffer length is insufficient for the list of hosts, then the buffer is unchanged, and false is returned.

Note:

- To get the list of hosts for distribution, the infrastructure first calls this function with a NULL buffer, and obtains the required length of the buffer for the list. After creating a buffer of the appropriate size, the infrastructure calls this function again, passing the pointer to the buffer in the buffer argument and a pointer to the size of the buffer in the length argument.
- The hostnames in the list provided by this function shall be used in calls to LaunchProcess(). These host names must be in a format that is accepted by that function. See the section below on LaunchProcess.

GetMessageStringToRegisterForSigTerm

Purpose

Obsolete. The string copied to the buffer should be an empty string.

Signature

```
extern "C" bool FN_PREFIX_GetMessageStringToRegisterForSigTerm
(char * buffer, unsigned int* length);
```

Arguments

buffer: Pointer to a character buffer to contain the string or NULL.

length: Pointer to a location to contain the length of the buffer. Must be a valid pointer to an unsigned int.

Return Value

If argument buffer is NULL, then the required length of the buffer is stored in the location to which argument length points, and true is returned.

If argument buffer is not NULL, then the value to which argument length points (the buffer length) is checked. If it is large enough to contain the string, including the terminal null byte, then the string is copied to the buffer and true is returned. If the buffer length is insufficient for the string, then the buffer is unchanged, and false is returned.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> To get the string, the infrastructure first calls this function with a NULL buffer, and obtains the required length of the buffer for the string. After creating a buffer of the appropriate size, the infrastructure calls this function again, passing the pointer to the buffer in the buffer argument and a pointer to the size of the buffer in the length argument.

LaunchProcess

Purpose

Launch a local or remote process to run an analysis engine. This function is called by the Ansys Electromagnetics desktop application to launch an engine process on a specified host. The hostname is one of the names in the list provided by the GetMachineListAvailableForDistribution function. If the hostname does not refer to the local host, then this function shall use the scheduler to launch the engine on the specified host. If the hostname refers to the local host, then the engine may be started as a child process, or it may be started using the scheduler.

Signature

```
extern "C" int FN_PREFIX_LaunchProcess(const char* hostName,
const char* exePathName, const char* arg1, const char* arg2);
```

Arguments

hostName: The name of the host where the process is to be launched.

exePathName: The pathname of the analysis engine executable to be started.

arg1: The first argument of the analysis engine command line.

arg2: The second argument of the analysis engine command line.

Return Value

Returns 0 on success. Returns a non-zero value if an error occurs.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- The hostName argument will be one of the hostnames provided by the function GetMachineListAvailableForDistribution().
- If the hostName argument is the same as the current host, then the analysis engine process may be started as a child process. If the hostName argument is not the same as the current host, then the analysis engine process will be started on the remote host using the facilities available in the scheduler environment. The command line of the analysis engine process is exePathName arg1 arg2. The command line arguments arg1 and arg2 may contain newlines, tabs, spaces or other characters that are interpreted by the command processor, such as single quote (') or double quote (") characters, or dollar signs ($). Newlines or tabs may be replaced by spaces, if the newline or tab characters cannot be easily handled. If the analysis engine command is processed by a shell, then it may be necessary to quote any special characters in the exePathName or in the arguments so that the special meaning is removed. If a scheduler command is used to request the scheduler to launch the command to start the engine process, the analysis engine command may be processed by the shell twice: once when the scheduler command is processed, and a second time when the analysis engine process is started. If this is the case, then the quoting of special characters needs to account for two passes through the command processor.

GetUseRsmForEngineLaunch

Purpose

This function is optional. If this feature is not needed, then the function need not be implemented. Most schedulers should not need this feature.

For some schedulers, it may be desirable for the Ansoft RSM service to launch the engine processes instead of using the scheduler proxy library. For example, if the scheduler proxy library is limited to launching one process per host, then the scheduler proxy library may be used to launch one Ansoft RSM service executable per host, and the Ansoft RSM executable will launch all of the engine processes.

If the Ansoft RSM service should be used to launch engine processes for this scheduler, then this function shall be implemented and it shall return true.

If the Ansoft RSM service should not be used to launch engine processes for this scheduler, then this function is not required. If it is implemented, it should return false. If it is not implemented, it will be treated the same as if it was implemented and returns false.

Signature

```
extern "C" bool FN_PREFIX_GetUseRsmForEngineLaunch(void)
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Arguments

None.

Return Value

Returns true if the Ansoft RSM service should be used to launch engine processes for this scheduler. Returns false if the Ansoft RSM service should not be used to launch engine processes for this scheduler.

> **Note:**
>
> This function is optional. If not implemented, then it is treated the same as if it was implemented and returns false.

GetThisJobID

Purpose

Get a string identifying the job currently running in the scheduler environment. This string is displayed to the end user to identify the job.

Signature

```
extern "C" bool FN_PREFIX_GetThisJobID(char * buffer, unsigned
int* length);
```

Arguments

buffer: Pointer to a character buffer to contain the Job ID or NULL.

length: Pointer to a location to contain the length of the buffer. Must be a valid pointer to an unsigned int.

Return Value

If argument buffer is NULL, then then required length of the buffer is stored in the location to which argument length points, and true is returned.

If argument buffer is not NULL, then the value to which argument length points (the buffer length) is checked. If it is large enough to contain the string identifying the current job, including the terminal null byte, then the job ID is copied to the buffer and true is returned. If the buffer length is insufficient for the job ID, then the buffer is unchanged, and false is returned.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- To get the job ID, the infrastructure first calls this function with a NULL buffer, and obtains the required length of the buffer for the job ID. After creating a buffer of the appropriate size, the infrastructure calls this function again, passing the pointer to the buffer in the buffer argument and a pointer to the size of the buffer in the length argument.
- For many schedulers, the job ID may be obtained from the value of an environment variable.

GetSchedulerDisplayName

Purpose

Get a string identifying the scheduler associated with the current scheduler proxy library. This string is displayed to the end user to identify the scheduler.

Signature

```
extern "C" bool FN_PREFIX_GetSchedulerDisplayName(char * buffer,
unsigned int* length);
```

Arguments

buffer: Pointer to a character buffer to contain the scheduler display name or NULL.

length: Pointer to a location to contain the length of the buffer. Must be a valid pointer to an unsigned int.

Return Value

If argument buffer is NULL, then required length of the buffer is stored in the location to which argument length points, and true is returned.

If argument buffer is not NULL, then the value to which argument length points (the buffer length) is checked. If it is large enough to contain the scheduler display name, including the terminal null byte, then the scheduler display name is copied to the buffer and true is returned. If the buffer length is insufficient for the scheduler display name, then the buffer is unchanged, and false is returned.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- To get the scheduler display name, the infrastructure first calls this function with a NULL buffer, and obtains the required length of the buffer for the scheduler display name. After creating a buffer of the appropriate size, the infrastructure calls this function again, passing the pointer to the buffer in the buffer argument and a pointer to the size of the buffer in the length argument.
- The scheduler display name is generally a fixed string.

Extending Scheduler Integration with Tight-Integration for MPI

AnsysEM supports tight-integration with a few popular schedulers by leveraging built-in tight-integration capabilities of intel MPI. Specifically on Linux, this works out of the box with LSF, SGE and Ansys Cloud Burst Compute. For other Linux based schedulers, this note describes how scheduler integration developers can extend scheduler integration to provide tight integration for MPI based solves.

MPI Based Solves

Like many other simulation applications, AnsysEM also uses an MPI-based message passing approach to provide portable parallel execution on shared, distributed and distributed shared memory computers.

An MPI solve involves running a mpiexec (or mpirun) command that runs the simulation's executable on each allocated/specified host. For most applications, the entire simulation is just a single MPI solve. However, an AnsysEM simulation consists of multiple MPI solves which are organized as following:

1. Multiple MPI solves in series: Each AnsysEM simulation runs multiple MPI solves.
2. Different host list: Each such MPI solves may use different amount of resources (mostly cores per host) that is specified using the host list and count of ranks.
3. Multiple MPI solves in parallel: In some case, multiple MPI solves may be launched in parallel from different hosts. So multiple MPI solves may be running simultaneously.
4. Multi-processing: Finally, within each MPI based solve, each rank can use multi-processing as well.

To launch an MPI based solve AnsysEM code uses an MPI integration related wrapper file, called mpirun.fl, which in turn invokes MPI's mpirun/mpiexec. Ansys EM installation includes all the needed files for the supported MPI implementation's.

Scheduler integration for MPI

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When a scheduler runs a simulation, each scheduler job usually runs just a single MPI solve. Each time MPI needs to launch a process on a remote allocated host, it will do so either with password-less ssh (loose integration), or with a scheduler specific script/command (tight-integration).

Custom Tight Integration

For custom tight integration, you provide a script that uses scheduler's utilities to launch processes on a remote host. The script needs to be a drop-in replacement for ssh. For AnsysEM to use this script for MPI solves, you need to specify the following environment variables and provide bootstrap selector and bootstrap script file.

Environment Variables	Value	Description
ANSYSEM_HPC_MPI_USE_SCHEDULER_CMD	A keyword that identifies the scheduler	Scheduler for which to enable tight-integration
ANSYSEM_HPC_MPI_CUSTOM_SCRIPT_FILE	Fully qualified path to the custom script file	Script that implements a function that returns the switch to be used for MPI implementation's command line.

Script Files

Script Files	Content	Description
Bootstrap selector BASH script file	implements a bash function that returns the switch to be used for MPI implementation's command line.	**Intel MPI**: Implements `ansysem_hpc_intel_mpi_bootstrap` either returns -bootstrap or -bootstrap-exec with a suitable argument
Bootstrap BASH script file	Shell script that is a ssh drop-in replacement	Launches a process on a remote host. Accepts the same arguments as ssh

Example: PBSPro

For custom tight integration with PBSPro, you provide a script that uses pbsdsh utility to launch processes on a remote host. The script is a drop-in replacement for ssh. It assumes that the first two arguments are -x and -q. and provide bootstrap selector and bootstrap script file

For an AnsysEM batch solve runnning under PBSPro to use this script for MPI solves, the job script will set the following environment variables.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Job script excerpt:

```
# The selector script will specify the MPI options for PBSPro
export ANSYSEM_HPC_MPI_USE_SCHEDULER_CMD=pbspro
# Bootstrap selector BASH script file
export ANSYSEM_HPC_MPI_CUSTOM_SCRIPT_FILE=${INSTALL_DIR}/ansys_
inc/v252/AnsysEM/schedulers/utils/pbspro-intelMPI-integration.sh
# Special environment variable to enable PBS Node file to become
available in Bootstrap script file.
export ANSYSEM_HPC_MPI_SCHEDULER_NODEFILE=$PBS_NODEFILE
```

pbspro-intelMPI-integration.sh

```
#!/bin/bash
function ansysem_hpc_intel_mpi_bootstrap()
{
  local custom_scheduler="$1"
  local bootstrap_switch=""
  if [[ -n "$custom_scheduler" ]]
  then
    if [[ "$custom_scheduler" == "pbspro" ]]
    then
      exec_file=$(dirname $ANSYSEM_HPC_MPI_CUSTOM_SCRIPT_FILE)
      exec_file_name=pbspro_dsh_wrapper.sh
      bootstrap_switch="-bootstrap-exec $exec_file/$exec_file_name"
    fi
fi
  echo "$bootstrap_switch"
}
```

pbspro_dsh_wrapper.sh

```
#!/bin/bash
# Bootstrap BASH script file for PBSPro
# Assumes that the first two arguments are -x and -q
# For debugging, set it to a suitable path
# DEBUG_DIR=$(HOME)/Ansoft/Test/DebugLog
PBS_BINDIR=/opt/pbs/bin
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
shift 2
host_name=$1
# Assumes that the next argument is host name shift 1
# Compute the index for the first entry
index1=$(( $(grep -n -m 1 $host_name $ANSYSEM_HPC_MPI_SCHEDULER_
NODEFILE | cut -f1 -d:) - 1))
if [[ -n "$DEBUG_DIR" ]]
then
  echo "running pbsdsh ON $host_name as index $index1 $@" >>
  ${DEBUG_DIR}/wrapper.log
  cat $ANSYSEM_HPC_MPI_SCHEDULER_NODEFILE >> ${DEBUG_
  DIR}/wrapper.log
fi
$PBS_BINDIR/pbsdsh -v -n $index1 -- "$@"
```

Intel MPI custom bootstrap feature

From Intel MPI docs:

```
-bootstrap-exec <bootstrap server>
```

Use this option to set the executable to be used as a bootstrap server. For example:

```
> mpiexec -bootstrap-exec <bootstrap_server_executable> -f hostfile
-env <VAR1> <VAL1> -n 2 test.exe
```

This -bootstrap-exec switch lets you specify a shell script that is a ssh drop-in replacement. For custom tight-integration, AnsysEM uses Intel MPI, and leverages its special switch. This -bootstrap-exec switch lets you specify a shell script that is a ssh drop-in replacement. Using it requires a small patch to an MPI integration related wrapper file, called mpirun.fl. This file, which is available in AnsysEm installation, is used by AnsysEM code to launch an MPI based solve (similar to using mpirun). Each time Intel MPI needs to launch a process on a remote allocated host, it will call a scheduler (or user) specified script using arguments that are same as those for password-less ssh.

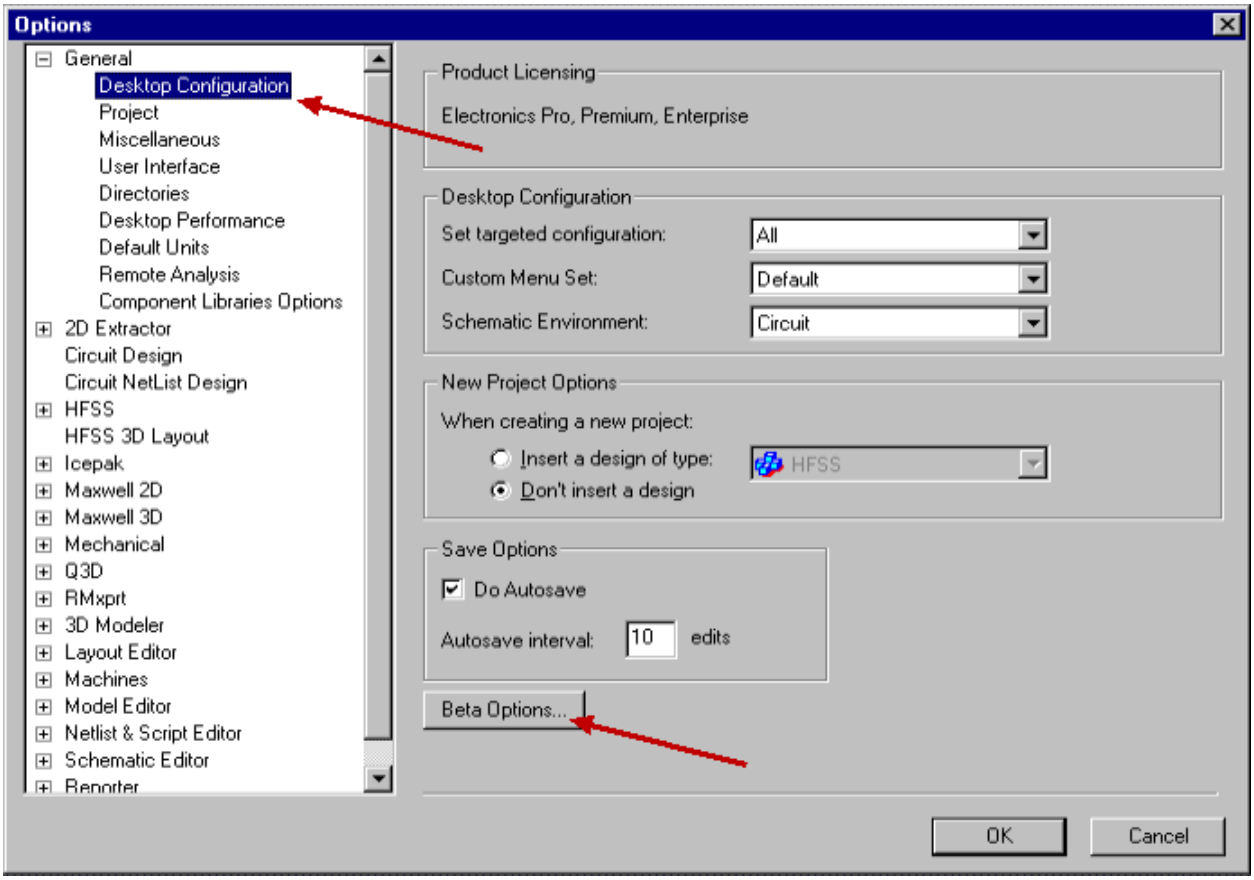
Support for Open MPI on Linux [Beta]

Support for Open MPI on Linux is available as a beta feature for HFSS, HFSS 3D Layout, Icepak, Maxwell (3D and 2D), and Q3D designs.

To enable the beta feature:

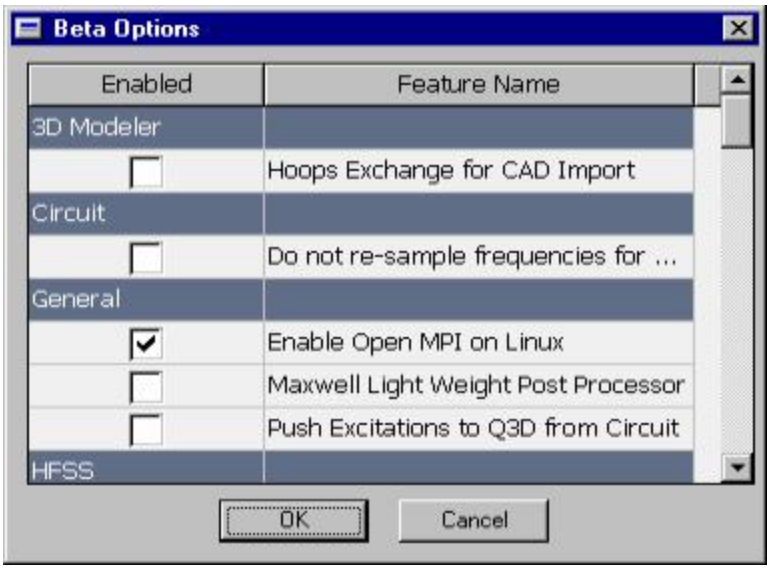
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Access the *Options* dialog box using one of the following two mehtods:
   - From the menu bar, click **Tools** > **Options** > **General Options**.
   - On the **Desktop** ribbon tab, click **General Options**.
2. In the tree on the left side of the dialog box, expand **General** and select **Desktop Configuration**.
3. Click **Beta Options**:



4. Select **Enable Open MPI on Linux**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. When making a change to any beta option, you may be prompted to restart the Ansys Electronics Desktop application. If prompted, click **Yes** to do so.

   The **Open MPI** option will now be available in the **MPI Vendor** drop-down menu within the *Options* tab of the *HPC and Analysis Options* dialog box. You only need to perform this procedure once; the Open MPI option will remain available for future sessions and projects.

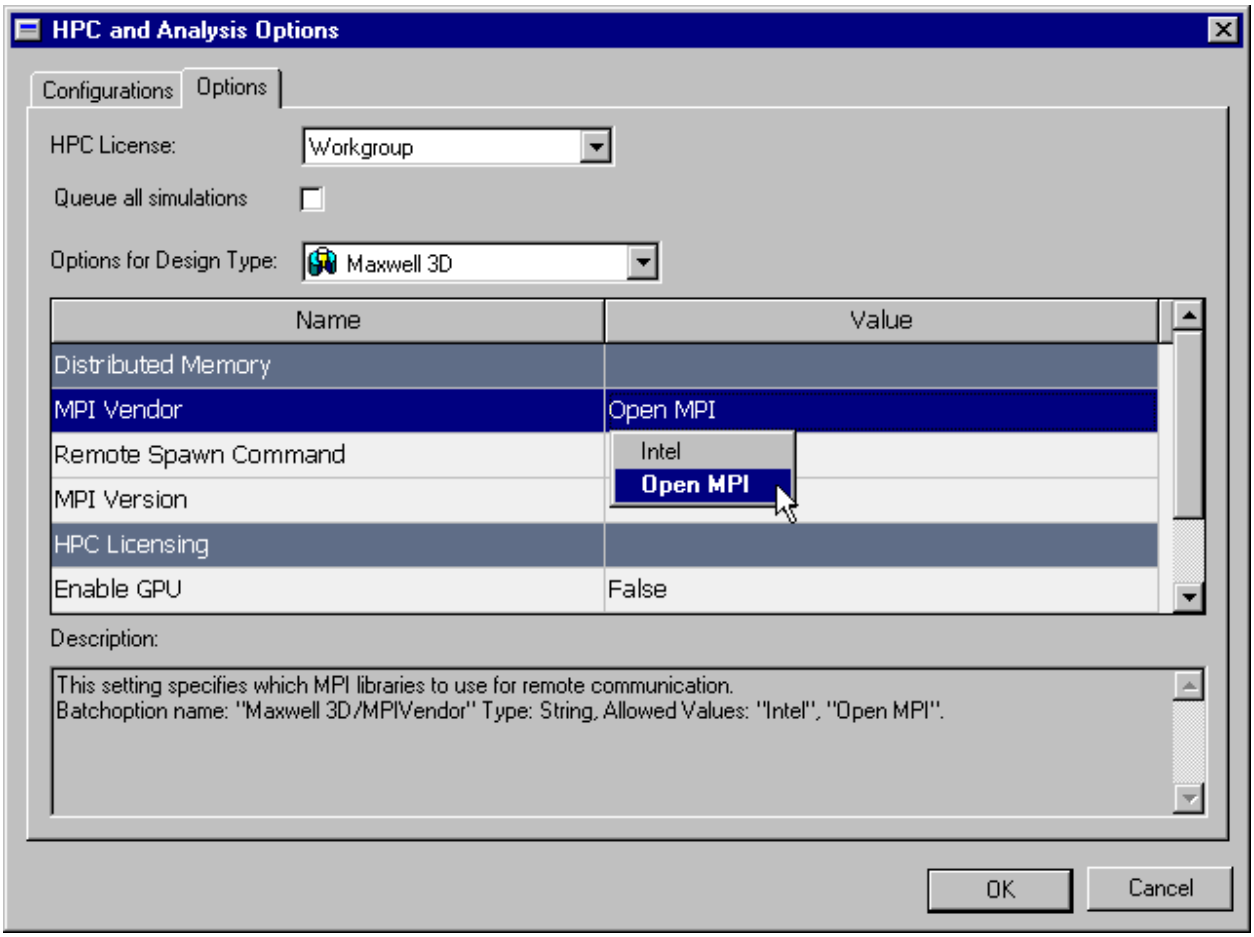
To choose Open MPI as the MPI Vendor:

1. Open the project in the Ansys Electronics Desktop application (interactive mode).
2. From the **Simulation** ribbon tab, click **HPC Options**
3. In the *HPC and Analysis Options* dialog box, select the **Options** tab.

   The contents of this tab vary by design type.

4. Select **Open MPI** from the **MPI Vendor** drop-down menu:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Click **OK**.

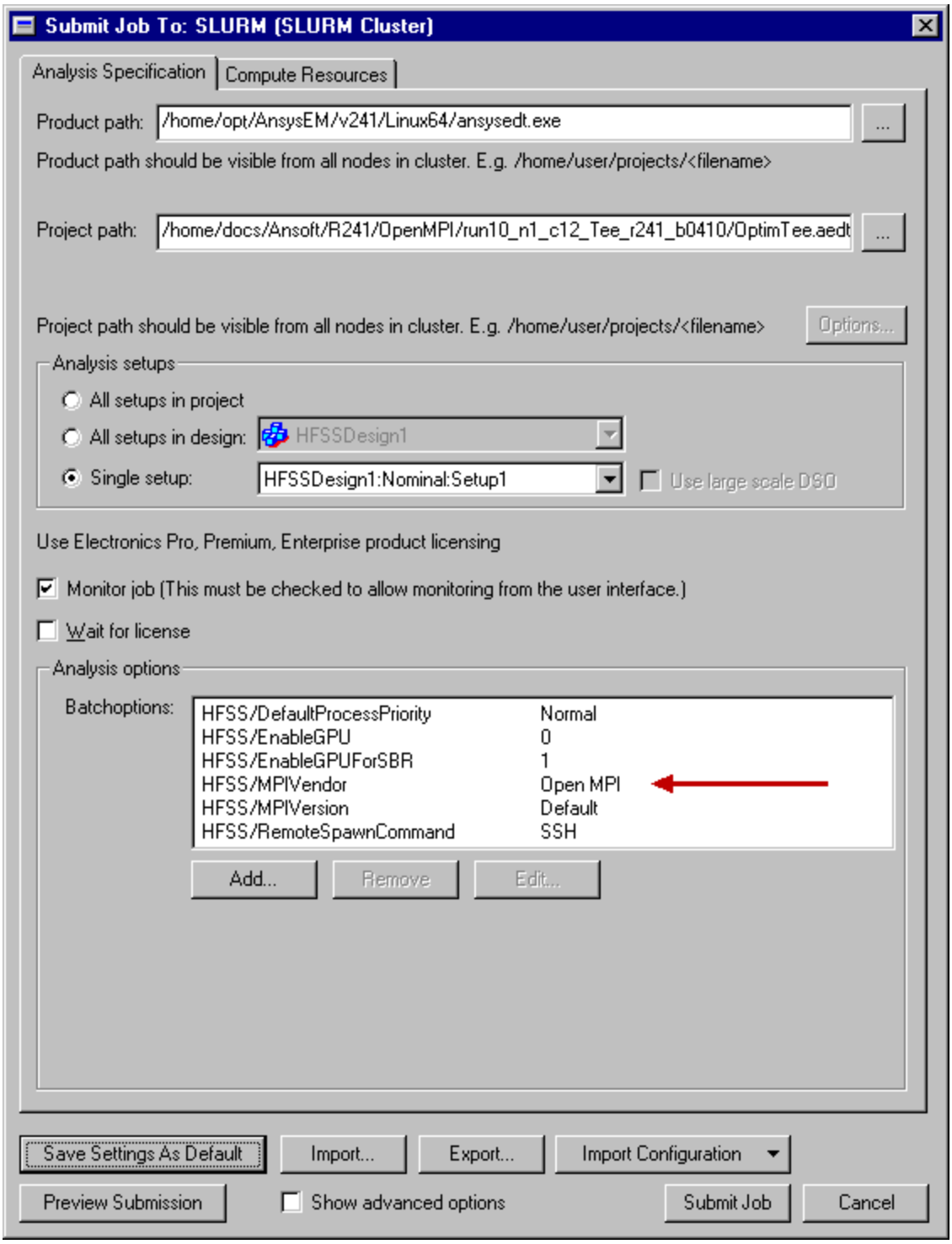
   You can then run the simulation on a Linux cluster as follows.

6. In the Project Manager, right-click the solution setup to solve and choose **Submit Job** from the shortcut menu.
7. Specify the desired parameters and click **Submit Job**.

   In the *Submit Job* dialog box, *Open MPI* is included as a batch option for each supported design type along with other applicable batch options.

   The following example is for an HFSS design:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Scheduler Proxy Interfaces

Scheduler proxy supports following new graphical interface functions. The scheduler specific prefix of each function is not shown in this listing.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

void Initialize(const std::string& config):

Initialize the proxy library for scheduler interaction. The **config** argument contains scheduler specific initialization information.

int CheckEnvironment(std::string& msg):

Check the environment in which the proxy library is running.

- Returns 0 (success) if the environment is appropriate for submitting jobs to the scheduler.
- Returns a non-zero error code if the environment is incorrect. If a non-zero error code is returned, an error message to display to the user is written to the msg argument.

int GetSchedulerInfo(std::string& msg, std::string& schedulerName, std::string& schedulerDescription, std::string& schedulerVersion):

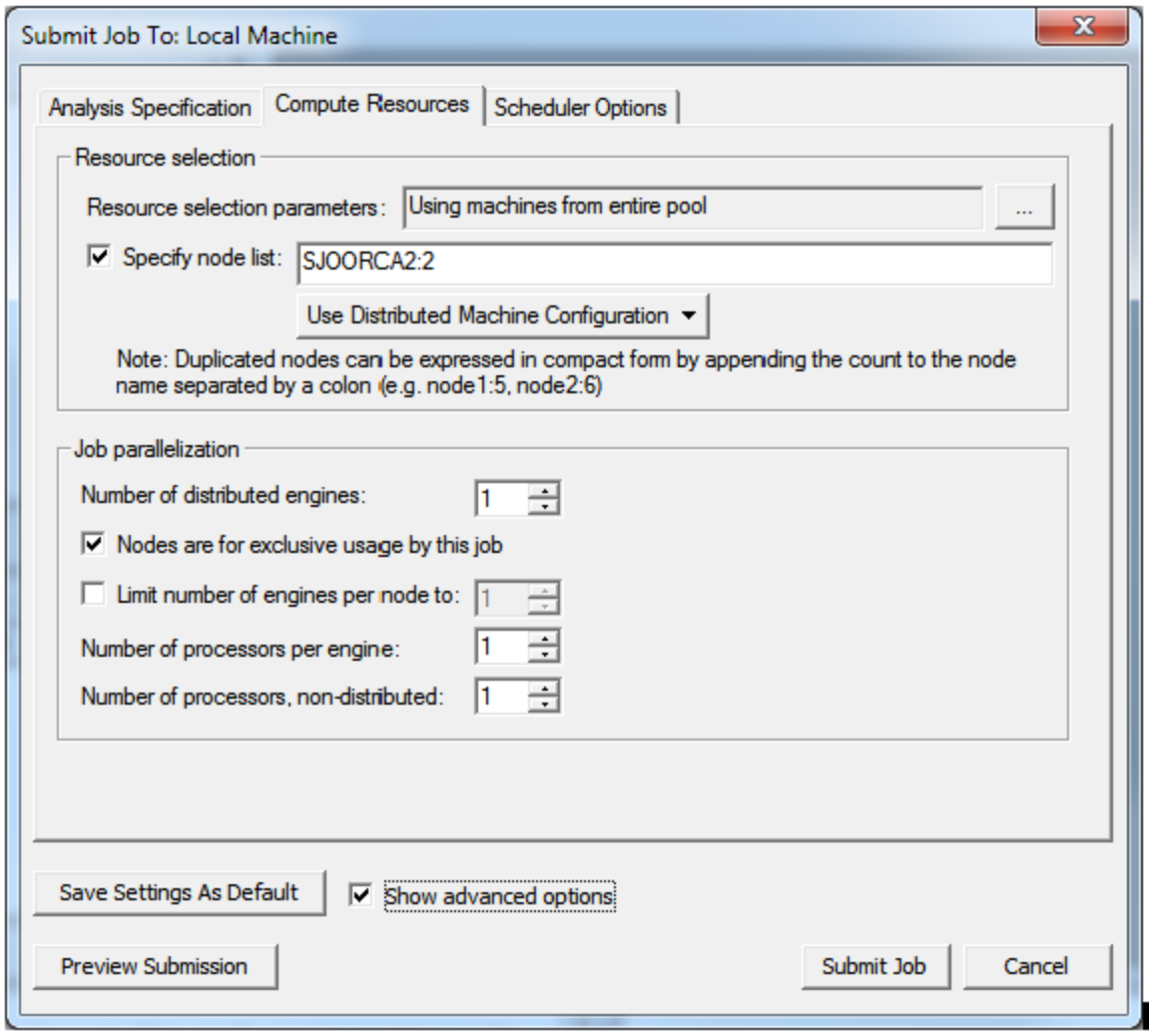
This function returns some basic information about the scheduler with which the scheduler proxy library interacts.

- On success, 0 is returned, and the scheduler name, scheduler description, and scheduler version are written to the **schedulerName**, **schedulerDescription** and **schedulerVersion** arguments.
- On failure, a non-zero error code is returned, and an error message to display to the user is written to the msg argument.

int GetComputeResourceAttributes(std::string& msg, AttributeDefinitionsStruct& attributeDefs):

The *Compute Resource* tab or the *Submit Job To* dialog box allows the user to specify scheduler specific resources. This function returns the information used to create and populate the *Compute Resource* tab.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Each line in the dialog box is defined by a single attribute definition in the **attributeDefs** argument. An attribute definition defines the name and description of an attribute, as well as information about the allowed values and the default value. In general, only the most commonly specified job attributes are included in the **attributeDefs** argument.

- On success, 0 is returned, and the attribute definitions are written to the **attributeDefs** argument.
- On failure, a non-zero error code is returned, and an error message to display to the user is written to the msg argument.
- If the scheduler proxy library does not support any attributes using this approach, the **attributeDefs** argument will contain no attribute definitions, and 0 will be returned.

int AbortJob(std::string& msg, const std::string& jobID, bool force, const SubmissionUserStruct& submissionUser):

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This function requests the scheduler to abort a job identified by the **jobID** argument. If the force argument is true, then errors should be ignored (the exact behavior is scheduler specific). The **submissionUser** argument contains information about the client user (the user running the Desktop process). The request to abort the job should run in the context of this user. If no user is specified, then the request to abort the job runs as the user of the process or thread running the function.

- If the request is successfully submitted, then 0 is returned.
- If there is an error, then a non-zero error code is returned, and an error message to display to the user is written to the msg argument.

int SubmitUniformJob(std::string& msg, std::string& jobID, const CmdLineStruct& cmdLineInfo, const JobParallelizationStruct& jobParallelization, const UniformComputeResourcesStruct& computeResources, const JobOptionsStruct& jobOptions, const JobAtributesStruct& jobAttributes, const SubmissionUserStruct& submissionUser,const IJobParameters* jobParametersCB):

This function submits a job to the scheduler.

- On success, 0 is returned, and the job identifier of the newly submitted job is written to the **jobID** argument.
- On failure, a non-zero error code is returned, and an error message to display to the user is written to the msg argument.

This function is used to submit jobs to the scheduler in which the resources allocated to the job are uniformly distributed across the nodes allocated to the job. All other arguments are input arguments, and they are described below:

The **cmdLineInfo** argument contains the command line arguments. The first argument is the command name.

The **jobParallelization** argument contains information on how the job should be parallelized. It contains the following integral parameters:

- the total number of distributed engines,
- the number of cores to allocate for each distributed engine,
- the maximum number of engines to allocate to a single node (optional), and
- the number of cores to allocated for the non-distributed portion of the analysis.
- It also contains a boolean parameter indicating whether nodes used for this job should be exclusively allocated to this job.

The **computeResources** argument is a reference to an object of type **UniformComputeResourcesStruct**. This **struct** contains zero or more resource attribute settings for the job. Each resource attribute setting consists or a resource name and a resource value. The resource name is the name of one of the resources defined in the **AttributeDefinitionsStruct** filled in by the **GetComputeResourceAttributes()** function. The

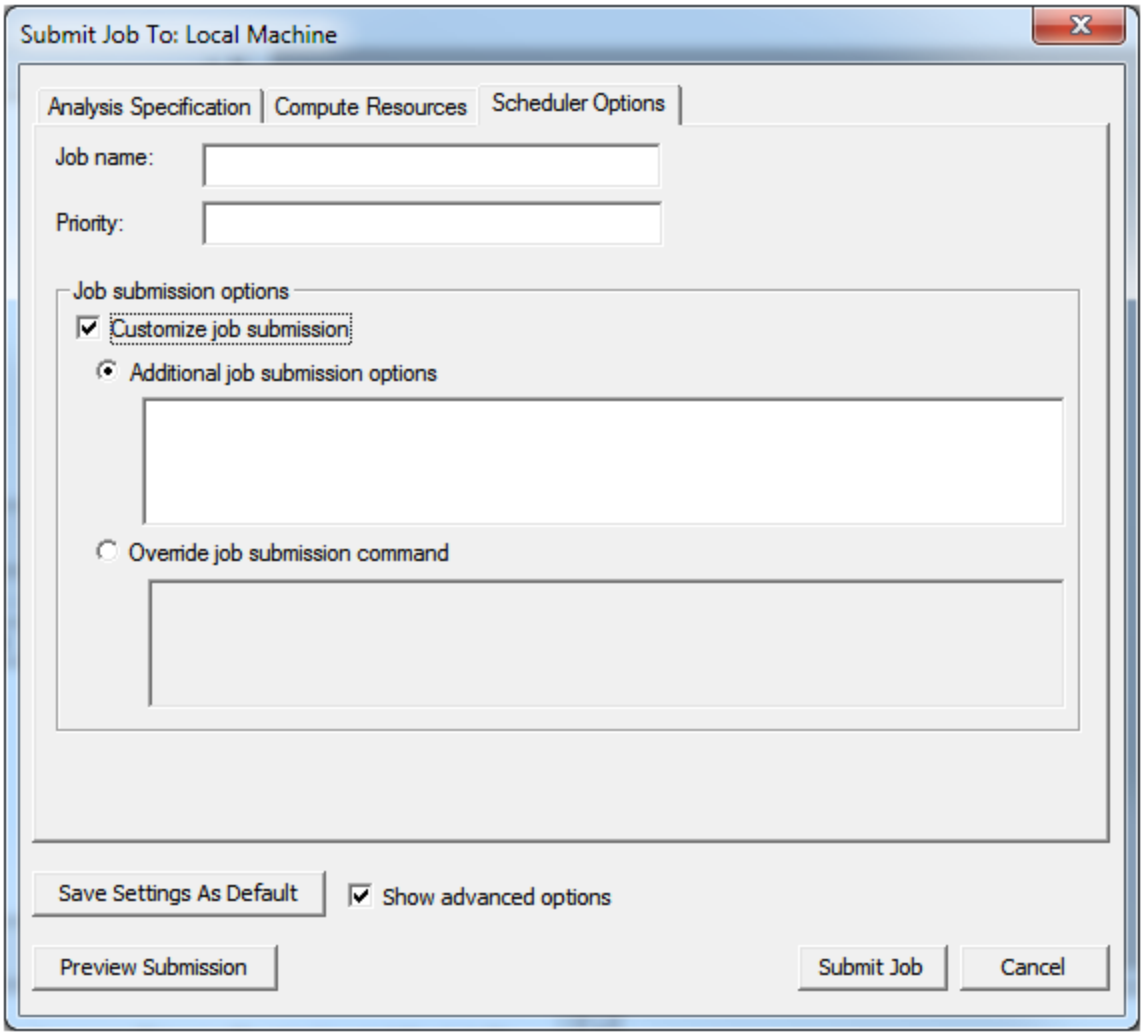
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

resource attribute value is the value specified for the resource attribute by the user using the *Compute Resource* tab of the *Submit Job To* dialog box. If no resource attributes are specified by the user in this dialog box, then the **computeResources** argument will contain no resource attribute settings.

The **jobOptions** argument contains the environment variable settings for the job.

The **jobAttributes** argument contains job submission attributes which are not necessarily related to the compute resources allocated to the job. The job name and the requested job priority are included in this data structure.

The *Scheduler Options* tab of the *Submit Job To* dialog box allows the user to either specify additional job submission options or to specify all submission options, replacing the settings from the other *Submit Job To* dialog box controls.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The user-specified submission options are included in this data structure, as well as a boolean setting indicating whether the user specified options are in addition to the automatically generated options, or whether they replace the automatically generated submission options.

The **submissionUser** argument contains information about the client user (the user running the Desktop process). The job is submitted to the scheduler to run as this user.

The **jobParametersCB** argument is a pointer to an object that implements the **IJobParameters** interface. This interface allows the scheduler proxy library to get additional information about the job. Specifically, the **GetWorkingDirectory()** interface function returns the working directory to be used for the job.

The **cmdLineInfo** argument contains the command line arguments. The first argument is the command name.

int SubmitNonUniformJob(std::string& msg, std::string& jobID, const CmdLineStruct& cmdLineInfo, const JobParallelizationStruct& jobParallelization, const NonUniformComputeResourcesStruct& computeResources, const JobOptionsStruct& jobOptions, const JobAtributesStruct& jobAttributes, const SubmissionUserStruct& submissionUser,const IJobParameters* jobParametersCB):

This function submits a job to the scheduler.

- On success, 0 is returned, and the job identifier of the newly submitted job is written to the **jobID** argument.
- On failure, a non-zero error code is returned, and an error message to display to the user is written to the msg argument.

This function is used to submit jobs to the scheduler in which the nodes to use and the number of engines to run on each node are specified by the user. All other arguments are input arguments, as for the **SubmitUniformJob()** function. These input arguments are the same as for the **SubmitUniformJob()** function, except that the **computeResources** argument is a reference to a **NonUniformComputeResourcesStruct**, as described below:

The **computeResources** argument is a reference to an object of type **NonUniformComputeResourcesStruct**. This object contains a vector of pairs, where each pair consists of the name of a node in the cluster, and the number of engines to run on the node.

int PreviewUniformJob(std::string& msg, std::string& preview, const CmdLineStruct& cmdLineInfo, const JobParallelizationStruct& jobParallelization, const UniformComputeResourcesStruct& computeResources, const JobOptionsStruct& jobOptions, const JobAtributesStruct& jobAttributes, const SubmissionUserStruct& submissionUser, const IJobParameters* jobParametersCB):

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This function is similar to the **SubmitUniformJob()** function, but instead of submitting the job, text representing how the job will be submitted is written to the preview argument. Typically the preview text includes the job submission command and the contents of the job script created for the job. For some schedulers, this content may not be meaningful, so the text returned could be different.

- On success, 0 is returned, and the job preview text is written to the preview argument.
- On failure, a non-zero error code is returned, and an error message to display to the user is written to the msg argument.

The other arguments are input arguments with the same meaning as for the **SubmitUniformJob()** function. The **submissionUser** argument is ignored for this function.

Testing Scheduler Integration

One way to test these functions is to run the analysis for an Ansys EM product in batch mode. When running in batch mode, a batch log file is created in the same directory as the project file. The batch log file has the same base name is the project file, with an extension of ".log". For example, if the project file is TestProject123.aedt, then the batch file is TestProject123.log. The batch log file contains useful information about the analysis run.

See the product specific help for details on running the product in batch mode, and for the command line options to use for distributed analysis.

- Testing IsProductLaunchedInYourEnvironment
- Testing GetSchedulerDisplayName and GetThisJobID
- Testing GetTempDirectory
- Testing GetMachineListAvailableForDistribution
- Testing LaunchProcess
- Testing GetUseRsmForEngineLaunch

Testing IsProductLaunchedInYourEnvironment

This function should be tested first. If the Ansys EM application is not able to load and run this function, or if it returns false, then none of the other functions will be called. If the batch analysis is running in a scheduler environment, and this function returns true, then there will be an "info" message near the beginning of the batch log indicating that the analysis is running as a scheduler job. This message will include the scheduler display name returned by the function GetSchedulerDisplayName, and it will also include the job ID returned by the function GetThisJobID. If the batch analysis is not running in a scheduler environment, then none of the messages will include a scheduler display name or job ID.

If this message does not appear when running in a scheduler environment, ensure that the scheduler proxy library is named correctly, that it is built correctly, that it is installed in the correct

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

directory, and that the function name prefix is the same is the library prefix converted to upper case.

Testing GetSchedulerDisplayName and GetThisJobID

As described above, when running a batch job in a scheduler environment, the scheduler display name and the job ID will appear in an "info" message near the beginning of the batch log. The values returned by these functions are copied to this message verbatim, so they can be directly compared to the expected values.

Testing GetTempDirectory

Many schedulers create a temporary directory for each job and delete the directory after the job finishes. One way to verify that this function is working correctly is to determine the pathname that the scheduler uses for the temporary directory and to monitor the contents of the temp directory as the job is running. If the analysis engines write files to this directory as the job runs, then this function is working.

Testing GetMachineListAvailableForDistribution

This function is used for distributed analysis. The analysis may be distributed across several machines if portions of the analysis are independent. For example, frequency sweeps, parametric analysis and domain decomposition allow different portions of the analysis to be distributed across machines. The analysis in a batch job will be distributed to multiple processors or hosts if the analysis includes a setup that may be distributed (e.g., frequency sweep, parametric analysis) and the **-Distributed** option is included in the desktop command line. The list of machines is displayed in an "info" message near the beginning of the batch log. The list in the info message can be directly compared to the expected list of machines.

To verify that the machine list is constructed correctly for a variety of cases, it may be necessary to test several jobs with different resource requirements and verify that the machine list is correct in each case. For example, one may run batch analyses with the following resource requirements:

- One processor on one host
- Several processors on one host
- One processor on each of several hosts
- Several processors on each of several hosts

Testing LaunchProcess

This function is used to launch analysis engines in the case where the analysis is distributed across multiple hosts. The analysis may be distributed across several machines if portions of the analysis are independent. For example, frequency sweeps, parametric analysis and domain decomposition allow different portions of the analysis to be distributed across machines. The

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

analysis in a batch job will be distributed to multiple processors or hosts if the analysis includes a setup that may be distributed (e.g., frequency sweep, parametric analysis) and the **-Distributed** option is included in the desktop command line. The list of machines is displayed in an "info" message near the beginning of the batch log. The batch log may also contain info messages when portions of the analysis distributed to different machines start or finish. These messages usually include the name of the host when the analysis ran or will run. One can verify that the analysis is actually running on the expected host or hosts using the Linux ps command or the Windows Task Manager.

In general, one analysis engine is started for each occurrence of each host in the list of machines available for distribution. For example, if the list of hosts is "hostA hostA hostA hostB hostB", then a total of five engines would be started, three on hostA and two on hostB. In some cases, an additional engine is started to perform the portion of the analysis which is not distributed; if this is the case, the non-distributed engine is idle during the portion of the analysis which is distributed. If this occurs in the case where the list of hosts is "hostA hostA hostA hostB hostB", then a total of six engines would be started, but at most five engines would be active at any given time. When each analysis engine is running, it may start additional child processes to do a portion of the analysis, but these are not counted as additional analysis engines because the parent of the sub-engine is inactive (waiting for the sub-engine results) when the sub-engine is active.

Testing should be sufficient to demonstrate that the scheduler proxy library can start multiple engine processes on the desktop host, and can also start multiple engine processes on other hosts.

Testing GetUseRsmForEngineLaunch

In most cases, this function will not be implemented or tested. If this function is implemented and returns true, then the Ansys Electromagnetics desktop application will not start the analysis engines using the LaunchProcess function directly. Instead, the Ansys Electromagnetics desktop application will start one AnsoftRSMService process on each host using the LaunchProcess function, and the engine processes will be started by these AnsoftRSMService processes. One may check for these processes using the Linux ps command or the Windows Task Manager. One AnsoftRSMService process should run on each host. These processes will be named ansoftrsmservice.exe or AnsoftRSMService.exe. These processes will be started on each host before any analysis engine is started on the host, and will remain running until the job is complete.

Troubleshooting Custom Scheduler Integration

- None of the Proxy Functions are Called
- Troubleshooting IsProductLaunchedInYourEnvironment Function
- Troubleshooting GetSchedulerDisplayName
- Troubleshooting GetThisJobID

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Troubleshooting GetTempDirectory
- Troubleshooting GetMachineListAvailableForDistribution
- Troubleshooting LaunchProcess
- Troubleshooting GetUseRsmForEngineLaunch

None of the Proxy Functions are Called

There are several problems which could result in none of the proxy functions being called.

The scheduler proxy library must be installed in the schedulers subdirectory of the Ansys Electronics Desktop installation directory.

The scheduler proxy library name must match "*_scheduler.dll" on Windows and "lib*_scheduler.so" on Linux. If the library name does not match this format, then the library will not be loaded. In addition, the function name prefix must be the same as the library name prefix converted to upper case. For example, if the library name prefix is "abc", then the function name prefix is "ABC". In this example, the library name is "abc_scheduler.dll" on Windows, and "libabc_scheduler.so" on Linux. In this example, the full name of the IsProductLaunchedInYourEnvironment function is ABC_IsProductLaunchedInYourEnvironment on Windows and Linux, and it must have extern "C" linkage.

Verify that the compile and link flags follow the guidelines in the section "Build Information", above. Incorrect compile or link flags may prevent the library from being loaded by the Ansys Electromagnetics product.

If there is a problem with calling the IsProductLaunchedInYourEnvironment function, then none of the other functions will be called. The other functions are only called if the IsProductLaunchedInYourEnvironment function is successfully called and returns true.

Troubleshooting IsProductLaunchedInYourEnvironment Function

Verify that the conditions specified in the section "None of the Proxy Functions are Called" are met.

Verify that this function returns true when called in an environment running under the scheduler, and that it returns false when called in an environment not running under the scheduler.

Troubleshooting GetSchedulerDisplayName

Verify that the IsProductLaunchedInYourEnvironment function returns true when running in the scheduler environment.

Verify that the scheduler display name is a valid ASCII string.

Verify that, if argument buffer is NULL, then the required length of the buffer is stored in the location to which argument length points, and true is returned. The required buffer length must include space for the string null terminator.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Verify that, if argument buffer is not NULL and the value to which argument length points (the buffer length) is large enough to contain the display name, including the terminal null byte, then the display name is copied to the buffer and true is returned.

Troubleshooting GetThisJobID

Verify that the IsProductLaunchedInYourEnvironment function returns true when running in the scheduler environment.

Verify that the job ID is a valid ASCII string.

Verify that, if argument buffer is NULL, then the required length of the buffer is stored in the location to which argument length points, and true is returned. The required buffer length must include space for the string null terminator.

Verify that, if argument buffer is not NULL and the value to which argument length points (the buffer length) is large enough to contain the job ID, including the terminal null byte, then the job ID is copied to the buffer and true is returned.

Troubleshooting GetTempDirectory

Verify that the IsProductLaunchedInYourEnvironment function returns true when running in the scheduler environment.

Verify that the temporary directory name is a valid ASCII string.

Verify that, if argument buffer is NULL, then the required length of the buffer is stored in the location to which argument length points, and true is returned. The required buffer length must include space for the string null terminator.

Verify that, if argument buffer is not NULL and the value to which argument length points (the buffer length) is large enough to contain the temporary directory pathname, including the terminal null byte, then the temporary directory pathname is copied to the buffer and true is returned.

Troubleshooting GetMachineListAvailableForDistribution

Verify that the IsProductLaunchedInYourEnvironment function returns true when running in the scheduler environment.

Verify that the list of hosts is a valid ASCII string containing a space separated list of host names. A host name will appear in the list a number of times equal to the number of processors or cores available to the job on that host.

Verify that, if argument buffer is NULL, then the required length of the buffer is stored in the location to which argument length points, and true is returned. The required buffer length must include space for the string null terminator.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Verify that, if argument buffer is not NULL and the value to which argument length points (the buffer length) is large enough to contain the list of hosts, including the terminal null byte, then the list of hosts is copied to the buffer and true is returned.

Troubleshooting LaunchProcess

Verify that the IsProductLaunchedInYourEnvironment function returns true when running in the scheduler environment.

The hostName argument is a host name from the list returned by the GetMachineListAvailableForDistribution function. Verify that the LaunchProcess function can accept host names in the format returned by the GetMachineListAvailableForDistribution function.

The exePathName argument is the pathname of the analysis engine executable to be started. This pathname may contain spaces or other characters special to the shell. Ensure that the LaunchProcess function is able to handle such cases.

The arg1 and arg2 arguments may contain newlines, tabs, single quotes, spaces, dollar signs, and other characters which may be special to the shell. Ensure that the LaunchProcess function is able to handle such cases. If needed, the newline characters may be replaced by other whitespace characters. One or both of these arguments could also be an empty string; verify that the empty string is correctly passed to the engine process command line.

If a scheduler command is used to launch the engine process on a remote machine, the engine command line may be processed by the shell twice, once when the scheduler command is processed by the shell, and again when the engine command is processed by the shell. In such cases, the quoting of characters special to the shell will need to be take these two passes through the shell into account. In some implementations, it may be necessary or convenient to use different approaches for launching engine processes on the local machine and on remote machines; if this is done, verify that the approach used to determine whether the hostName argument represents the local machine is correct.

Troubleshooting GetUseRsmForEngineLaunch

In most cases, this function will not be implemented. If it is implemented, then follow the suggestions below.

Verify that the IsProductLaunchedInYourEnvironment function returns true when running in the scheduler environment.

If the RSM should be used for launching engines, verify that this function returns true. Otherwise, verify that this function returns false.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using an IronPython Program for Integration with a Scheduler

This section describes how to create an IronPython program for integration with a scheduler. Each such program is used for a single specific scheduler environment. If the program is installed with a valid name and in the correct location, then it will automatically be loaded and used by Ansys EM tools.

Installation Details

The IronPython program must be installed in the schedulers subdirectory of the Ansys EM installation directory. For example, if the installation directory is "C:\Program Files\ANSYS Inc\v252\AnsysEM", then the IronPython program must be installed in directory C:\Program Files\ANSYS Inc\v252\AnsysEM\schedulers".

The program file extension must be ".py". Select the program name so that it does not conflict with other IronPython programs in the same directory. If the Operating System or file system treat file names in a case sensitive manner, the file extension ".py" must be lower case.

Python Programming Notes

The scheduler program will be run in the IronPython environment both on Microsoft Windows and on Linux. There are some differences between IronPython and CPython. The version of IronPython in use is 2.7.0.40

Implementation Details

The program must contain the following:

Import the ISchedulerPluginExtension interface as follows:

```
from Ansys.Ansoft.SchedulerPluginDotNet import
ISchedulerPluginExtension
```

Define a class which implements the ISchedulerPluginExtension interface. In this document, this class is named SamplePluginExtension, but any class name may be used. The class member functions are described in the next section. The class definition will look similar to the following:

```
class SamplePluginExtension(ISchedulerPluginExtension):

def GetName(self):
return "SamplePluginExtension"

def GetDescription(self):
return "Example python script plugin extension"
. . .
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Include the following line in the program so that the class that you have defined, SamplePluginExtension, is loaded by the infrastructure:

```
ExtensionRegistrar.RegisterPluginExtension(SamplePluginExtension())
```

The infrastructure will make the EntensionRegistrar object available in the environment where the program is loaded.

Each of the functions to be implemented in the SamplePluginExtension class is described below.

- GetName
- GetDescription
- IsProductLaunchedInYourEnvironment
- GetSchedulerDisplayName
- GetThisJobID
- GetUseRsmForEngineLaunch
- GetTempDirectory
- GetMessageStringToRegisterForSigTerm
- GetMachineListAvailableForDistribution
- LaunchProcess

GetName [IronPython]

Purpose

Return a short string containing the name of the plugin extension. This string is used to identify the scheduler plugin extension in logs or program output.

Signature

```
GetName(self)
```

Arguments (excluding self)

None.

Return Value

Returns a string containing the name of the plugin extension.

> **Note:**
>
> The plugin extension name is generally a fixed string.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

GetDescription [IronPython]

Purpose

Return a string containing the description of the plugin extension. This string is used to identify the scheduler plugin extension in logs or program output.

Signature

```
GetDescription(self)
```

Arguments (excluding self)

None.

Return Value

Returns a string containing the description of the plugin extension.

> **Note:**
>
> The plugin extension description is generally a fixed string.

IsProductLaunchedInYourEnvironment [IronPython]

Purpose

Determine if the program is running in the context of the scheduler for which this program was written.

Signature

```
IsProductLaunchedInYourEnvironment(self)
```

Arguments (excluding self)

None.

Return Value

Returns True if the current process is running as a job of the scheduler. Otherwise, False is returned.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> For many schedulers, the presence of certain environment variables or their values may be checked to determine if the current process is running as a job of the scheduler.

GetSchedulerDisplayName [IronPython]

Purpose

Get a string identifying the scheduler associated with the current plugin extension. This string is used to identify the scheduler.

Signature

```
GetSchedulerDisplayName(self)
```

Arguments (excluding self)

None.

Return Value

Returns a string containing the description of the scheduler for which this plugin extension was written.

> **Note:**
>
> The scheduler display name is generally a fixed string.

GetThisJobID [IronPython]

Purpose

Get a string identifying the job currently running in the scheduler environment. This string is displayed to the end user to identify the job.

Signature

```
GetThisJobID(self)
```

Arguments (excluding self)

None.

Return Value

Returns a string containing the Job ID for the current job.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> For many schedulers, the job ID may be obtained from the value of an environment variable.

GetUseRsmForEngineLaunch [IronPython]

Purpose

For some schedulers, it may be desirable for the AnsoftRSM program to launch the engine processes instead of using the scheduler plugin extension directly. For example, if the plugin extension is limited to launching one process per host, then the plugin extension may be used to launch one AnsoftRSM executable per host, and the AnsoftRSM executable will launch all of the engine processes.

If AnsoftRSM should be used to launch engine processes for this scheduler, then this function shall return True.

If AnsoftRSM should not be used to launch engine processes for this scheduler, then this function shall return False.

Signature

```
GetUseRsmForEngineLaunch(self)
```

Arguments (excluding self)

None.

Return Value

Returns True if AnsoftRSM should be used to launch engine processes for this scheduler. Returns False if the plugin extension should be used to directly launch engine processes for this scheduler.

> **Note:**
>
> If this function returns True, then the plugin extension will directly launch only one process on each host.

GetTempDirectory [IronPython]

Purpose

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Get the pathname of the temporary directory provided by the scheduler for the current job. The pathname is an empty string if the scheduler does not provide a temporary directory for the current job.

Signature

```
GetTempDirectory(self)
```

Arguments (excluding self)

None.

Return Value

Returns a string containing the pathname of the temporary directory provided by the scheduler for the current job. Returns an empty string if no temporary directory is provided by the scheduler for the current job.

> **Note:**
>
> If the return value is an empty string, then the temporary directory specified on the command line or in the registry will be used.

GetMessageStringToRegisterForSigTerm [IronPython]

Purpose

Obsolete. This function should return an empty string.

Signature

```
GetMessageStringToRegisterForSigTerm(self)
```

Arguments (excluding self)

None.

Return Value

Returns an empty string.

> **Note:**
>
> This function should always return an empty string.

GetMachineListAvailableForDistribution [IronPython]

Purpose

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Get the names of the hosts allocated to the current job. A host name will appear in the output string multiple times if the scheduler has allocated multiple processors or cores on the host to the job. The number of times the host name appears in the string is equal to the number of processors or cores of the host that are allocated to the current job. The output is a text string containing the host names separated by space characters.

Signature

```
GetMachineListAvailableForDistribution(self)
```

Arguments (excluding self)

None.

Return Value

A string containing the names of the hosts allocated to the job, separated by space characterss. The number of times the host appears in the string is equal to the number of processors or cores of the host that are allocated to the current job.

> **Note:**
>
> The hostnames in the string provided by this function shall be used in calls to LaunchProcess(). The host names must be in a format that is accepted by that function. See the section below on LaunchProcess.

LaunchProcess [IronPython]

Purpose

Launch a local or remote process to run an analysis engine. This function is called by the Ansys Electromagnetics desktop application to launch an engine process on a specified host. The hostname is one of the names provided by the GetMachineListAvailableForDistribution function. See the GetMachineListAvailableForDistribution section above. If the hostname does not refer to the local host, then this function shall use the scheduler to launch the engine on the specified host. If the hostname refers to the local host, then the engine may be started as a child process, or it may be started using the scheduler.

Signature

```
LaunchProcess(self, hostName, exePathName, arg1, arg2)
```

Arguments (excluding self)

hostName: The name of the host where the process is to be launched.

exePathName: The pathname of the analysis engine executable to be started.

arg1: The first argument of the analysis engine command line.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

arg2: The second argument of the analysis engine command line.

Return Value

Returns 0 on success. Returns a non-zero value if an error occurs.

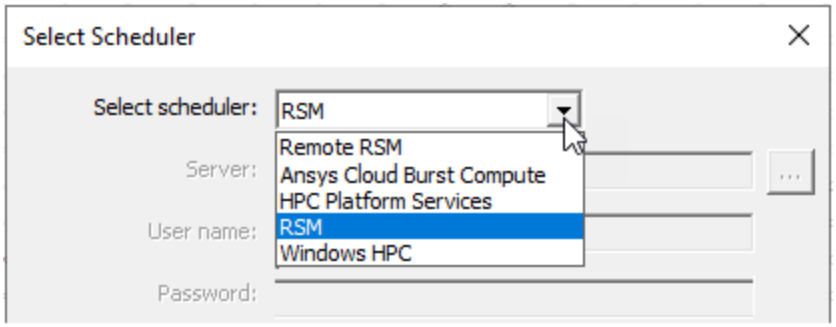
Note:

- The hostName argument will be one of the hostnames provided by the function GetMachineListAvailableForDistribution().
- If the hostName argument is the same as the current host, then the analysis engine process may be started as a child process. If the hostName argument is not the same as the current host, then the analysis engine process will be started on the remote host using the facilities available in the scheduler environment. The command line of the analysis engine process is exePathName arg1 arg2. The command line arguments arg1 and arg2 may contain spaces or other characters that are interpreted by the command processor, such as backslash (\), single quote (') or double quote (") characters, or dollar signs ($). If the analysis engine command is processed by a shell, then it may be necessary to quote any special characters in the exePathName or in the arguments so that the special meaning is removed. If a scheduler command is used to request the scheduler to launch the command to start the engine process, and that command is processed by a command shell, then the analysis engine command may be processed by the shell twice: once when the scheduler command is processed, and a second time when the analysis engine process is started. If this is the case, then the quoting of special characters needs to account for two passes through the command processor.
- The command line arguments arg1 and arg2 may be empty strings. These arguments must be preserved, even if they are empty strings. In some versions of the IronPython subprocess module, empty argument strings are discarded, resulting in an incorrect number of command line arguments. A workaround for this issue is to replace an empty string argument by a string consisting of a single space character.

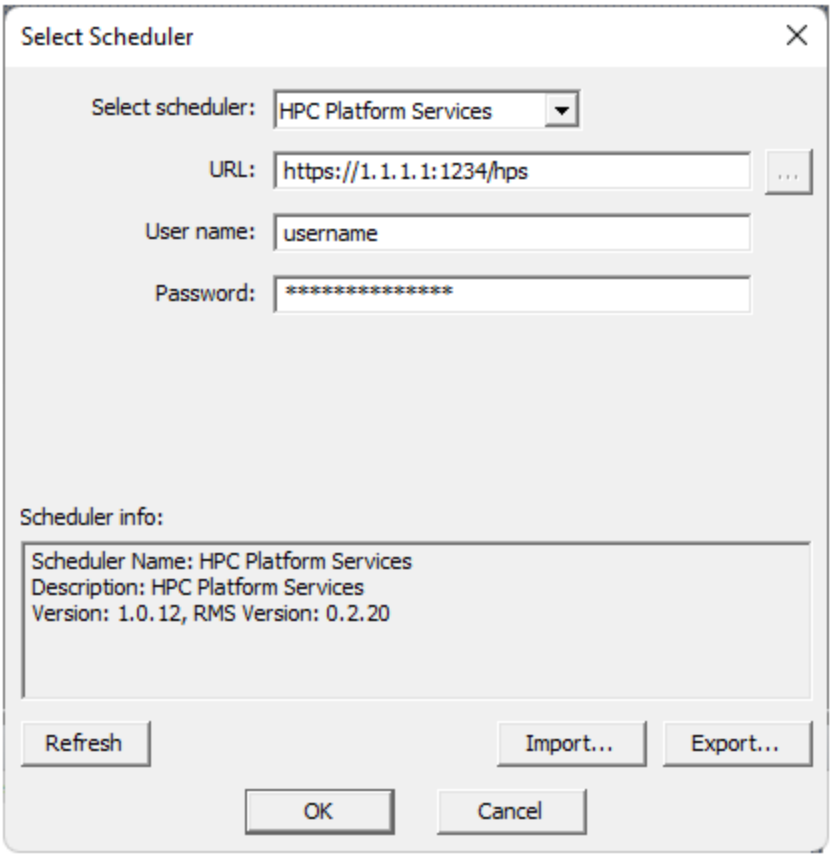
Ansys HPC Platform Services Support for HPC Job Management

Ansys Electromagnetics Desktop supports the Ansys HPC Platform Services mechanism that allows quick and easy access to HPC resources, to include Ansys Cloud Burst Computing. HPC Platform Services can be selected through both the **Simulation Ribbon (Scheduler** button) and **Tools > Job Management > Select Scheduler**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This opens the **Select Scheduler** window.



You must supply the URL for the HPC Platform Services. You must also enter the username and password for the HPC Platform Services account to be used for job submission. If you press the **Refresh** button, the versions of the HPC Platform Services server and the JMS (Job Management Service) are reported in this dialog.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Uploads to and downloads from Ansys HPC Platform Services use archives to improve speed. The Project browser shows both .aedt and .aedtz (archive) files. If you submit an unarchived project, an archive will be automatically created for upload. When the job completes, the results are packaged into an archive. When the download is initiated, the repackaged archive file is transferred to the download folder along with the other job-related files, such as the batch log.

Prerequisites for Using Ansys HPC Platform Services with Ansys Electronics Desktop

You must select HPC Platform Services as your Job Management Scheduler, including an endpoint URL, username, and associated password.

Job Management UI for Ansys HPC Platform Services

You can use Ansys Electronics Desktop to submit batch jobs to Ansys HPC Platform Services and monitor those jobs.

- To select HPC Platform Services as the scheduler, use **Tools** > **Job Management** > **Select Scheduler**.
- To submit a batch job to Ansys HPC Platform Services, use **Tools** > **Job Management** > **Submit Job**.

Ansys HPC Platform Services-Specific Settings:

The main use case for Ansys HPC Platform Services is where a SLURM cluster is used for computing at the endpoint.

Other schedulers could potentially be used in a compute cluster for Ansys HPC Platform Services.

When a job is submitted from AEDT, HPC Platform Services in turn submits a job to the compute cluster using the scheduler running on that cluster.

In case there are multiple queues (SLURM partitions) set up in the cluster, the **Compute Resources** tab has a drop-down list box for selecting the queue.

See your cluster administrator for the correct queue/partition name.

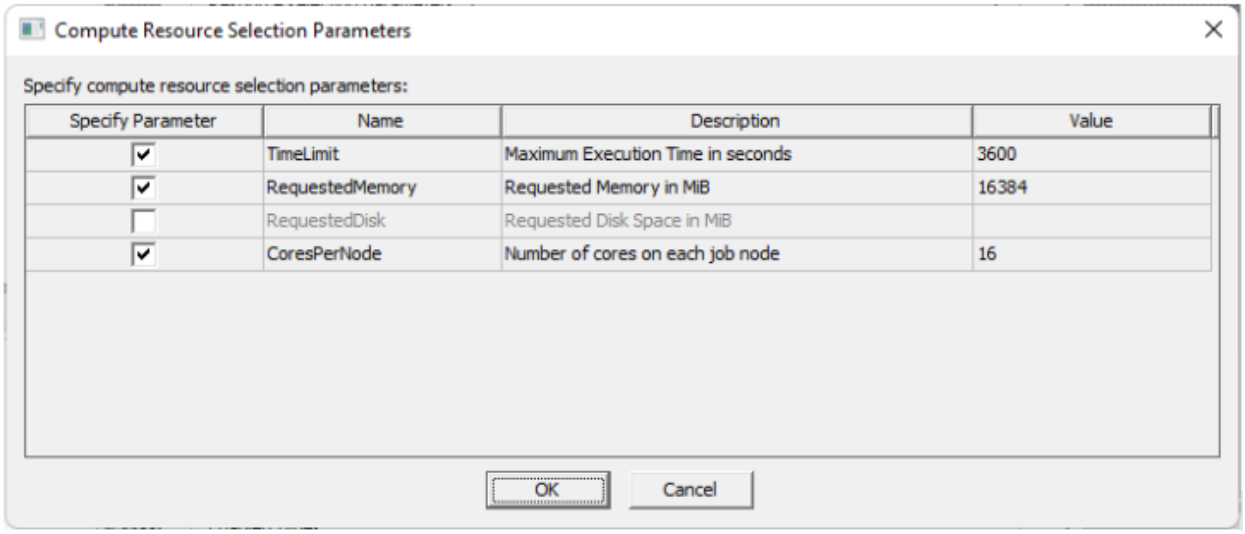
Compute Resources Tab

On the **Compute Resources** tab, **Resource Selection Parameters** allows you to select the following:

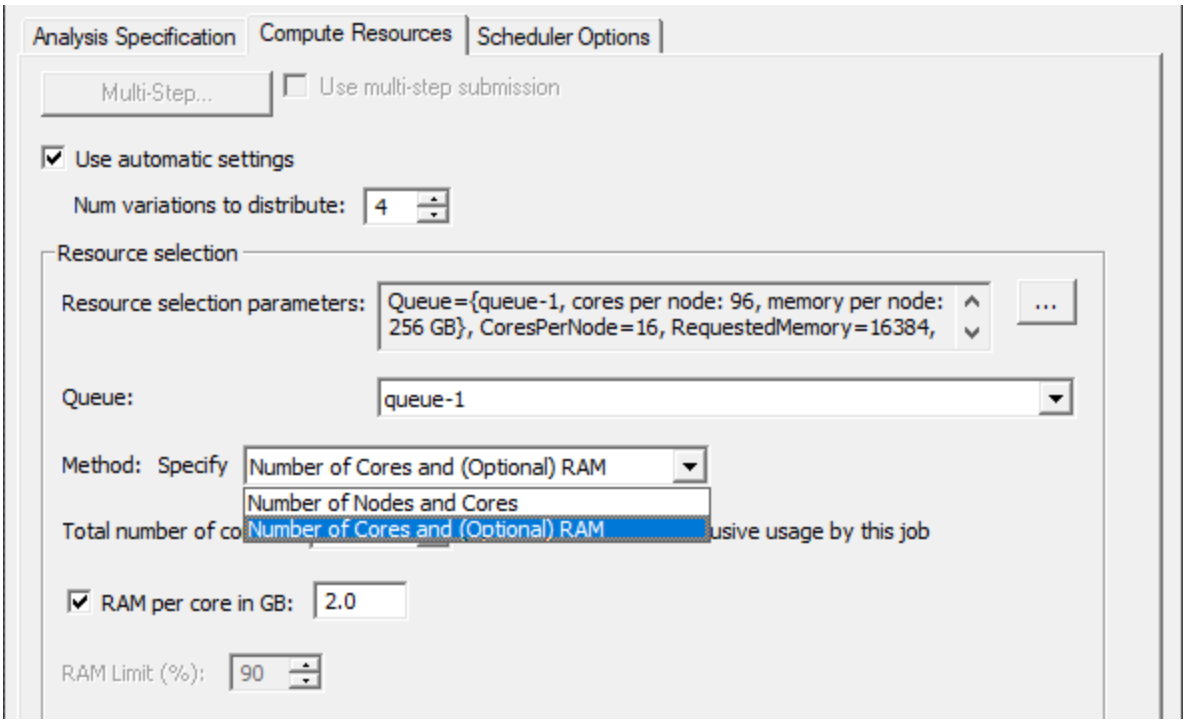
- **TimeLimit**: Maximum execution time for the job in seconds
- **RequestedMemory**: Requested memory for the job (all nodes combined) in megabytes

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **RequestedDisk**: Requested disk space for the job (all nodes combined) in megabytes
- **CoresPerNode**: The number of cores allocated to the job on each node



On the **Compute Resources** tab, if you select **Use automatic settings** with **Num variations to distribute** set to 1, Optimetrics variations will be solved sequentially. Other distribution types will be distributed automatically.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you check the **Use automatic settings** checkbox, then you may select one of the following methods for specifying job resources:

Specify Number of Cores and (Optional) RAM

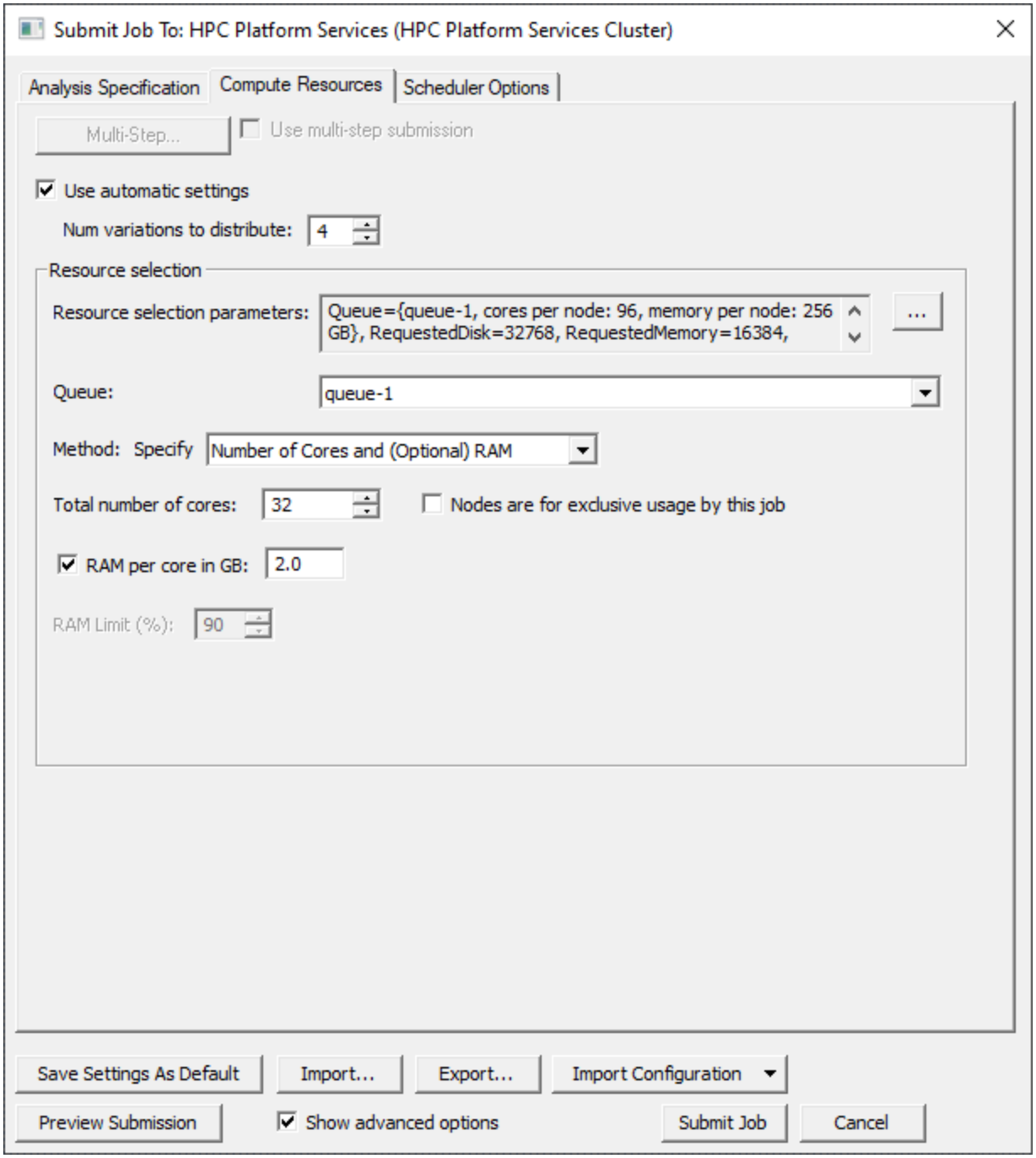
You must select the total number of cores for the job. If you check the associated checkbox is checked, you may also specify the RAM per core for the job. The total RAM requested will be the product of the RAM per core and the number of cores for the job.

Specify Number of Nodes and Cores

The job will request all cores on the specified number of nodes. The analysis will be limited to the specified number of cores; this may be useful to limit the licenses used by the job. This method may only be used if a queue is specified and if all queue execution nodes have the same number of cores. The number of cores specified must be less than or equal to the number of cores available on the specified number of nodes. If these requirements are not met, then the job will not be submitted, and an error message will be displayed.

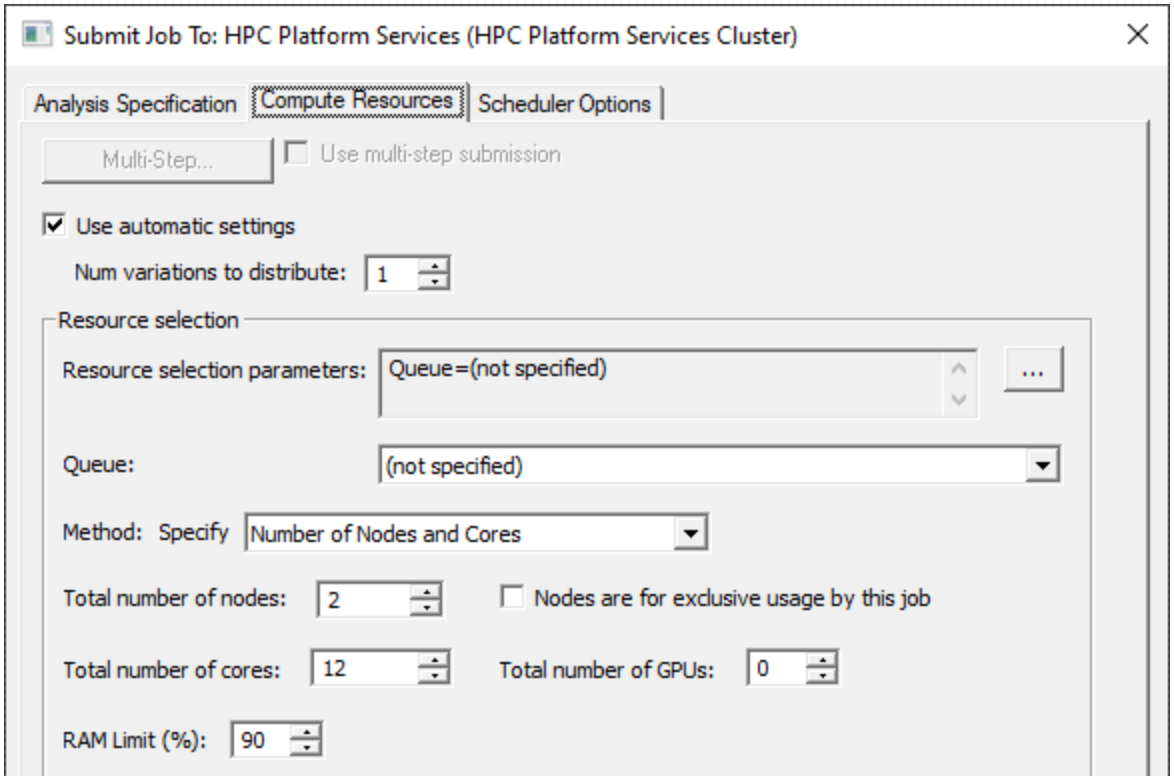
If you set **Num variations to distribute** to 2 or more, Optimetrics variations will be solved in parallel. Other distribution types will be distributed automatically.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you set **Num variations to distribute** to 1, Optimetrics variations will be solved in sequentially. Other distribution types will be distributed automatically.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

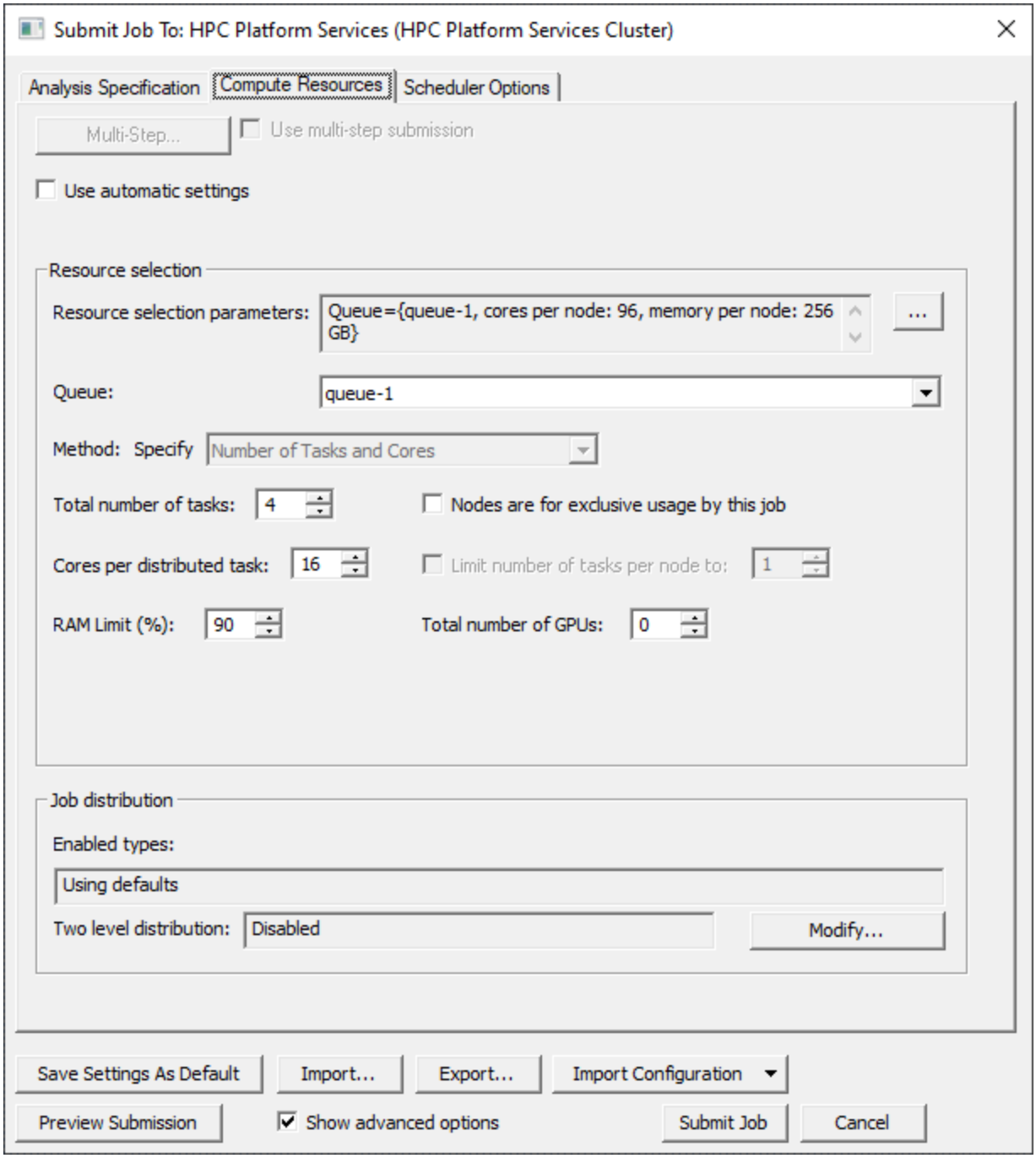


If you do not check **Use automatic settings**, then you may only select the following method for specifying job resources:

Specify Number of Tasks and Cores

You must specify the number of distributed tasks and the number of cores for each distributed task. The number of cores allocated to the job on each host must be an integral multiple of the number of cores per distributed task. If this requirement is not met, then the job will not start, and an error message will be output. In this case, you may specify a different submission method, or set the CoresPerNode resource selection parameter to an integral multiple of the number of cores per task and resubmit the job.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Use **Tools > Job Management > Monitor Job** to monitor the job's progress.

To get more detailed monitoring information, there is a **Job Management Service** that can be accessed from the base endpoint URL (without "/hps" at the end).

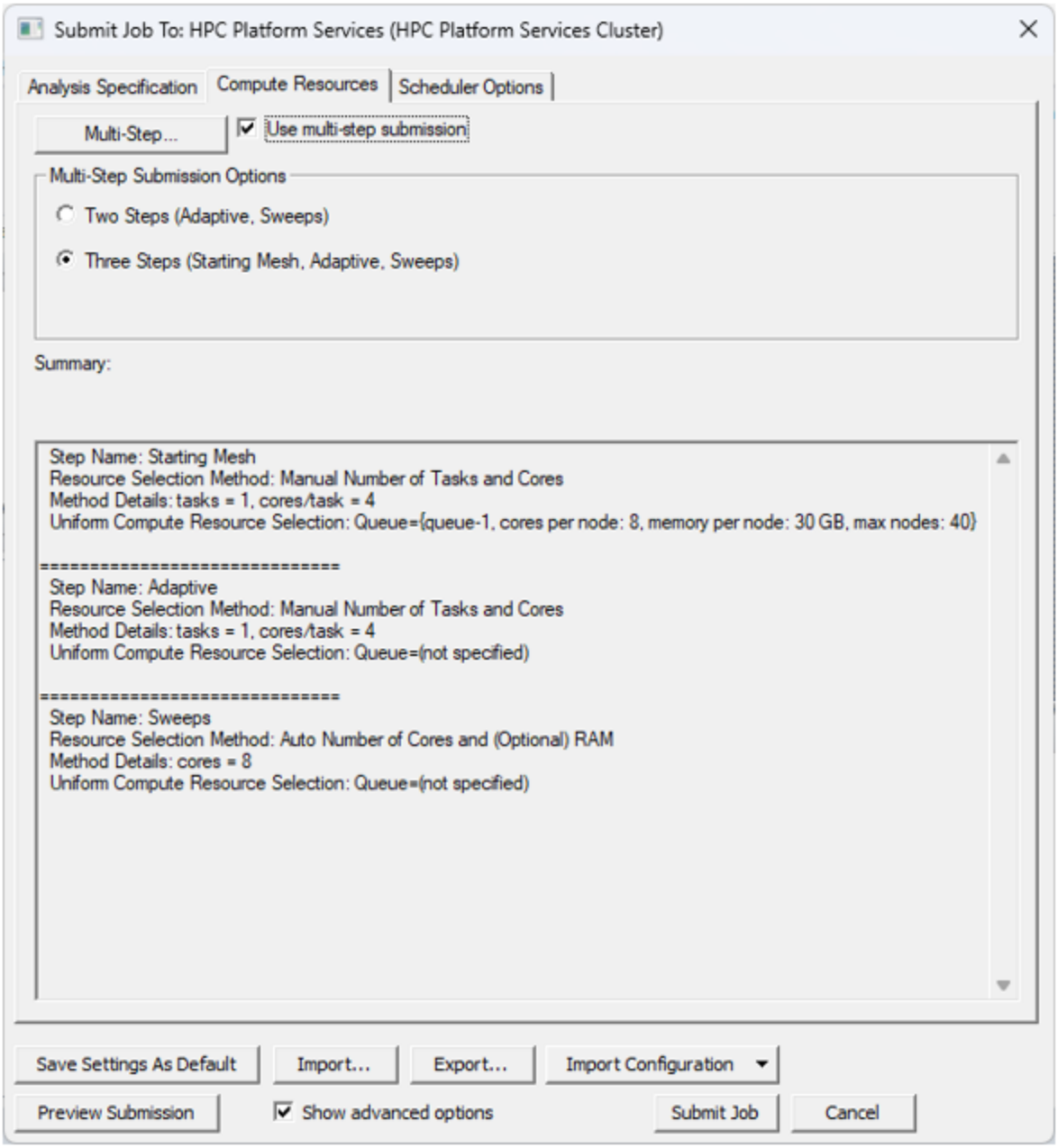
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you download results or logs from an Ansys HPC Platform Services multi-step job, the batchinfo, aedtexport, and other archives (e.g., for debug logs if debug logging is enabled), the folder names have a numeric suffix indicating the step index, rather than a step name. (The step index is 0 for the first step, 1 for the second step, and so on.)

Multistep Submissions

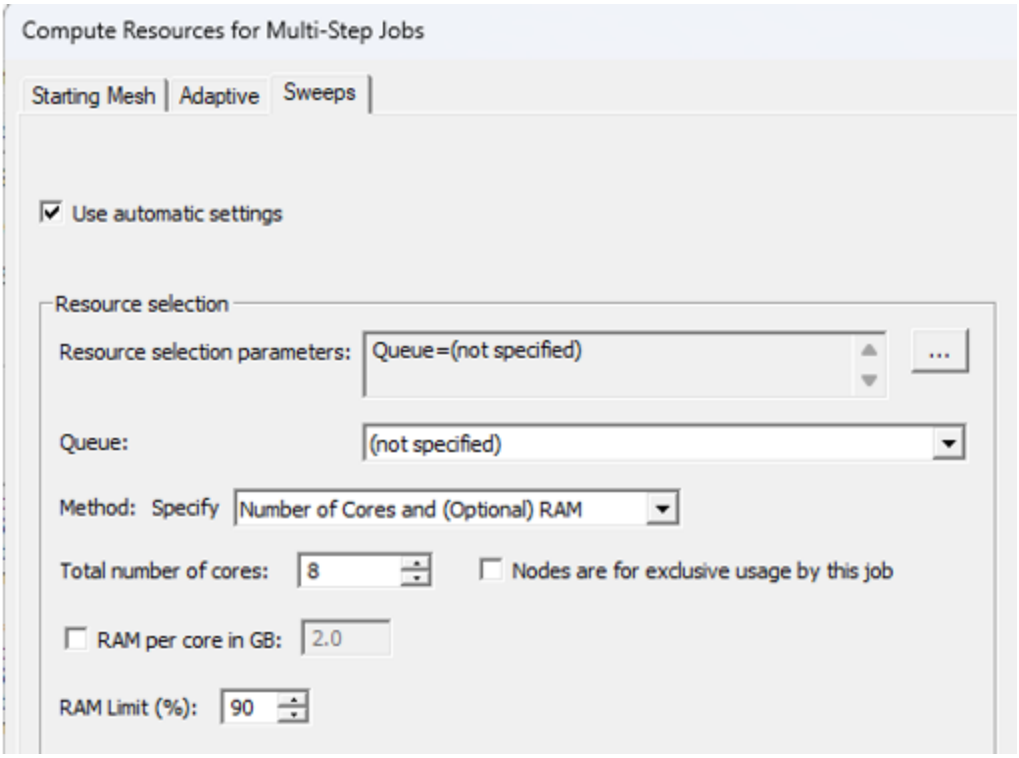
You have the option to define mult-step submissions. You can select Two Steps, consisting of Adaptive and Sweeps, or Three Steps, consisting of Starting Mesh, Adaptive, and Sweeps.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



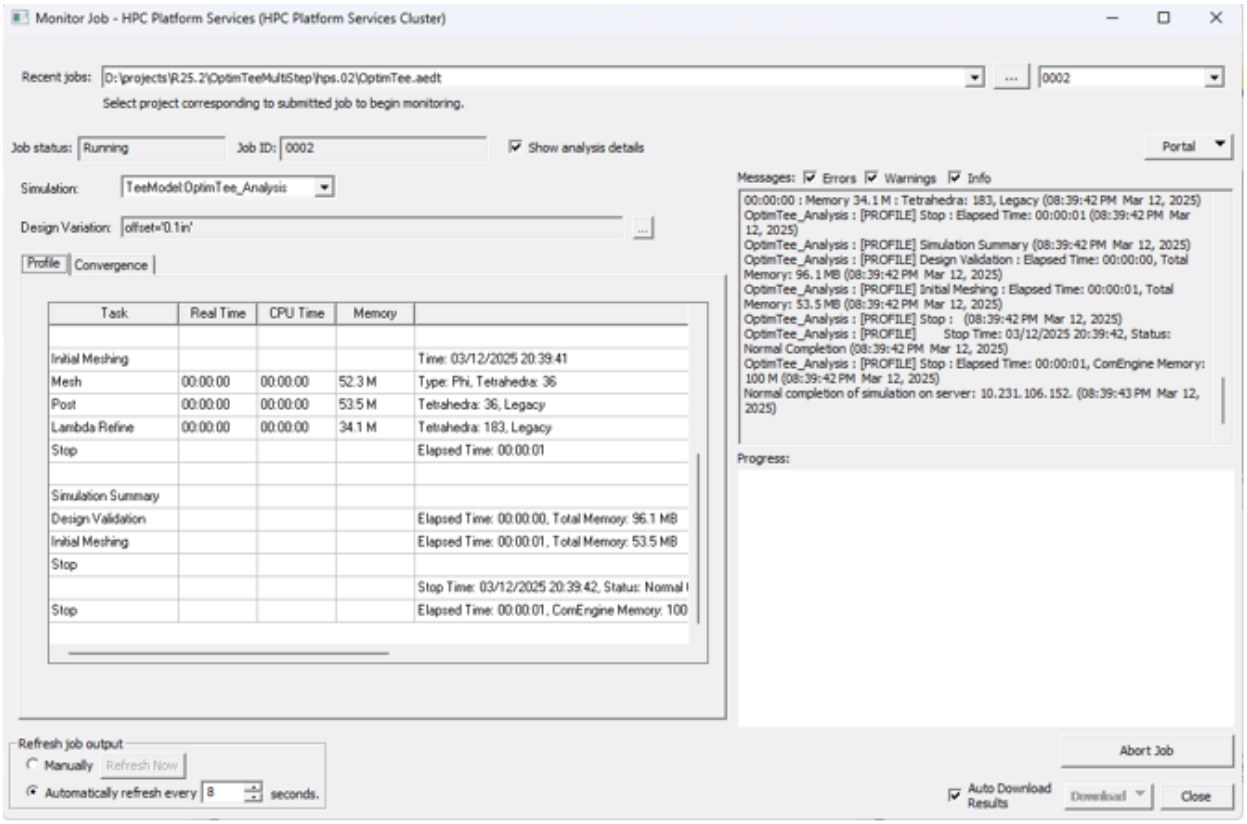
Selecting Use multi-step submission, lets you specify the number of steps. You then select **Multi-step ...** to use a dialog to specify accept Automatic settings, or to specify Resources for multi-step jobs. Tabs appear for the steps you define.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

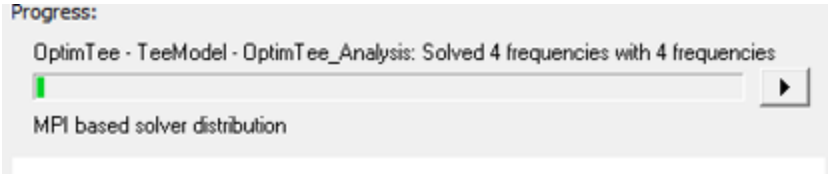


Steps are reported in the Solution profile and messages.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

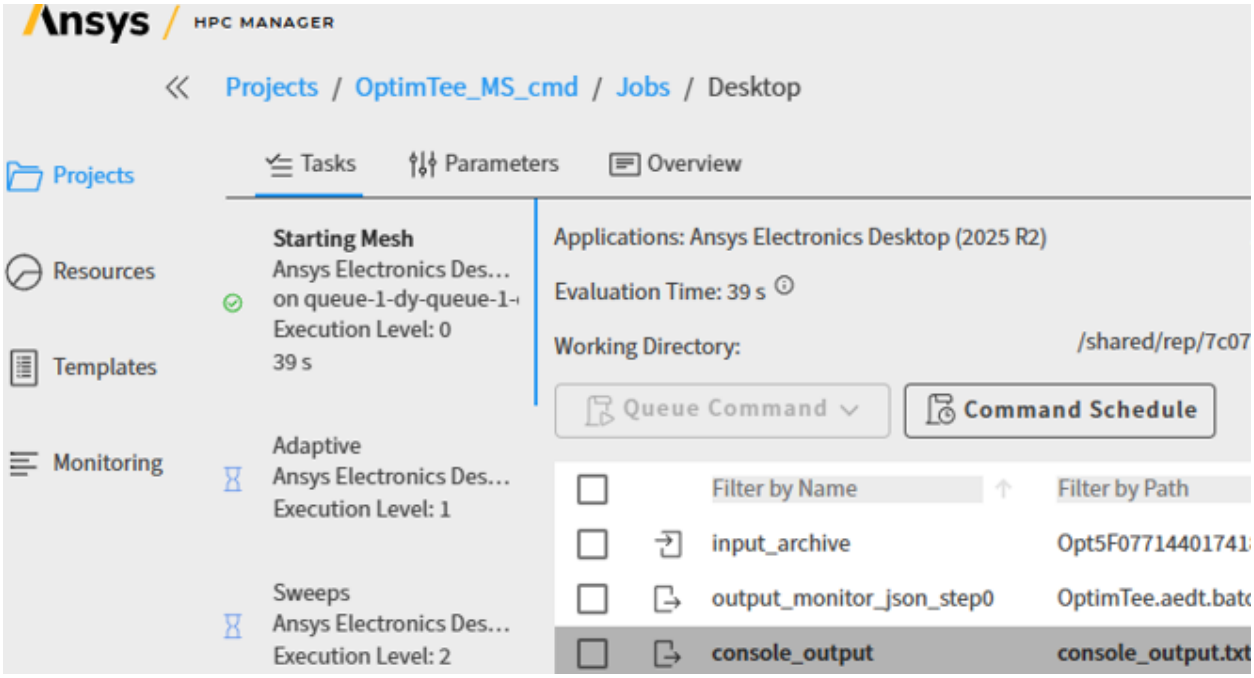


Progress bars reset after each completed step.



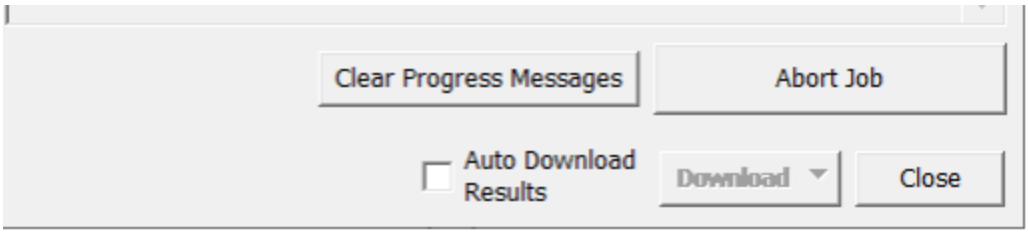
If you use the HPC manager, the steps are also reported.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Aborting an Analysis

Jobs submitted via Ansys HPC Platform Services from can be aborted through the **Monitor job** dialog. To abort a job in a Queued state, you should press the **Abort Job** button on the **Monitor Job** dialog.



To abort any running job, you can either **Abort** or **Clean-Stop** the job. Using either of these abort types, debug logs can be collected from all compute nodes in the job.

For more information, please see Aborting an Analysis.

Cluster Specific Considerations

Ansys HPC Platform Services

Job Submission Methods

Manual:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Number of Tasks and Cores: allows specification of Total number of tasks, Cores per distributed task, and RAM Limit %. Also allows manual specification of allowed Job distribution types and two-level distribution.

Automatic:

- Number of Nodes and Cores: allows specification of Total number of nodes, Total number of cores, Total number of GPUs, RAM Limit %, and Num variations to distribute.
- Number of Cores and (Optional) RAM: allows specification of Total number of cores, RAM Limit %, and Num variations to distribute.

Configuration Guidelines

Compute resource selection for Ansys HPC Platform Services involves choice of queue. These will depend on the cluster setup. The queue can also be left as “(not specified)” unless the submission method is automatic and Method is Number of Nodes and Cores. Information for the selected queue can be seen in the Resource selection parameters box. This includes cores per node and memory per node in GB.

Option Settings for Ansys HPC Platform Services

The following option is automatically added for Ansys HPC Platform Services jobs:

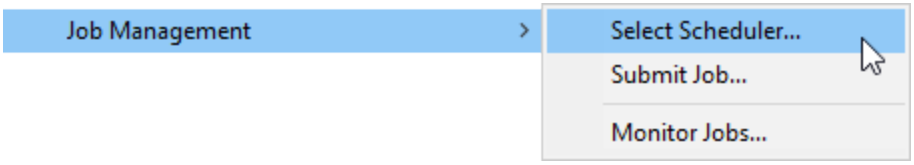
"-autoextract reports".

See: Running Electronics Desktop from a Command Line.

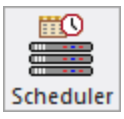
Selecting a Scheduler

Access the **Select Scheduler** window one of three ways:

- Click **Tools** > **Job Management** > **Select Scheduler...**



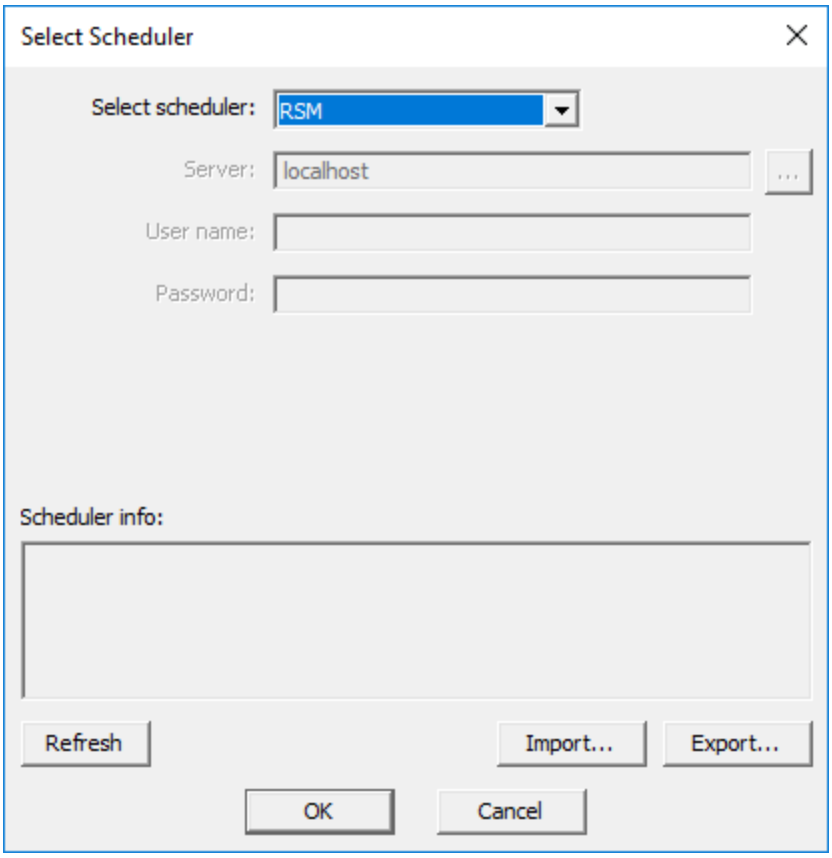
- Select the **Simulation** tab and click the **Scheduler** icon.



- From a command window, use the -showselectscheduler command.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Select Scheduler** window appears:



From the **Select Scheduler** window:

1. Use the **Select scheduler** drop-down menu to select a scheduler.

   > **Note:**
   >
   > If you select a scheduler that is unsupported in your environment, you will receive a warning message.
   >
   > See HPC Integration for a list of currently supported schedulers.

2. If applicable for the scheduler type, enter server and user information. If Ansys Cloud Burst Computing is selected then the “Login” button will open the default browser and prompt for login.

   Information about the selected scheduler appears in the **Scheduler info** field.

3. Click **OK** to complete your selection.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Submitting a Job

Ansys Electromagnetics Desktop supports its own Remote Simulation Management (RSM) and Ansys Cloud Burst Computing along with other High Performance Computing (HPC) software management programs (See: HPC Integration). The **Simulation** tab of the ribbon includes icons for setting HPC Options, creating and selecting analysis configuration, selecting the scheduler, submitting jobs, and monitoring jobs.

There are two ways that the GUI may be used to submit jobs. The first (and most common) mode requires that the Desktop (UI) process run on a host which is also a submission host for the job scheduler. This mode is called local mode or working mode.

The second mode is only supported on Linux in the Ansys Electromagnetics Suite. In the second mode, an administrator configures the RSM Service to act as an interface to the job scheduler, and starts the RSM Service on a submission host for the cluster. The user runs the Desktop (UI) process on another host (which may be called the post-processing host). To submit a job, the user specifies the host where the RSM Service is running, and the Desktop process connects to the RSM Service over the network to submit the job. In this mode, some configuration is required, and the RSM Service typically must run as a privileged user (for example, root), so that it can launch processes as any user. This mode is useful for cases in which the submission hosts are not able to run graphical processes.

> **Note:**
>
> For certain methods of resource selection for Windows HPC job submission, Ansys Electronics Desktop checks for nodes being online. In an auto-scaling cluster, nodes can fail the online check because they're not online until a job is running. An environment variable allows customers to bypass this check:
>
> ANSYSEM_SKIP_NODE_ONLINE_CHECK
>
> Set the variable value to 1 to disable the online check. Debug logging fully verifies what aspects of the cluster are causing this check to fail.

Job submission in Windows works without changing settings if environment variable ANSYSEM_SUBMIT_JOB_REQ_NODE_ONLINE is set with value 0. The installation doesn’t set this environment variable. ANSYSEM_SUBMIT_JOB_REQ_NODE_ONLINE accepted values: 0 or 1 (0 = disable; 1 = enable, i.e. require nodes to be online for submission). Default: 1 (i.e. if not set, Ansys Electronics Desktop requires nodes to be online for submission).

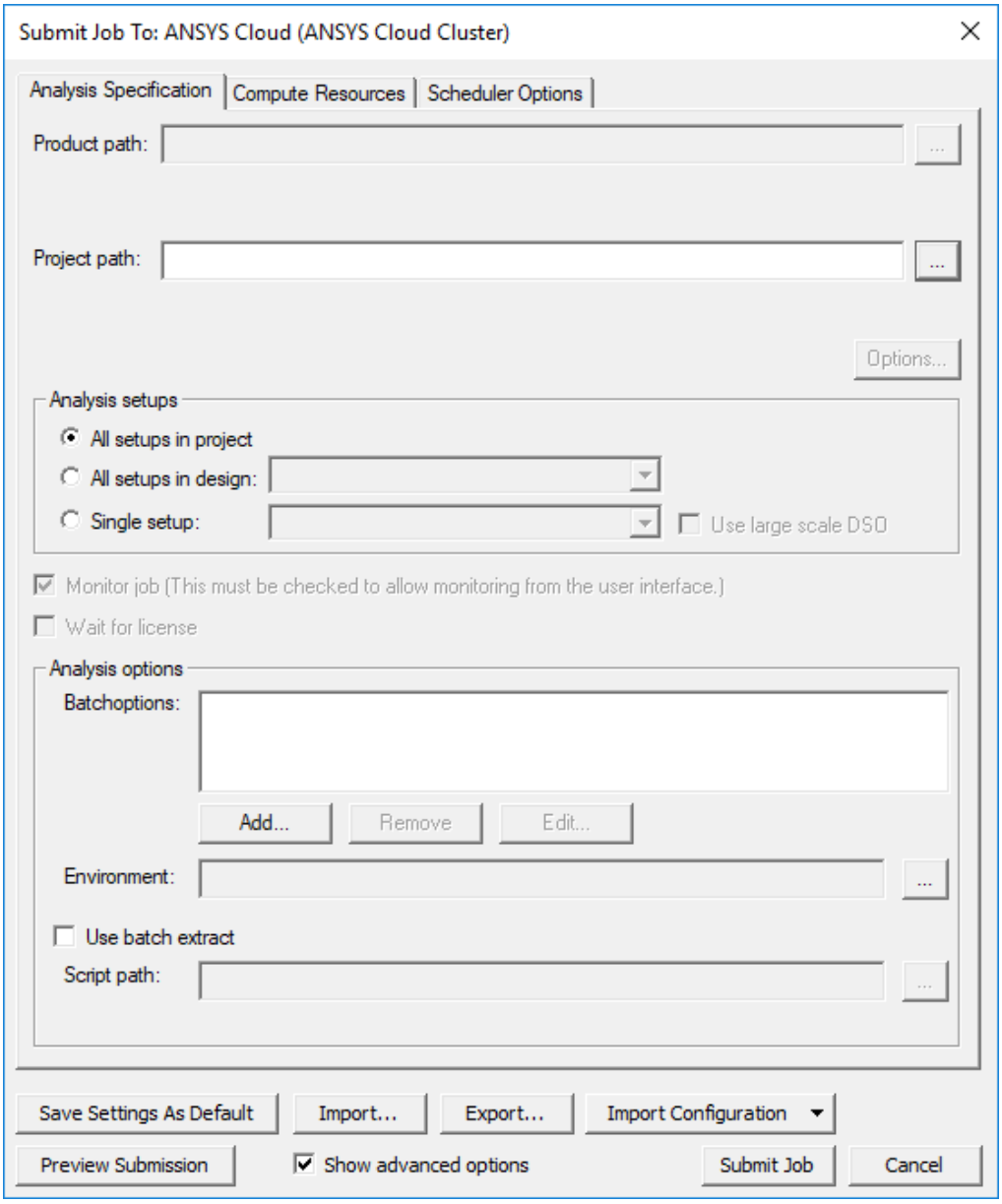
To submit a job:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Prepare your design.
2. Open the **Submit Job To** window one of the following ways:
   - On the **Simulation** tab, click the **Submit** icon.
   - Select **Tools** > **Job Management** > **Submit Job**.
   - Select **Maxwell 3D**, **Maxwell 2D**, or **RMxprt** > **Submit Job**.
   - Right-click a solution setup and select **Submit Job**. *In this case, the information in the **Submit Job To** window will be pre-populated from the setup*.
   - From a Command window, use the -showsubmitjob command.

   The **Submit Job To** window appears. The window header indicates your selected scheduler.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This window contains up to three tabs:

- **Analysis Specification** – allows you to specify the product path, project name, setup and analysis options, batchoptions, and environment variables (for advanced users).
- **Compute Resources** – allows you to specify whether to use automatic settings, and to set resource selection and job distribution parameters.
- **Scheduler Options** – allows you to specify the job name and priority. *This tab does not appear if you have selected local RSM as the scheduler*.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

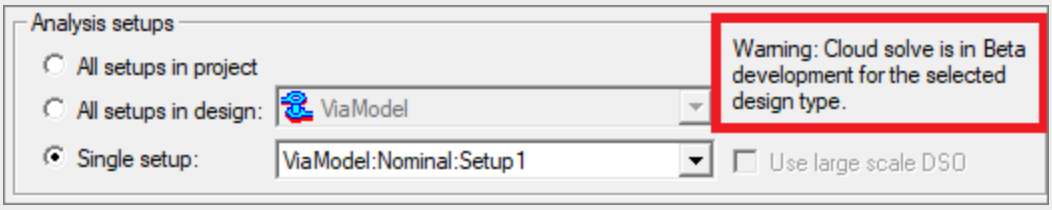
3. On the **Analysis Specification** tab, specify your desired options.

   The **Product path** and **Project** fields support mapped drives. Click the ellipses (**...**) to select files.

   The project can be an archive. The project file pathname must be a UNC path that is accessible from each compute host used for Ansys Electromagnetics jobs. After clicking the **Project path** field's ellipsis button (**...**), a check box allows you to **Use converted UNC path if mapped drive specified**. If you select a project or product on a mapped drive, and check the option, the converted UNC path equivalent to the mapped drive pathname is used.

   Note:

   If you select Ansys Cloud as your scheduler for a design type that is in Beta, you will receive a warning:

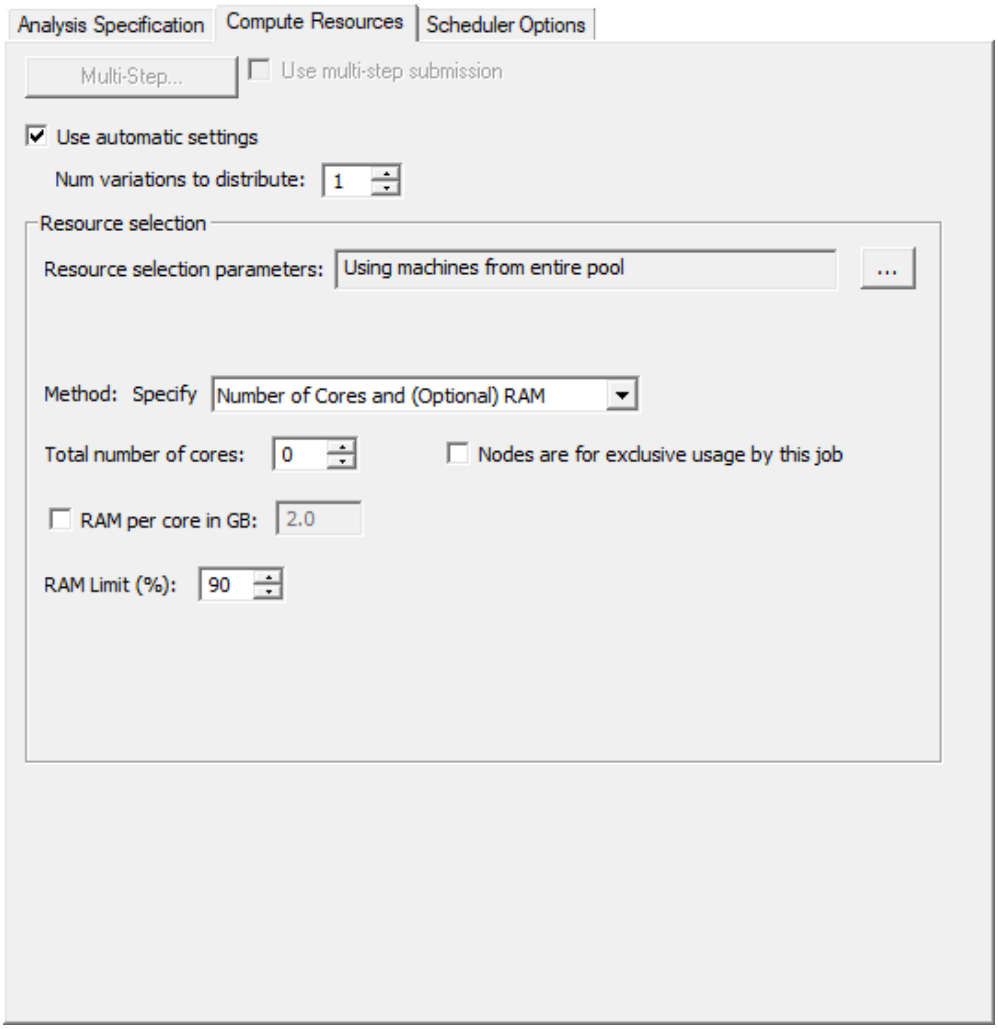
   

   In the **Analysis Setups** area, select the radio button for **All setups in project**, **All setups in design**, or a **single setup**.

   For Parametric setups, you have the option to select **Use Large Scale DSO**. See: Large Scale DSO for Parametric Analysis.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Select the **Compute Resources** tab.



Options may vary slightly depending on your selected scheduler.

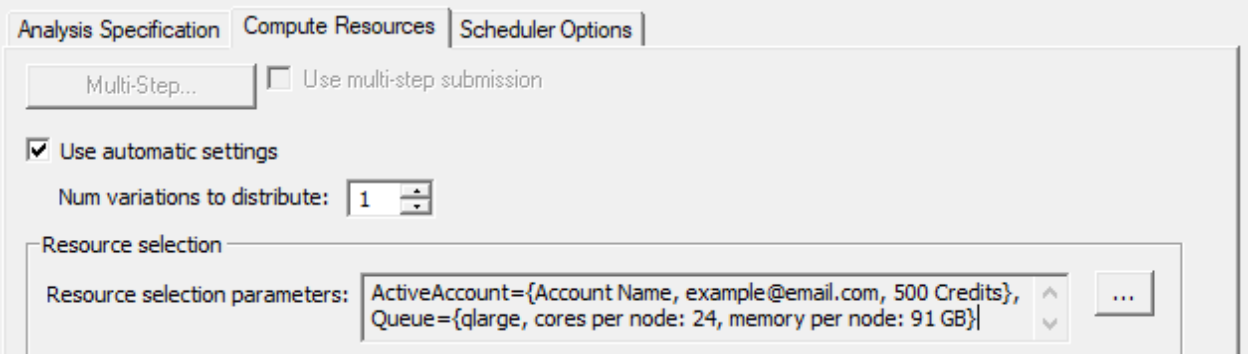
5. If applicable, determine whether or not to **Use multi-step submission**.
6. Determine whether or not to **Use automatic settings**.

> **Note:**
>
> Ansys strongly recommends using Automatic settings.

7. Use the buttons in the **Resource selection** area to allocate resources. See: Job Management for Large Scale DSO. If you are using Ansys Cloud Burst Compute select an active account using the ellipsis […] button to the right of the **Resource selection parameters:** text box and ensure that it has sufficient Core Hour Balance. This balance

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

will be displayed in the **Resource selection parameters:** text box.



8. If you opted not to use automatic settings, you can also set **Job distribution** parameters. See: Distributed Analysis.
9. If applicable, select the **Scheduler Options** tab and set the **Job name** and **Priority**.

10. If desired, import or export job configurations.
11. If desired, click **Save Settings as Default** to save the current settings and overwrite defaults. These settings are saved on a per-scheduler basis.
12. If desired, click **Show advanced options** to see options for advanced users. See: Running Electronics Desktop from a Command Line.
13. If desired, click **Preview Submission** to view the commands to be sent to the scheduler. Ansys Cloud Burst Computing submissions contain queue (pool) configuration details, including the job's hourly cost in Ansys Elastic Units (AEUs).

    The preview text can be copied to the clipboard.
14. Click **Submit Job** to submit the job to your selected scheduler.

Using Advanced Job Submission Options

From the job submission window, you can select **Show advanced options** to enable additional analysis options and job submission options.

This topic covers the following:

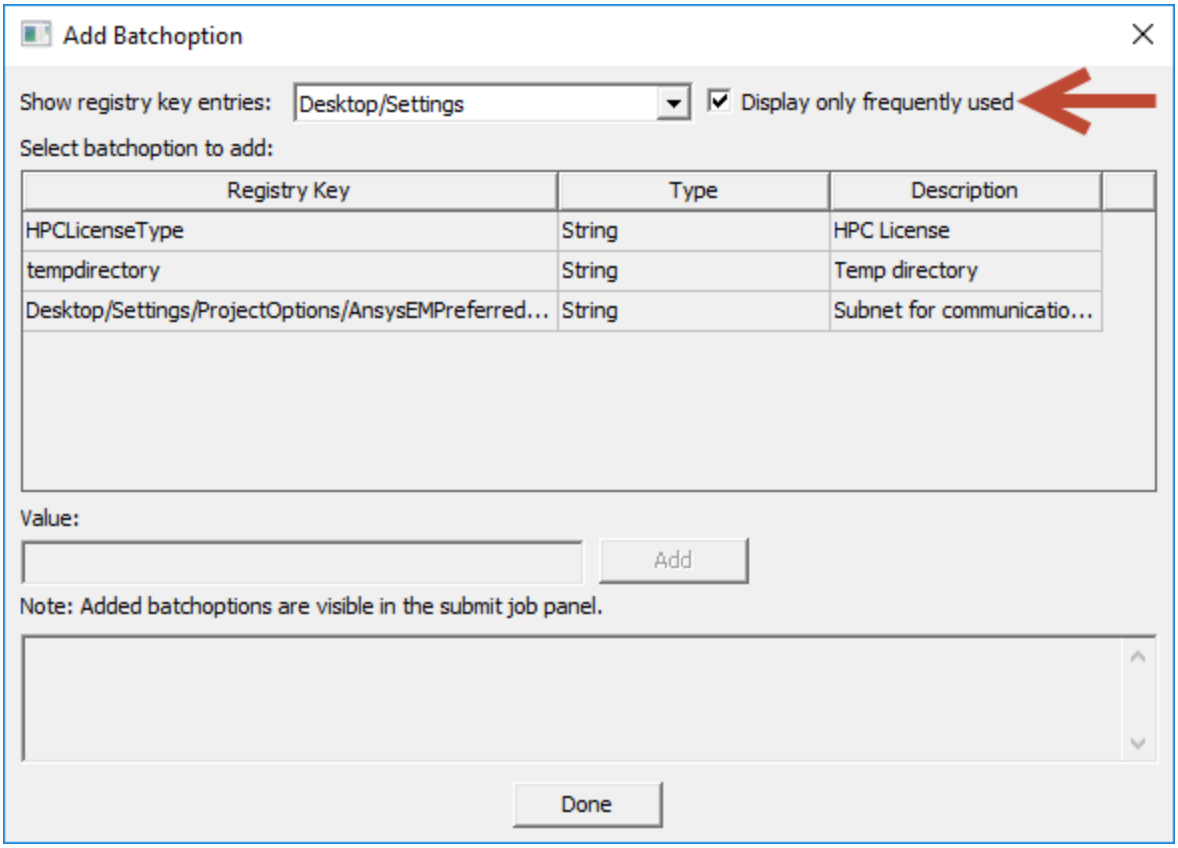
- Batchoptions
- Environment Variables

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Batch Extract
- Customize Job Submission

Batchoptions

In the **Submit Job To** window, under **Batchoptions**, click **Add** to open the **Add Batchoption** window.

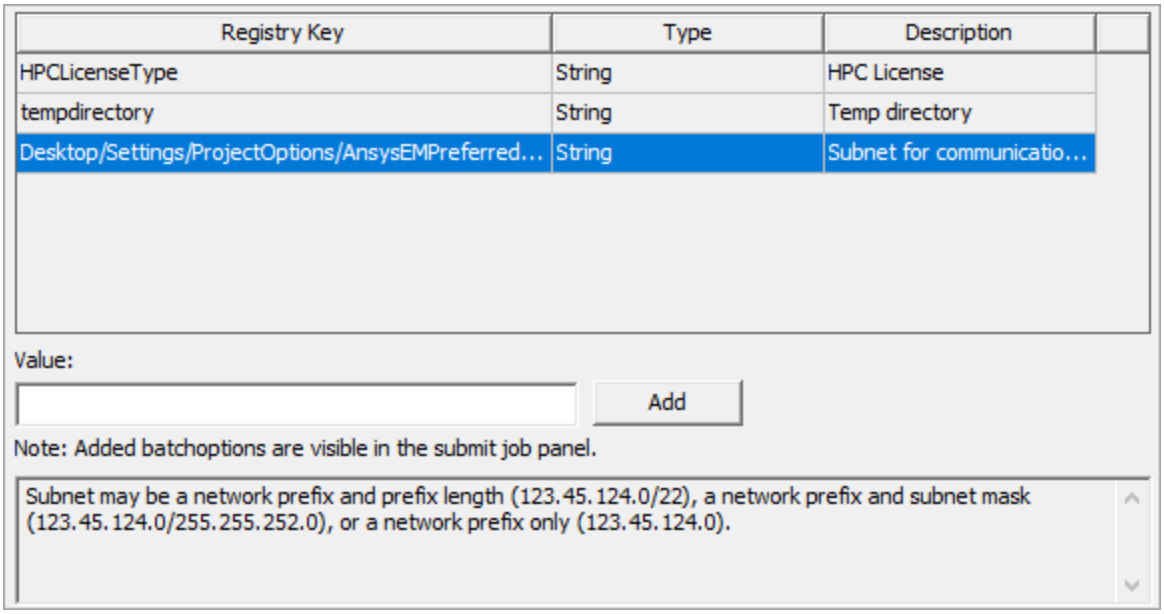


The **Show registry key entries** drop-down menu allows you to select categories of registry keys to display.

Note that **Display only frequently used** is selected by default. Deselect this to view all options for the selected category.

Select a registry key to activate the **Value** field, where you can enter a value. Selecting a key also populates the bottom of the window with a description of that key:

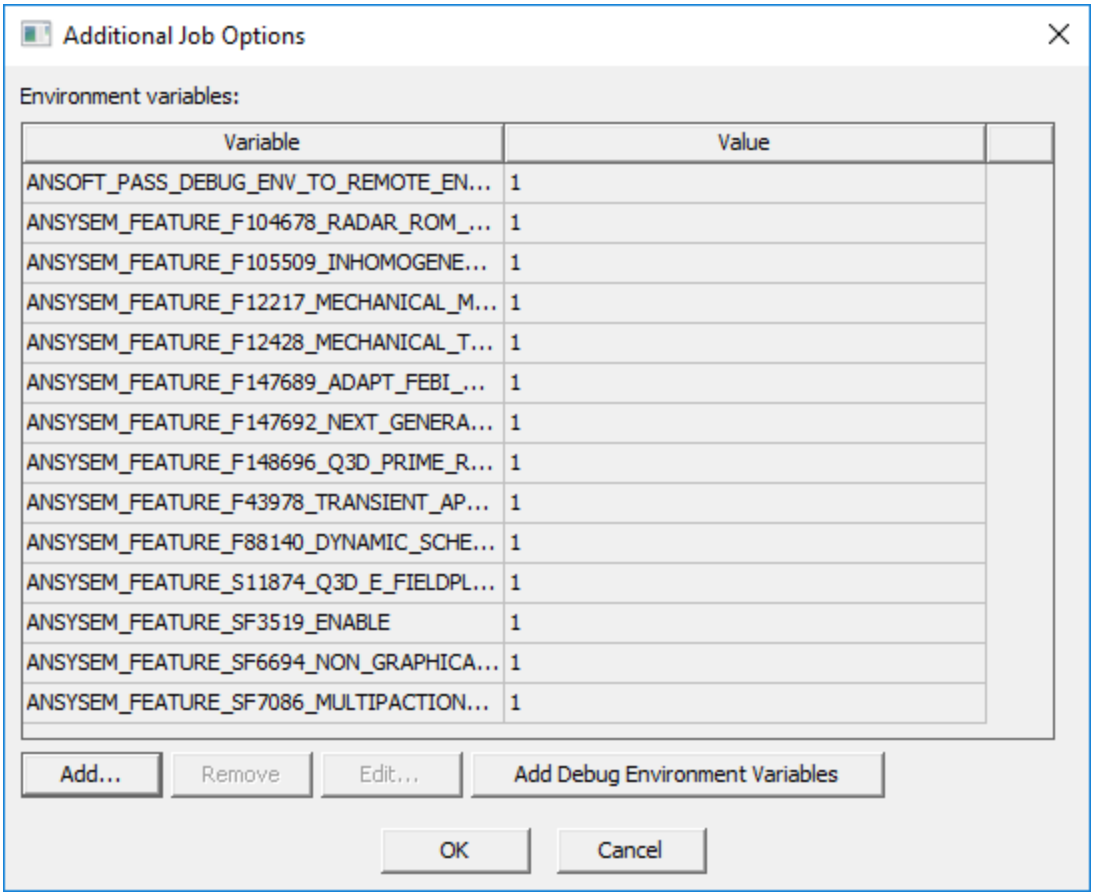
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Environment Variables

In the **Submit Job To** window, under **Environment**, click the ellipses (**...**) to open the **Additional Job Options** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This window lists all currently activated environment variables.

Active variables display a value of 1. Inactive variables display a value of 0. Non-binary variables may contain project paths, integer values, etc.

You can perform the following actions:

- **Add** – Add an environment variable. You will be prompted to enter the variable name and value.
- **Remove** – Select and remove an environment variable from the list.
- **Edit** – Select an environment variable and enter a new value.
- **Add Debug Environment Variables** – Activates a selection of debugging variables and adds them to the list. The log files created are only useful to development and if a customer or an application engineer needs to set these environment variables they should be working with a developer directly or indirectly who will know what needs to be set.

Batch Extract

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the **Submit Job To** window, select **Use batch extract** to enable the **Script path** field. Click the ellipses (**...**) to browse and select a Python script to execute along with the job. See: Running Ansys Electronics Desktop from the Command Line for a description of Batch Extract.

Customize Job Submission

In the **Submit Job To** window's **Scheduler Options** tab, advanced options for some schedulers allow you to **Customize job submission**. When the **Override job submission** radio button is selected, user-specified options replace most of the job submission options. When the **Additional job submission options** radio button is selected, user-specified options are appended to the bsub command.

Using the Command Line to Submit HPC Jobs

Ansys Electronics Desktop can be run from the command line. When using the command line to perform HPC jobs, take the following into consideration.

Distributed Jobs

An Ansys EM batch job which distributes the analysis over several hosts may also be called a distributed job. To submit a distributed job, the following Ansys EM desktop command line options should be used:

- The -Distributed option should be present, and the -Local option should be absent. When running as a batch job under one of the schedulers with direct integration, this option is a directive to the job to 1) obtain the list of hosts allocated to the job, directly from the scheduler, and to 2) use the scheduler to launch the analysis engines on the hosts allocated to the job. The -Distributed option may now have additional options, such as includetypes=xxx, excludetypes=xxx, maxlevels=n, and numlevel1=n, where n indicates and integer, and xxx indicates a list of distribution types or "default".
- The -Machinelist num=*num_distributed_engines* option must be included, where *num_distributed_engines* is the total number of analysis engines to be started on the hosts assigned to the job.

Other examples:

- "Serial Job on a Single Processor" below
- "Distributed Job Using Four Processors" on the facing page
- "Multiprocessing Job Using Four Cores" on the facing page
- "Distributed Analysis and Multi-Processing in the Same Job" on the facing page

Serial Job on a Single Processor

Suppose Ansys Electronics Desktop is installed at "C:\Program Files\ANSYS Inc\v252\AnsysEM\" and you are using RSM for DSO:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
"C:\Program Files\ANSYS Inc\v252\AnsysEM\ansysedt.exe" -ng -
BatchSolve -machinelist num=2
-monitor \\shared_drive\projs\OptimTee.aedt
```

User is using LSF for remote-analysis/DSO

```
bsub -n 1 "C:\Program Files\ANSYS Inc\v252\AnsysEM\ansysedt.exe" -ng
-BatchSolve -machinelist num=3 -monitor -local \\shared_
drive\projs\OptimTee.aedt
```

Distributed Job Using Four Processors

Ansoft RSM

```
"C:\Program Files\ANSYS Inc\v252\AnsysEM\ansysedt.exe" -ng -
Batchsolve -monitor -Distributed
-machinelist list="10.1.1.221, 10.1.1.222, 10.1.1.223, 10.1.1.224"
\\shared_drive\projs\OptimTee.aedt
```

LSF

```
bsub -n 4 "C:\Program Files\ANSYS Inc\v252\AnsysEM\ansysedt.exe" -ng
-Batchsolve -monitor
-Distributed -machinelist num=4
\\shared_drive\projs\OptimTee.aedt
```

Multiprocessing Job Using Four Cores

Multi-processing job using 4 cores

```
bsub -n 4 -R span[ptile=4] "C:\Program Files\ANSYS
Inc\v252\AnsysEM\ansysedt.exe" -ng -monitor
-Local -BatchSolve -machinelist num=4 -batchoptions \\shared_
drive\registry.txt \\shared_drive\projs\OptimTee.aedt
```

This requests 4 cores to come from the same machine, as multi-processing needs cores to be on the same machine

Distributed Analysis and Multi-Processing in the Same Job

Distributed-processing using 4 engines and multi-processing using 4 cores, using a total of 16 cores

```
bsub -n 16 -R "span[ptile=4]" "C:Program Files\ANSYS
Inc\v252\AnsysEM\ansysedt.exe" -ng
-BatchSolve -Distributed -machinelist num=4
-batchoptions \\shared_drive\registry.txt
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
\\shared_drive\projs\OptimTee.aedt
```

Job Import and Export

The bottoms of the scheduler selection and job submission window contain buttons for **Import...**, **Export...**, and **Import Configuration**. Import and Export may be used to save and then restore a frequently used collection of job submission settings, to save multiple sets of settings, or to transfer settings from one machine to another. Electronics Desktop uses Ansoft Electronics Registry Settings (*.areg) files for these purposes:

- **Export...** – exports most of the settings of this window (all tabs) to an *.areg file.
- **Import...** – updates most of the settings in this window (all tabs) from an *.areg file.
- **Import Configuration** – updates DSO settings in this window from any DSO configuration, as shown in the **Configurations** and **Options** tabs of **HPC and Analysis Options**.

> **Important:**
>
> The Design Type of the DSO configuration must match the design type of one of the designs in the project, so the **Project path** must be specified before using the **Import Configuration** button.

Scripting

The SubmitJob scripting command uses job submission settings that have been exported from the **Submit Job** window to an *.areg file. The path to this *.areg file is the first argument to the SubmitJob scripting command. See: Job Submission Scripting.

Multi-Step Job Submission

Multi-step job submission allows you to divide the simulation of a project on a cluster as two or more jobs, each of which has unique resource specifications. For example, creating the initial mesh and doing adaptive refinement can use a single machine, while frequency sweeps can easily be distributed over many machines. Breaking up the simulation into multiple steps allows the first job to do initial meshing and adaptive passes, only reserving a single compute node, or maybe even reserving just a partial node. The second job can then do the frequency sweep(s), reserving and using multiple nodes. Note that while the first job runs, because it may only be using one node, other nodes are available for other jobs. The Ansys Electronics Desktop job submission GUI allows you to submit multi-step jobs, and specify compute resources individually for each step. Electronics Desktop can also be used to monitor the execution of multi-step jobs.

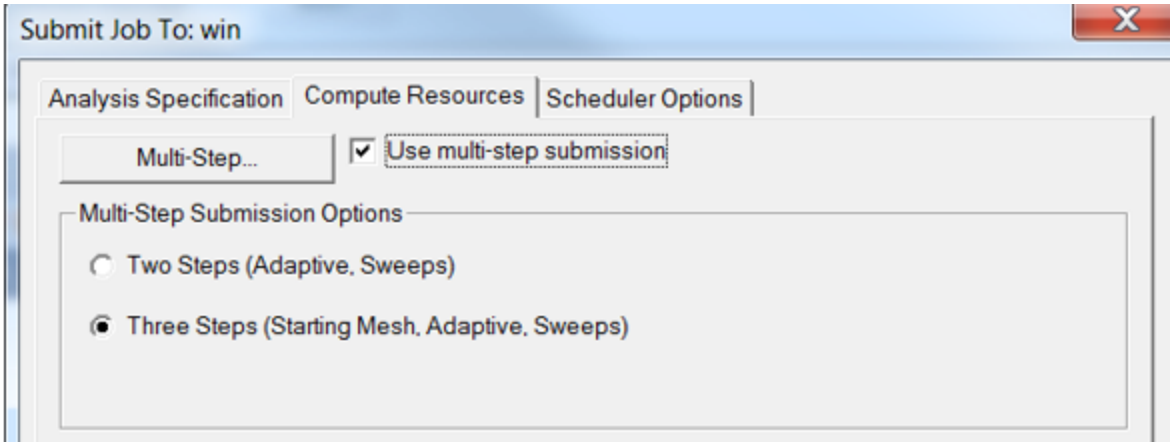
Limitations

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

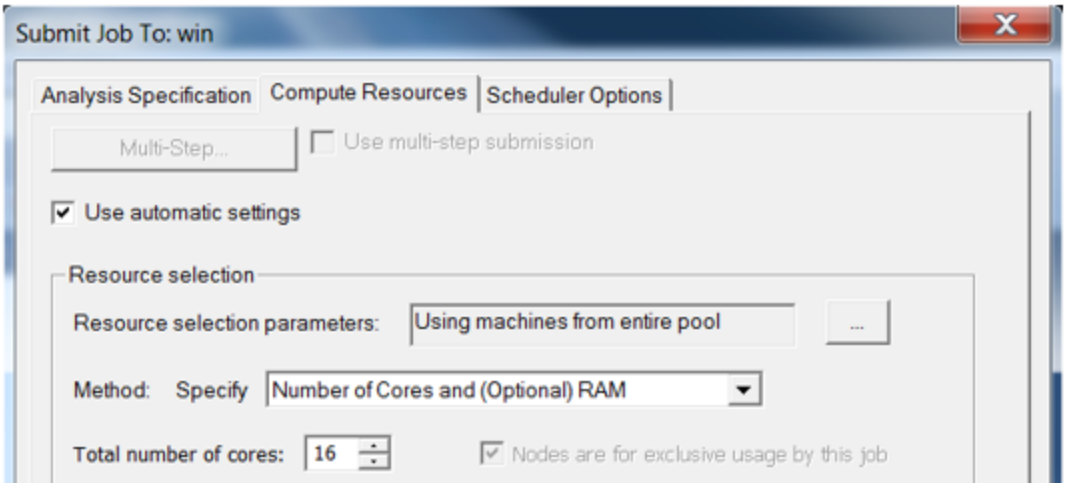
Multi-step job submission is only allowed for a single setup of a project. Only certain design types (or setup types for a given design type) offer this functionality. Ansoft RSM does not support multi-step jobs because it does not have queuing capabilities.

How-to Specify Multi-Step Job Submission

From an open project, right-click a setup in the project tree, and select **Submit Job...** on the shortcut menu. This pre-populates the **Submit Job To** window's **Analysis Specification** tab for the selected setup. Select the **Compute Resources** tab. If the design setup and selected scheduler allow for multi-step submission, and your computing resource supports it, the **Compute Resources** tab shows the **Multi-Step...** button and the **Use multi-step submission** check box is enabled.



If the **Multi-Step** button and check box are not enabled, it could be because you have not selected a single setup, the design type of the setup does not support Multi-Step, or the scheduler type (e.g., RSM) does not support it.

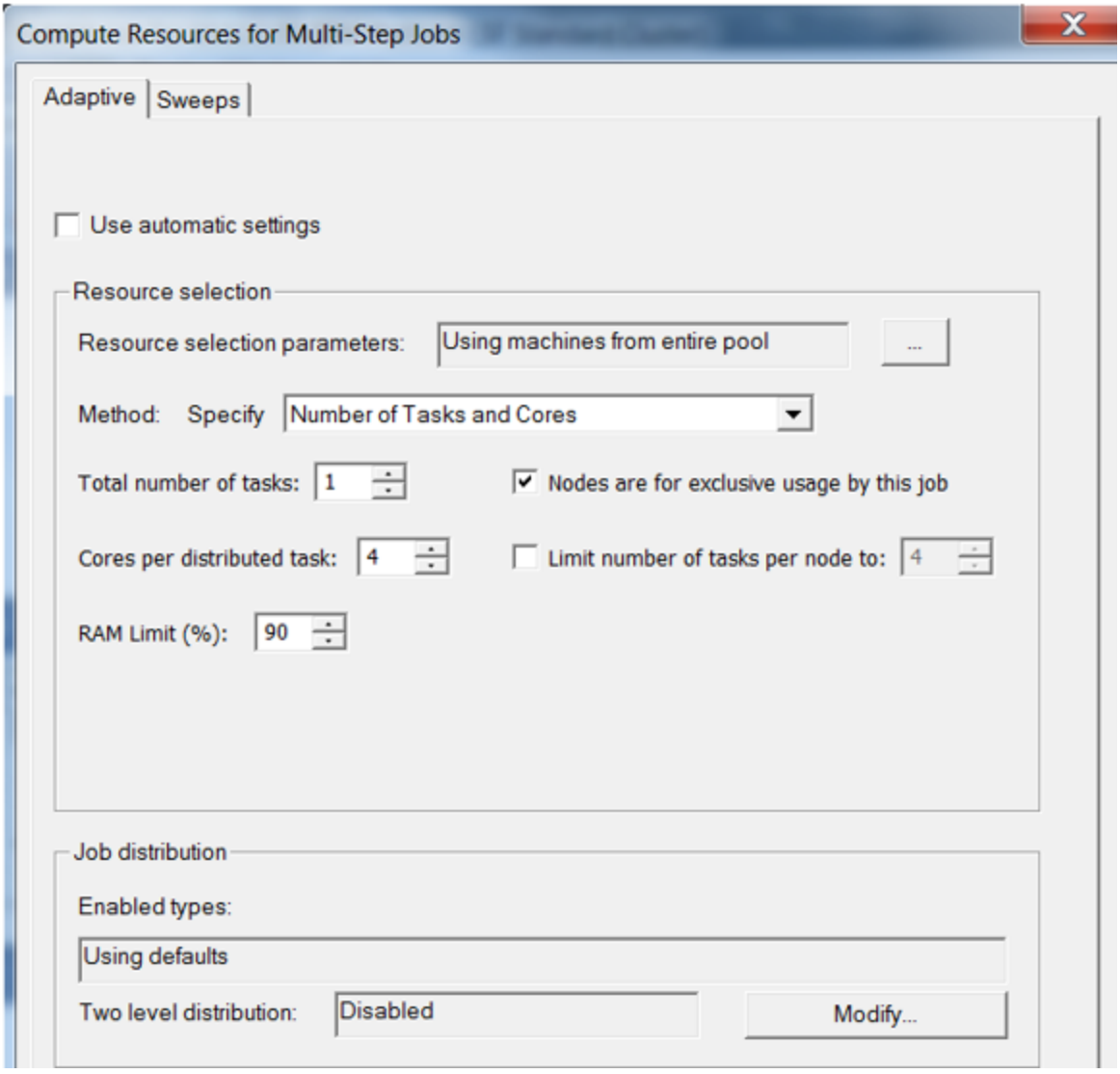


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

On the **Compute Resources** tab, check the **Use multi-step submission** box and select the appropriate submission option:

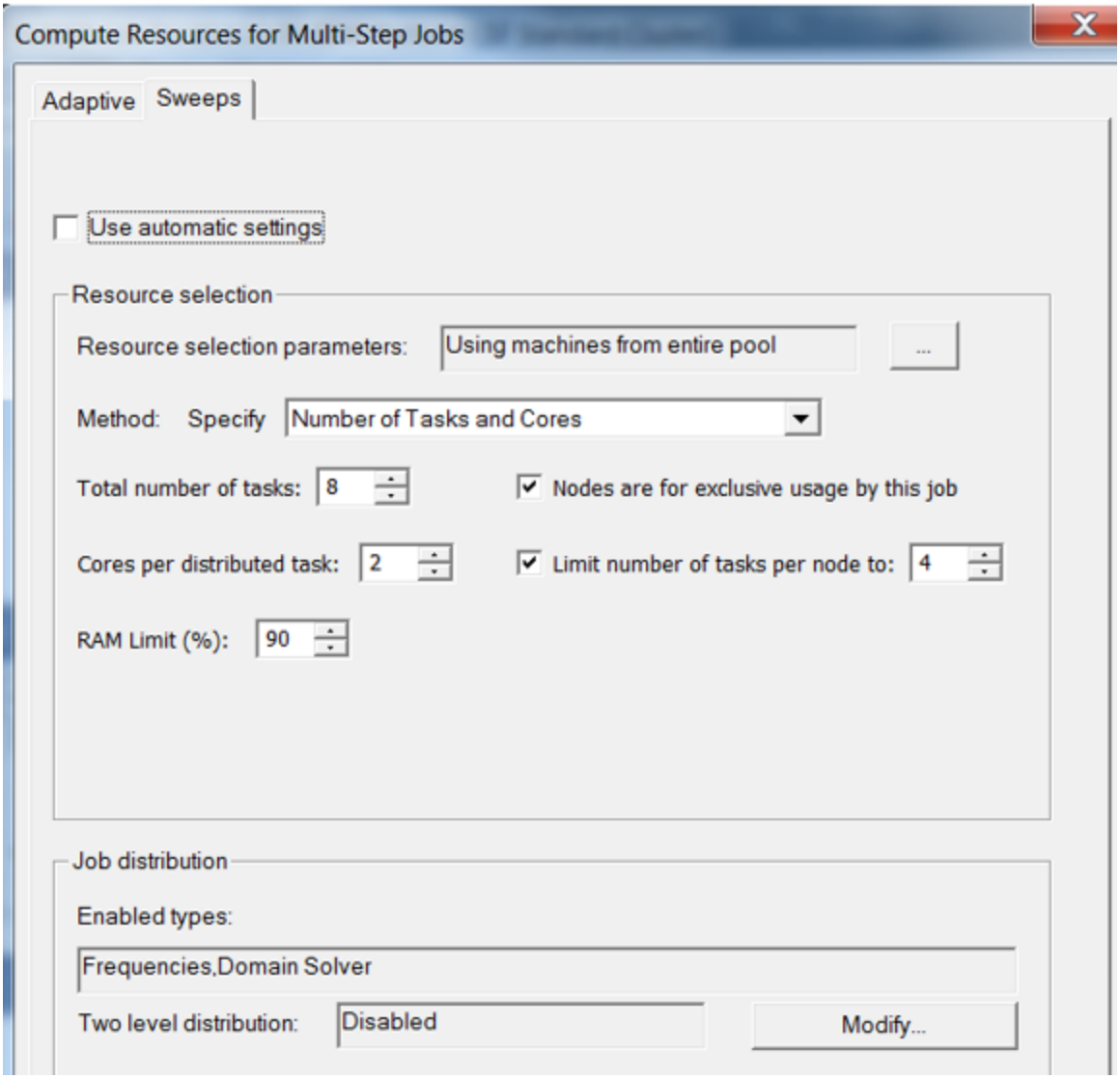
- Two Steps for Adaptive and Sweeps
- Three Steps for Mesh, Adaptive, and Sweeps

Then click **Multi-Step…**, which will bring up a **Compute Resources for Multi-Step Jobs** dialog box where there is a tab for each step used, that is for potential for Mesh, Adaptive, and Sweeps.



Notice, for example, that the **Sweeps** tab lists the same resources choices but they can be assigned differently.

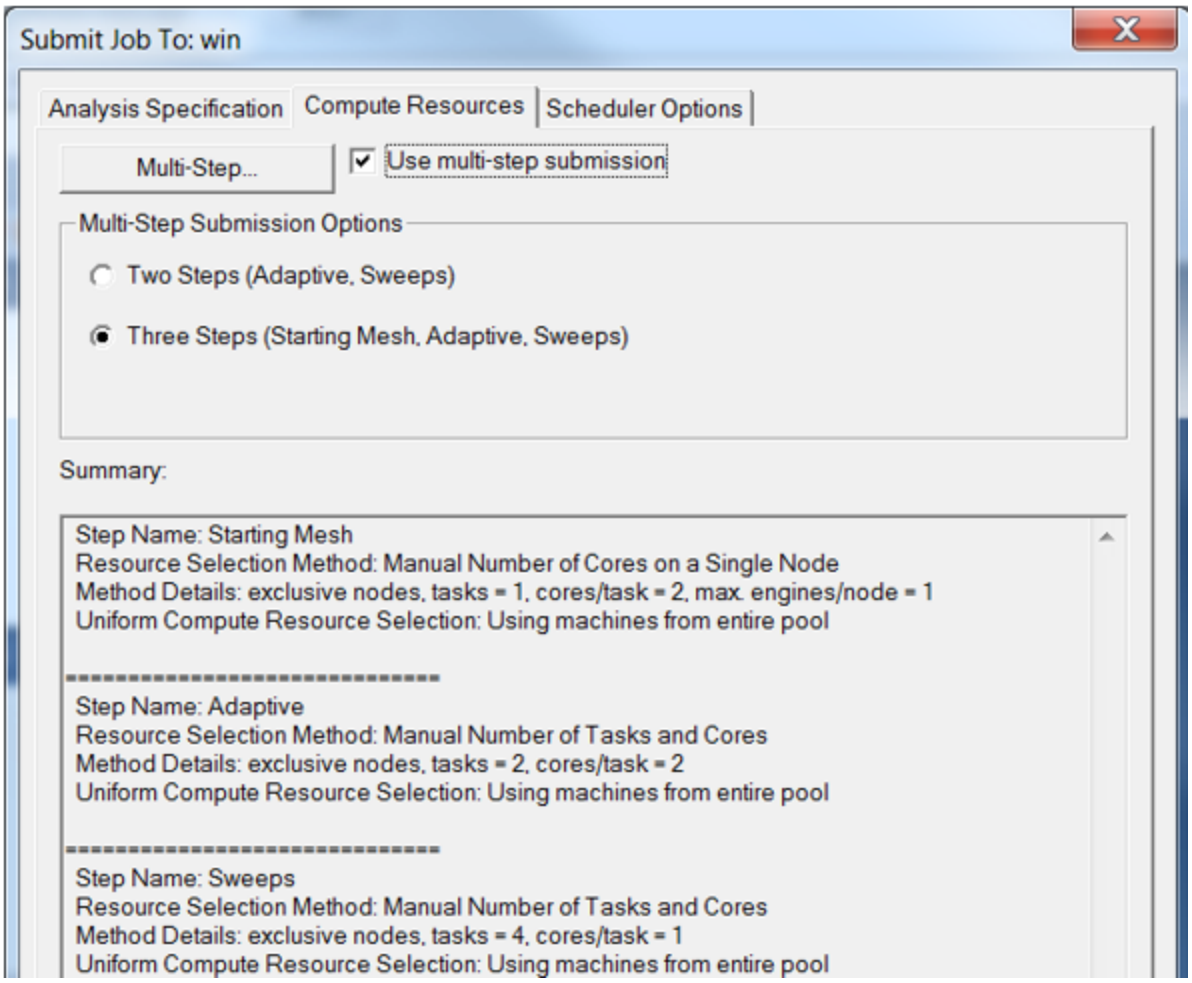
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



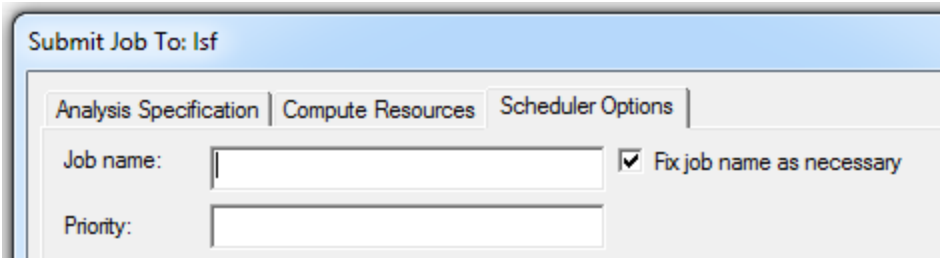
Set the desired compute resources for each step and click **OK**.

The Summary field of the **Compute Resources** tab shows a text summary of resource specifications used for each step.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



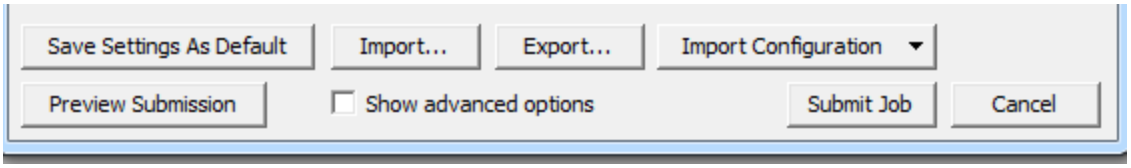
If desired, set the job name on the **Scheduler Options** tab. When the jobs are submitted, each will have a name given by job name (if any) appended with the name of the step for the job. For example, if the job name entered is “MultiStep”, then the individual jobs might be named “MultiStep-Adaptive” and “MultiStep-Sweeps”.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

There is a new **Fix job name as necessary** check box for the job name. This applies only to certain schedulers where there are constraints on the job name. It is useful to ensure proper submission in the case where appending the step name results in an invalid job name.

When you click **Preview Submission**, you see a preview for each step, all in the same output window. Any errors or warnings for a step, generated during preview validation, are displayed with the text for the preview of that step.



When you click **Submit Job**, each job is submitted individually, and dependencies are set with the scheduler so that subsequent jobs wait for the prior step's job to complete before starting. Any errors in job submission for any step prevent further steps from being submitted.

The window reporting successful submission has been modified slightly for multi-step jobs. It will show the jobs IDs of all jobs that are part of the multi-step job sequence.

Multi-Step Job Monitoring

You can monitor the job step in progress. When one job completes, the status shows "Completed", but monitoring restarts once the next job step is running.

Aborting a Job Step

You can also abort the job step in progress. With multi-step jobs submitted from the Electronics Desktop GUI, this will also cause remaining jobs (which would otherwise remain queued in the scheduler) to be canceled.

Archive Projects for Multi-Step Job Submission

Note that you can submit archive projects. Monitoring is based on the archive for the first step, then on the extracted project for subsequent steps.

Submitting a Job without Opening the Project

You can also submit a job without opening the project. This can be done by choosing **Tools** > **Job Management** > **Submit Job…** and then manually entering the project path (You can also use **Browse** can to select the project.) Note that you must also select a single nominal setup before the **Use multi-step submission** check box is enabled on the **Compute Resources** tab.

Windows to Linux Job Submission

Given a set of prerequisites, Ansys Electronics Desktop can permit Windows to Linux job submission as part of HPC.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Prerequisites for Job Submission

Directory Shared between Windows and Linux

For all jobs submitted to a Linux cluster, the project file is required to be in a directory that is accessible from all execution hosts used by the job. For submission of jobs from a Windows host to a Linux cluster, the project file must also be accessible from the Windows host where the GUI runs. There must be a directory shared with both Windows and Linux hosts, and the project file may be in a subdirectory (at any level) of the shared directory.

Network Access from Windows Host to Linux Job Management Host

The job is submitted to the cluster from a Linux host configured for submission of jobs to the Linux cluster. We call this Linux host the “Job Management” host. The information about the job to be submitted is transmitted to the Job Management host over the network. As a result, the Windows host where the GUI runs must have network access to the Job Management host. If this communication is blocked, then job submission from a Windows host to the Linux cluster will not be possible. Communication could be blocked if there is a firewall or if the Linux cluster is only on a private network, for example.

RSM Service Running on Job Management Host

The **ansoftrsmservice** must be running on the Linux Job Management host. Before the ansoftrsmservice is started, it must be configured for submission of jobs to the cluster. The SchedulerName and ConfigString fields in the Scheduler block of the ansoftrsmservice.cfg configuration file must be specified. The contents of these fields are described in the following table:

Field Name	Contents	Examples
SchedulerName	Identifier of Scheduler	IBM Spectrum LSF: ’`lsf`’ PBSPro or Torque: ’`pbs`’ Univa, SGE, etc.: ’`sge`’ SLURM: 'generic'
ConfigString	Directory containing scheduler commands	IBM Spectrum LSF: ’’ (not required) PBSPro or Torque: ’`/opt/pbspro/PBSPro_13.0.0/default/bin`’ Univa, SGE, etc.: ’`/opt/univa/bin/lx-amd64`’ SLURM:'{"Proxy":"slurm"}'

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The environment should be configured for job submission before starting the ansoftrsmservice. The ansoftrsmservice should be run as a non-privileged user; no special privileges are required. It should be run as a user without login privileges, so that only privileged users have access to this process.

Prerequisites for Job Monitoring

For job monitoring, all prerequisites for job submission are required. One additional requirement, described below, is also required for job monitoring.

Network Access from Windows Host to Linux Cluster Hosts

In order to obtain full monitoring information from a job, the Windows host needs access to some of the job processes. That is, the Windows monitoring host requires network access to the processes running on the Linux cluster execution hosts. If this communication is blocked, then only limited monitoring information is available.

Supported Schedulers

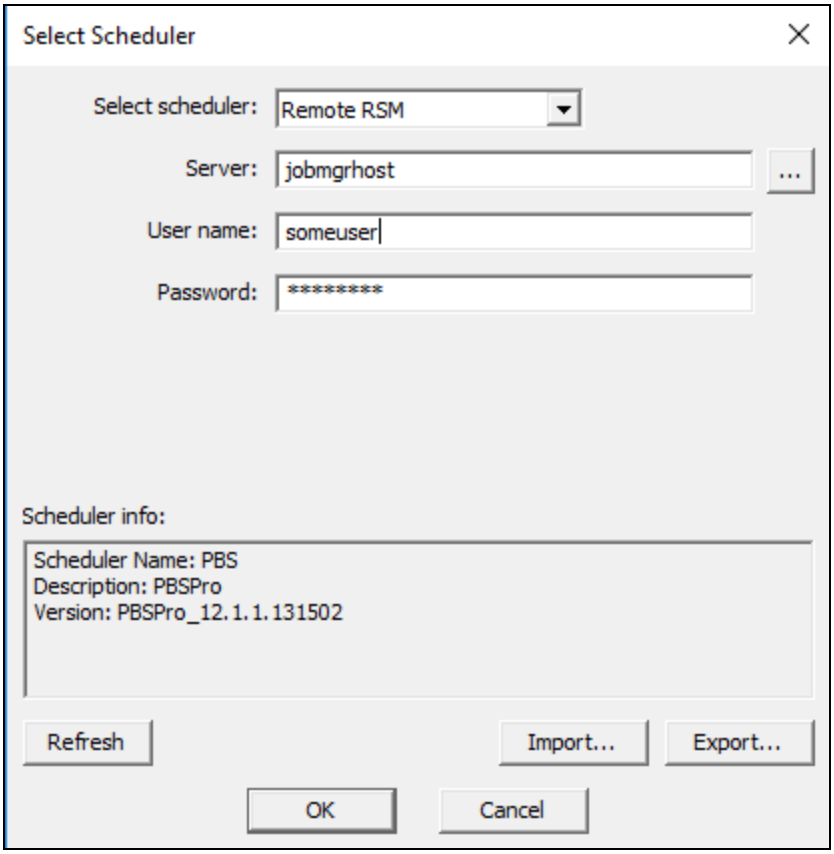
This feature may be used with all Linux schedulers for which job submission from the GUI is supported:

- IBM Spectrum LSF
- Univa Grid Engine (GE)
- PBSPro
- Torque

Select Scheduler Dialog

If you select the **Use a computer on the network** option, you can enter a username and password. This username and password are used when the job is submitted to the Linux scheduler.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

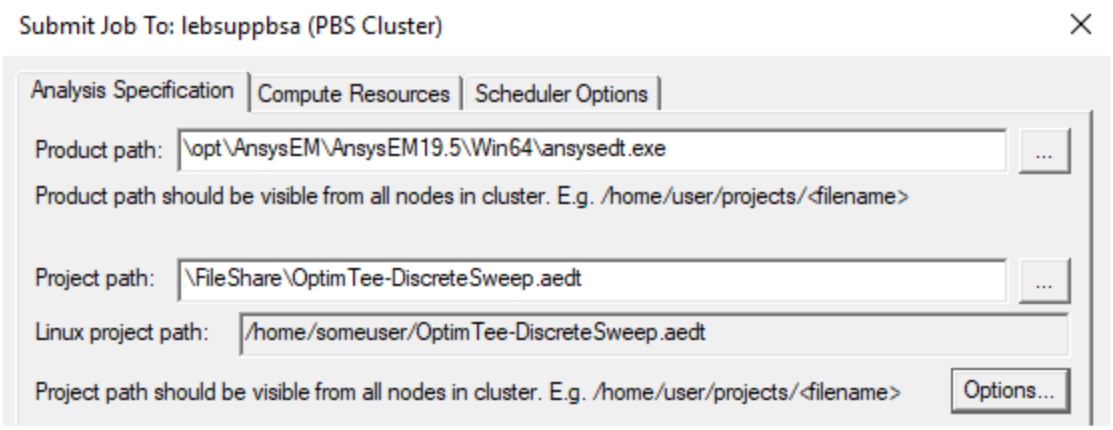


Submit Job

In the ***Submit Job To*** window, you must enter the Linux path to the product in the Product path edit control. The browse button (labeled "...") may not be used to browse for the product. There is no requirement for the product installation directory to be accessible from the Windows GUI host.

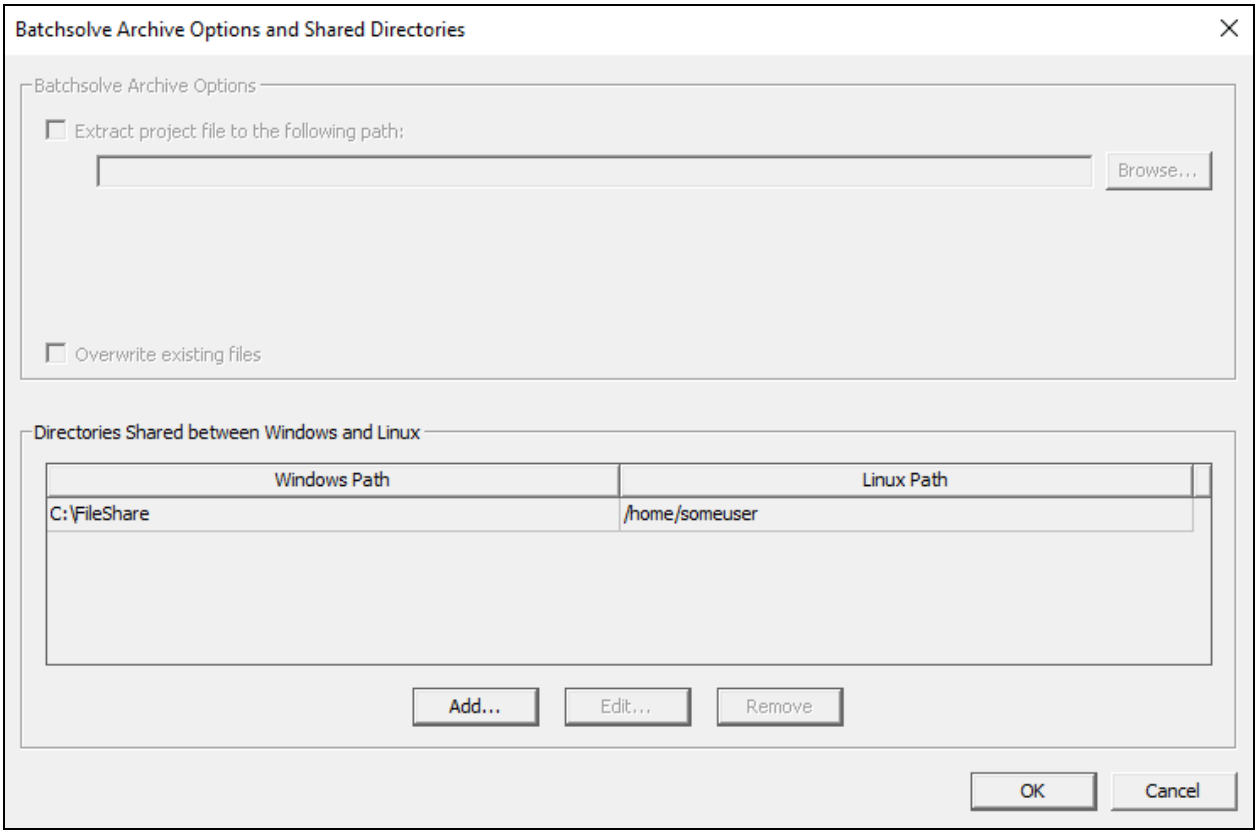
You must enter the Windows path of the project file in the Product path edit control or use the browse button (labeled "...") to select the Windows path of the project file. If the Linux path of the project file can be determined from the specified Windows path and the directories shared between Windows and Linux, then the Linux path of the project file is shown in the Linux project path edit control. This edit control cannot be edited directly.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



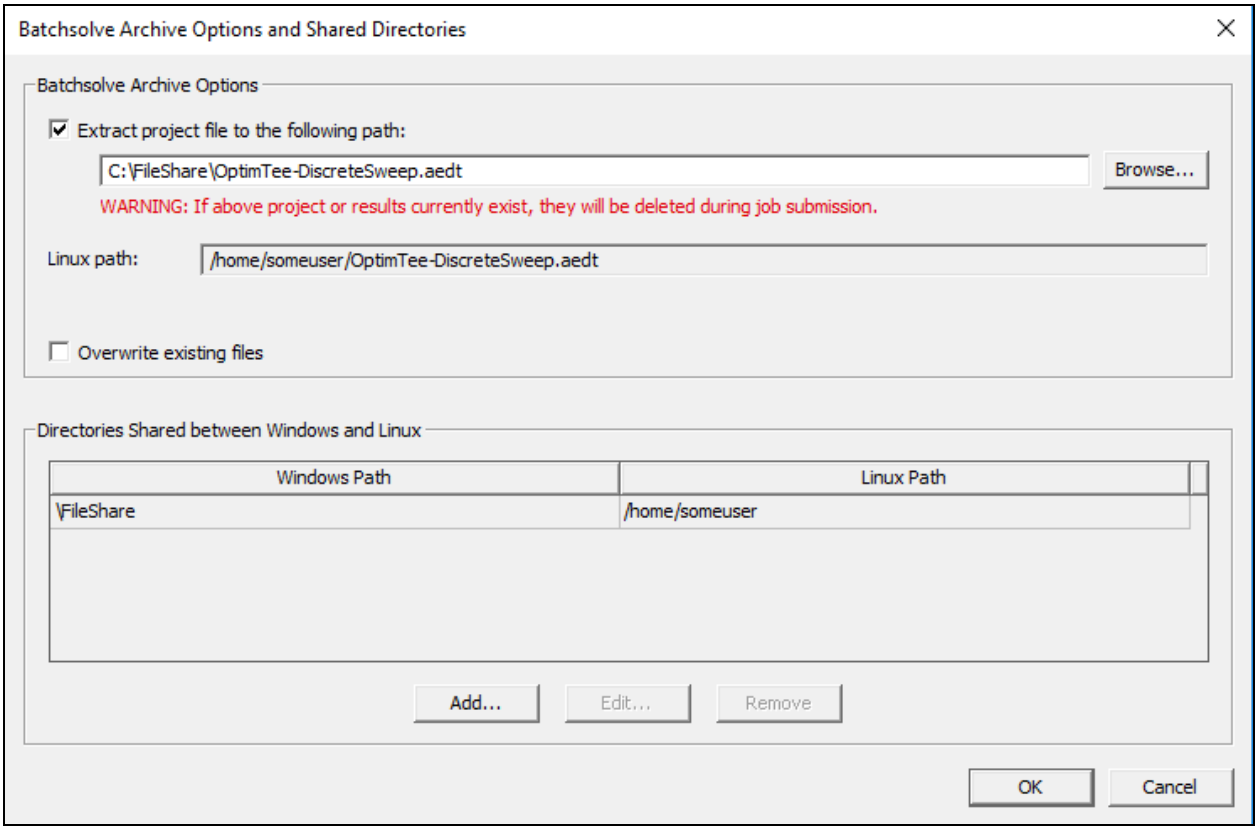
The **Options** button opens a window that you can use to specify archive options for a job. The lower portion of this window allows you to specify one or more directories shared between Windows and Linux. The mapping of directories between Windows and Linux is shown in a grid which displays the Windows path and the Linux path for each shared directory. There are also buttons to add a new shared directory, to edit an existing shared directory, or to delete one or more shared directories. The Windows path or the Linux path of any shared directory may be selected in the grid and directly edited, as well.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



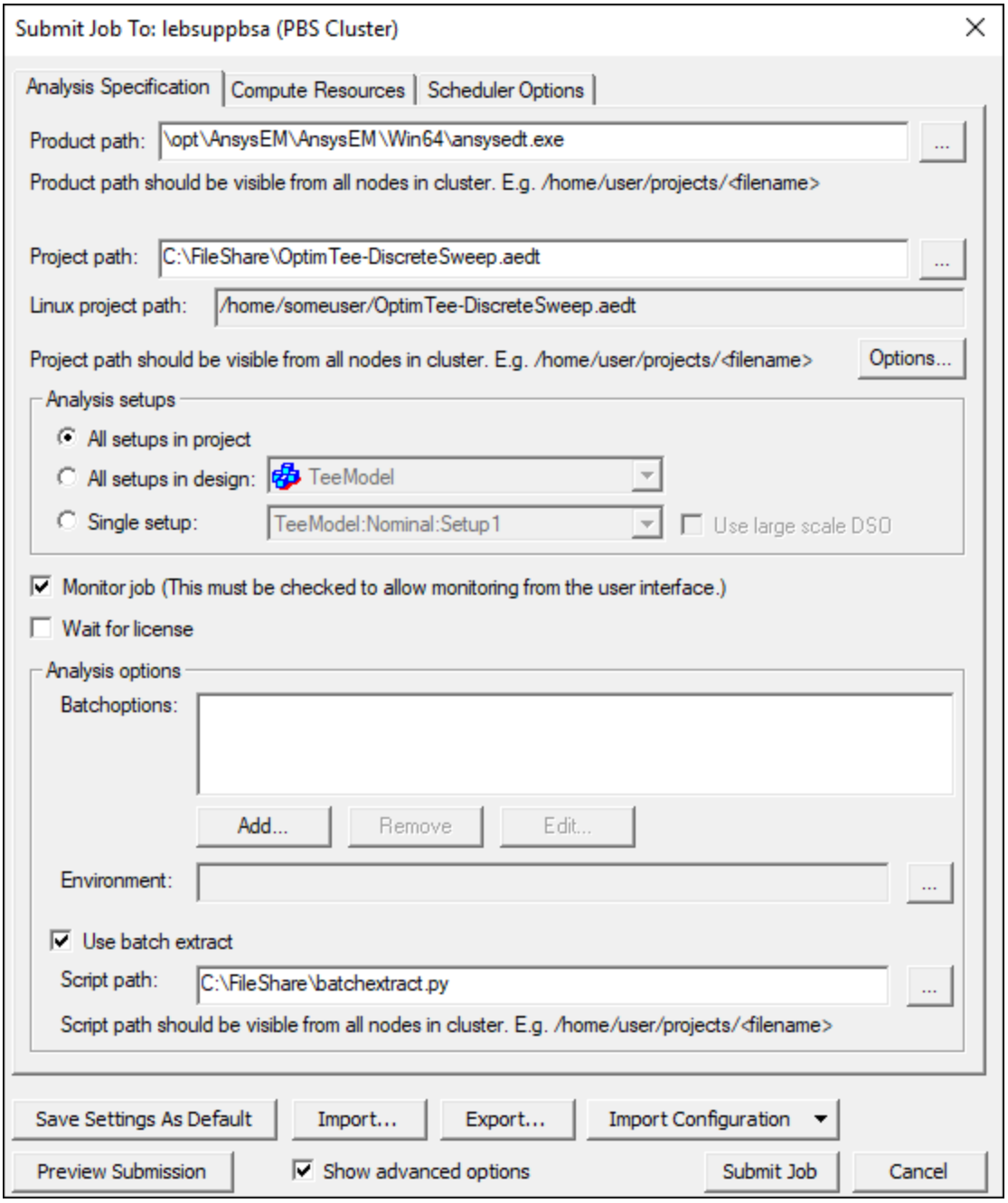
If you specify a project in an archive, the window may be used to specify the Windows pathname of the project to be extracted from the archive. If this is done, the Linux pathname of the target project is determined from the directories shared between Windows and Linux and shown in the upper portion of this window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you specify a batchextract script, the Linux path of the batchextract script is determined from the Windows path of the batchextract script and the directories shared between Windows and Linux.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



User Passwords are Encrypted

Jobs are submitted to the Linux cluster using the user name and password entered in the **Select Scheduler** window. These settings are persistent; in general, these settings need to be entered only if they change. To ensure security, user passwords are stored in an encrypted format. When a job is submitted from a Windows host using the ansoftrsmservice running on a Linux submission host, the user credentials are sent over the network in an encrypted format.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Job Submission Scripting

To help with automation, you can submit batch jobs can through script commands of the **oDesktop** object. The **SubmitJob** script command uses job submission settings that have been exported from the **Submit Job** window to an .areg file. The path to this .areg file is thus the first argument to the SubmitJob command. Additional arguments include the path to the project file, the design name (if restricting the solve to a particular design), and the setup name (if further restricting the solve to a single setup within a design).

For further automation, you can use the **SelectScheduler** scripting command to determine what scheduler to use for submission, to include options for head node, username, and whether to require password entry from the user. (If the username differs from the cached username, or the force password flag is set, then the *Select Scheduler* dialog box appears.) If there are any issues with the scheduler selection (for example, a password is required or the requested host wasn't found), then the *Select Scheduler* window appears. If Ansys Cloud Burst Computing is selected then the "Login" button will open the default browser and prompt for an AnsysID login. This is the only part of job submission scripting that may required user intervention. This same mechanism is used if, from within the SubmitJob command, there is failure to connect to the scheduler. Even though there are allowances for graphical user intervention if something goes wrong, if the password (if required) is cached and all settings are correct, the entire submission process can run non-graphically and fully automated.

Limitations

All settings besides the arguments passed to the **SubmitJob** command must be stored in the .areg file containing settings exported from the job submission window. These include (but aren't limited to) batch options, environment variables, batch extract settings, and compute resource selections. To run many job submission scripts with variation of these settings, there must be multiple .areg files available.

Note that the same project can be submitted multiple times with a single script. Care must be taken in this situation because each time a project is submitted, the state-keeping files used for monitoring are removed so that the job can create them from scratch to ensure consistency. While this ensures proper monitoring for a job that is just being submitted, it could interfere with monitoring (or even correct solving if a lock file is deleted) of a job that is already in progress. Because of this, if the same project is to be resubmitted from within a single script, the job should be monitored (waiting for completion) before trying to submit it again. This monitoring/waiting can be done with a combination of a single **LaunchJobMonitor** command followed by a loop that checks the result of a **RefreshJobMonitor** command.

How to Perform Job Submission Scripting

The typical scenario for job submission scripting would be to do the following:

1. Manually select the scheduler. Use the **Select Scheduler** window to open the **Submit Job To** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Choose a representative project (with the desired design type), and select appropriate analysis settings.
3. Make the required compute resource selections and try to preview the job.
4. If preview is successful, export the dialog settings and record the path to the new .areg file.
5. Create a script containing at a minimum a **SubmitJob** command with the path to the .areg file, and the path to the project file. Note that there must be double backslashes for each backslash of a path because the backslash is an escape character. When the script is run, and all is successful, there should be a message in the message windows stating that the job was submitted, including the job ID(s). There could be multiple job IDs if multi-step submission is used.

See the Scripting help (click **Help** > **MaxwellScripting Help**) for details on the **SelectScheduler**, **SubmitJob**, **LaunchJobMonitor** and **RefreshJobMonitor** commands.

Monitoring Jobs

There are a number of tools that allow you to monitor jobs, including the **Monitor Job** window and detailed .log files.

- Monitor Job Window
- Monitoring Ansys Cloud Burst Jobs
- Monitoring Large Scale DSO Jobs
- Web Client for Batch Solve Monitoring and Reporting

Monitor Job Window

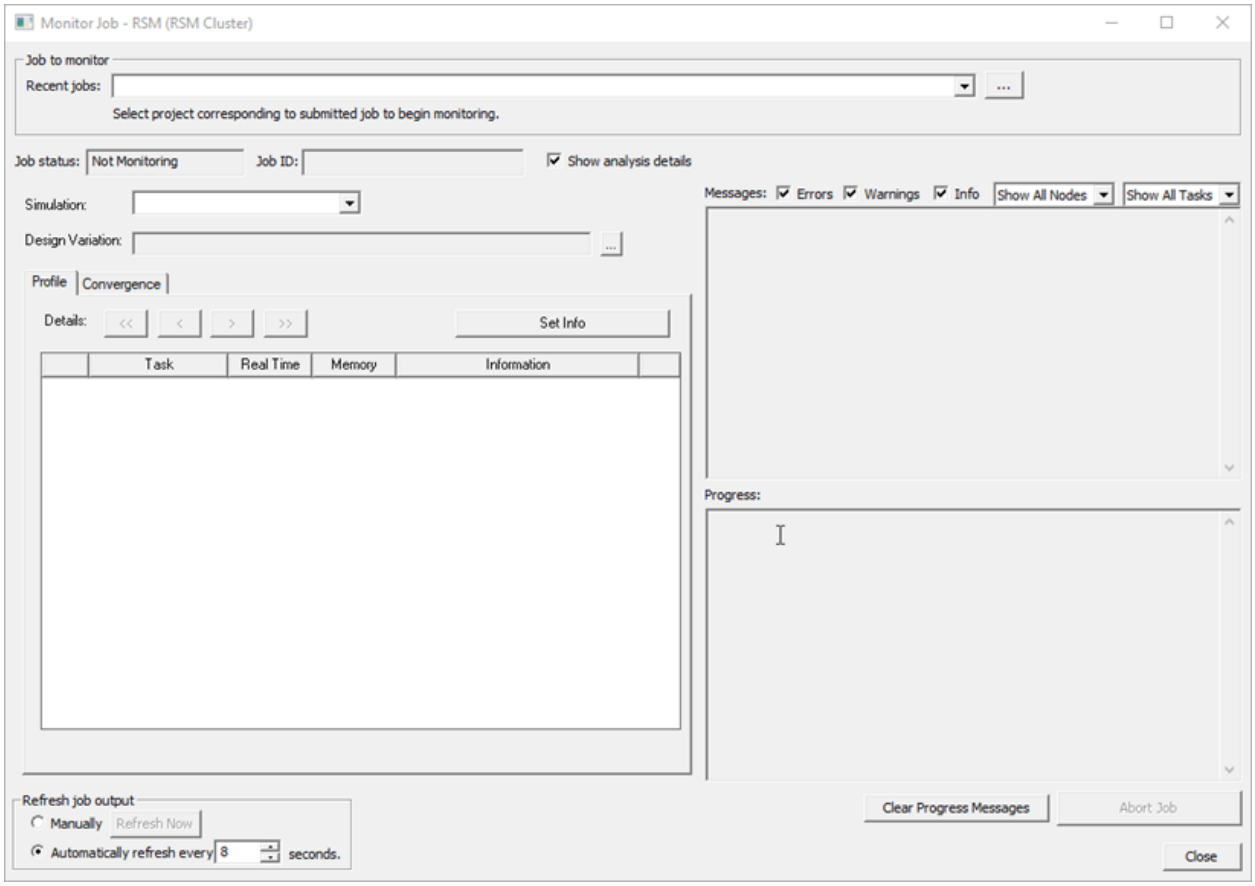
The **Monitor Job** window allows you to monitor the progress and status of jobs, including information on variations solved so far, variations currently solving, and the number of variations remaining.

To monitor jobs:

1. Access the Monitor Job window one of the following ways:
   - Click **Tools** > **Job Management** > **Monitor Jobs**.
   - Select the **Simulation** tab and click the **Monitor** icon.
   - From a Command window, use the -showmonitorjobs command.

   The **Monitor Job** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> The window may look slightly different, depending on your selected scheduler and design type.

The **Monitor Job** window contains the following areas:

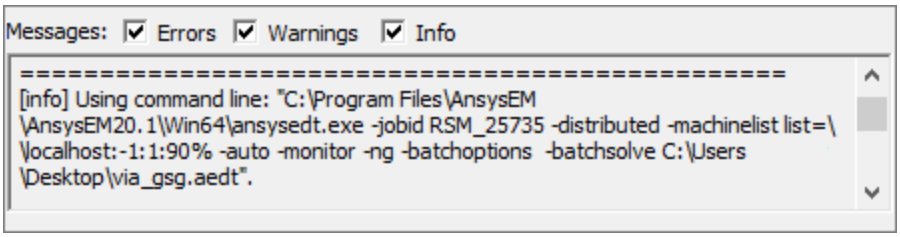
- **Job to Monitor** – Allows you to select either a recent job, or a project containing the job you want to monitor. You can select an archive.
- **Job Status** – The current job status. The normal progression is: Starting Monitoring, Queued, Running, Shutting Down, Completed. A status of Unknown may indicate a connection problem.
- **Job ID** – Identified by the scheduler prefix and job number.
- **Simulation** – Drop-down menu that allows you to select an individual simulation setup.
- **Design Variation** – Click the ellipses button (**...**) to select or deselect design variations.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

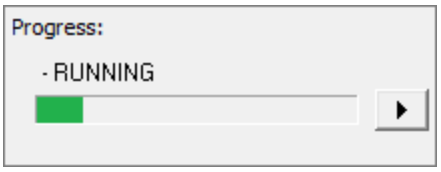
- **Profile Tab** – Displays detailed information about completed tasks, including execution time and memory usage.

Task	Real Time	CPU Time	Memory	Information
Start				Time: 10/16/2019 14:17:38; Host:: Processor: 12;
				Executing from C:\Program Files\AnsysEM\AnsysEM20.1\Win64\Q
				Performing minimal design validations
InitMesh	00:00:08	00:00:07	106 M	70131 triangles
PreProc	00:00:00	00:00:00	106 M	70131 triangles
AdaptMesh_1	00:00:01	00:00:00	107 M	78514 triangles
Solve(1p)_1	00:00:00	00:00:00	126 M	51207 unknowns
ErrorCalc_1	00:00:00	00:00:00	126 M	78514 triangles
PostProc	00:00:01	00:00:00	126 M	995 elements
Solution Process				Elapsed Time: 00:02:15
Stop				Time: 10/16/2019 15:15:30; Status: Normal Completion

- **Convergence Tab** – Displays the completed, maximum, and minimum **Number of Passes**.
- **Messages** – Displays errors, warnings, and job information. Use the check boxes to choose which of these to display.



- **Progress** – Displays progress bars when tasks are currently in progress.



The arrow button opens a menu that allows you to either **Abort** that analysis, or perform a **Clean Stop**.

- **Refresh Job Output** – Allows you to select either manual or automatic refresh for the **Monitor Job** window.

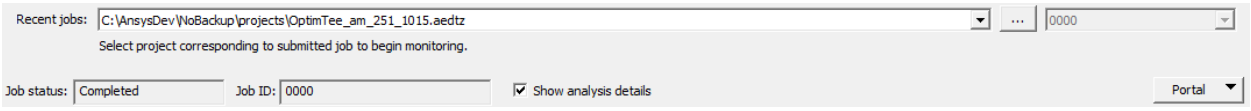
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To display less information, deselect **Show analysis details**. This removes the **Simulation**, **Design Variation**, **Profile**, and **Convergence** information, instead displaying only **Messages** and **Progress**.

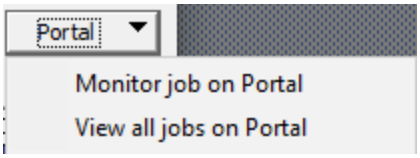
2. Click **Close** when you are done monitoring.

Monitoring Ansys Cloud Burst Jobs

When you submit a job to Ansys Cloud Burst Computing, the job status is shown in the **Monitor Jobs** window. The Job ID shown here is a short numeric identifier used to distinguish multiple jobs for the same project. Use the drop-down list at the upper right to select a different job for the same project. **The Monitor Job** window includes two additional items, **Portal**, and **Download**.

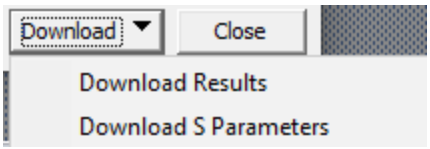


The **Portal** drop-down button provides a link to the Ansys Portal, where job details and additional monitoring information are available. For more information, see the Ansys Portal Guide.



Monitor job on Portal opens a job details view for the current job in Ansys Portal. **View all jobs on Portal** opens a job list view in Ansys Portal, enabling you to select a job to monitor.

Download allows you to download the results or s-parameters. These are saved in a folder with same name as the job ID inside the /<*project-name*>.`aedtdownload` folder (or /<*project-name*>.`aedtzdownload` folder, if the submission was based on an archive). You can also select the **Auto Download Results** check box to have the results downloaded to this location automatically.



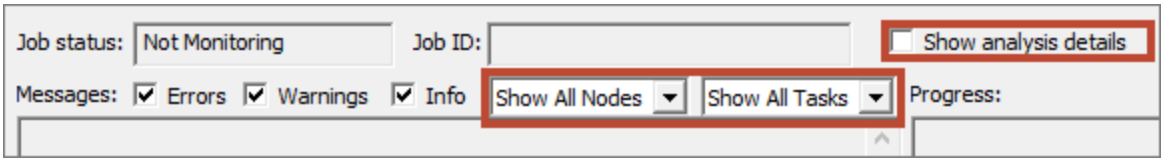
Monitoring Large Scale DSO Jobs

For Large Scale DSO submissions, the Monitor Job window includes the following items:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Show Messages in Node Tree** – Instead of viewing job details, you can select this option to view nodes and tasks in a tree format.

- Drop-down boxes allow you to filter which Nodes and Tasks to display in the tree.



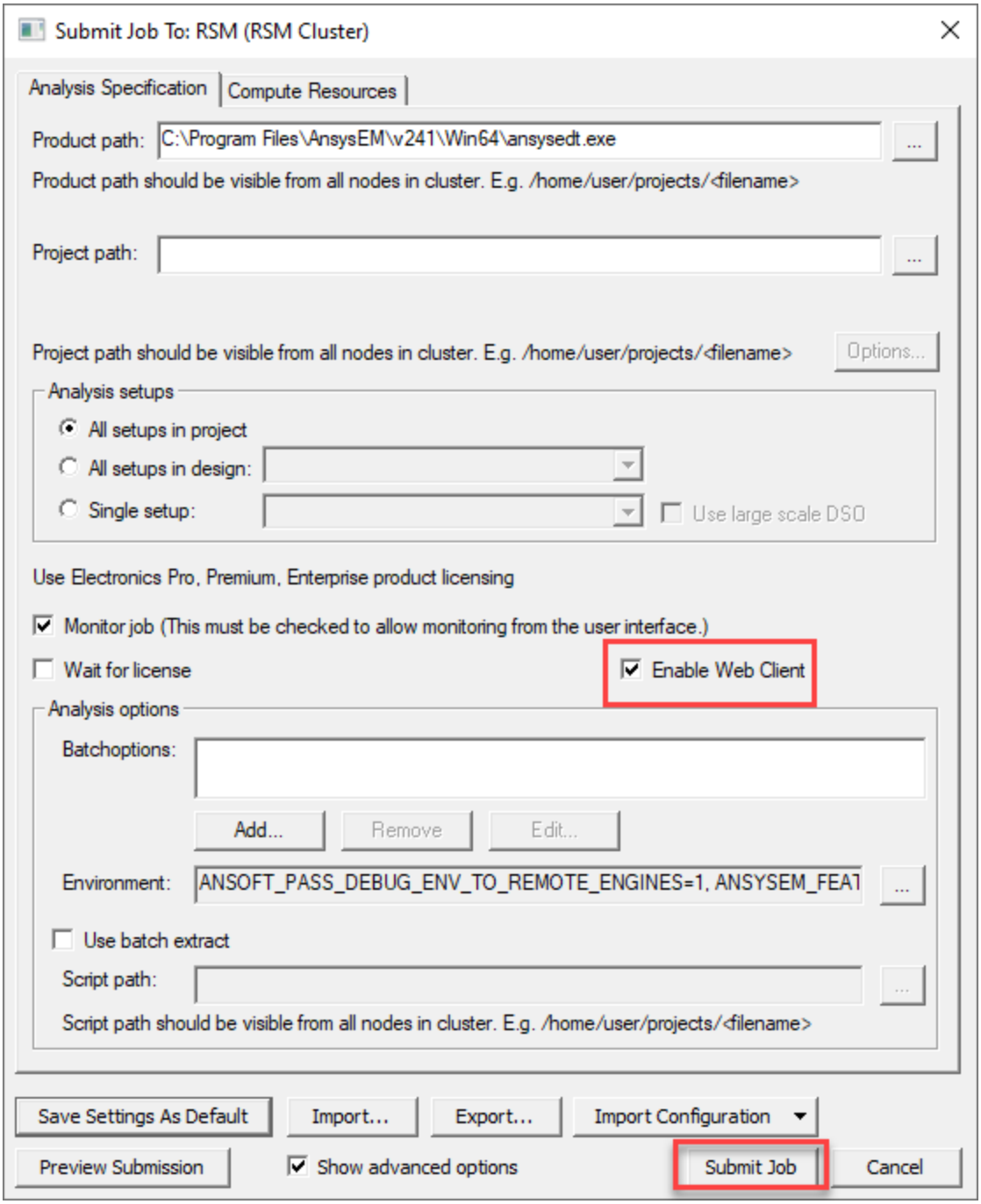
To leave this mode, select **Show analysis details**.

Web Client for Batch Solve Monitoring and Reporting

There is a Beta Option to use a Web client for batch solve monitoring and reporting. If you set the Beta Option under **Tools>Options>General Options> Desktop Configuration** for "Enable Web Client for Monitoring," you will have access to the feature.

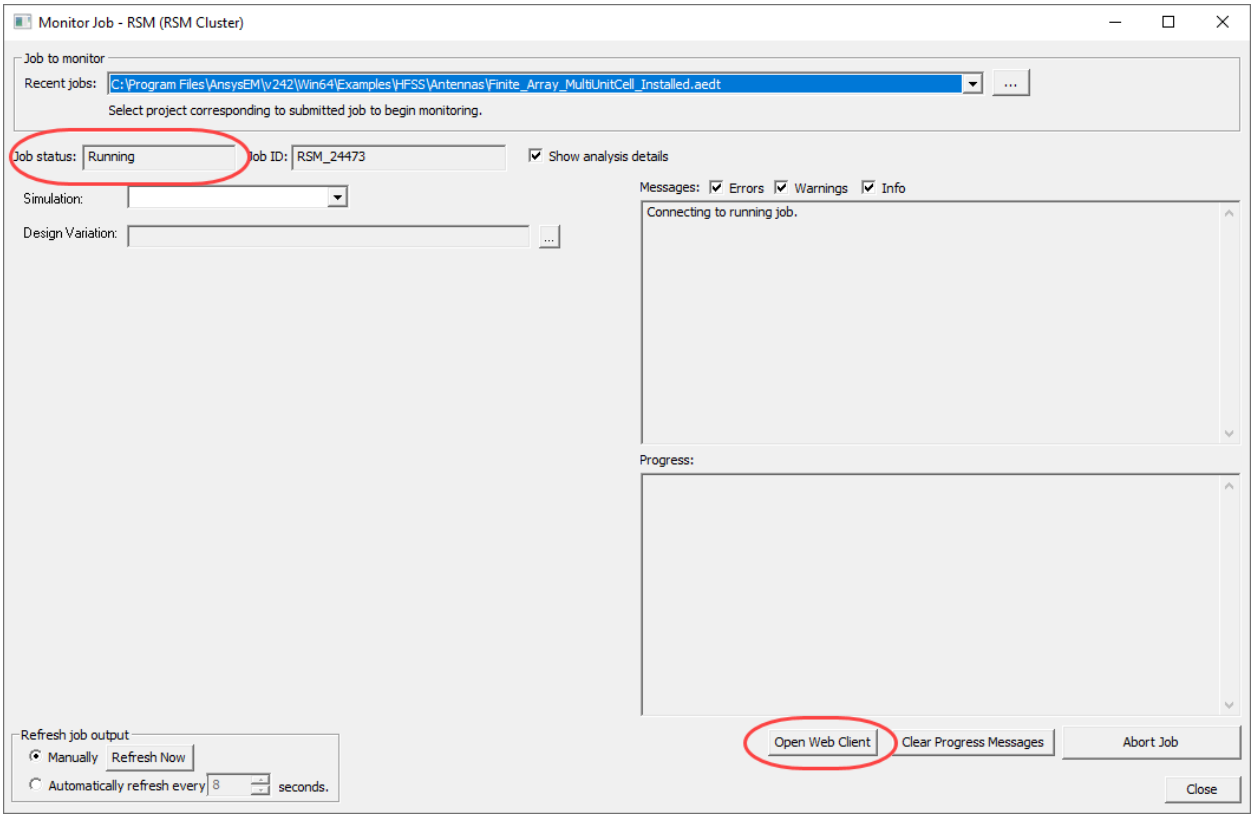
After you have selected the scheduler to use, when you access the **Submit Job To:** dialog, you well see Enable Web Client checked.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



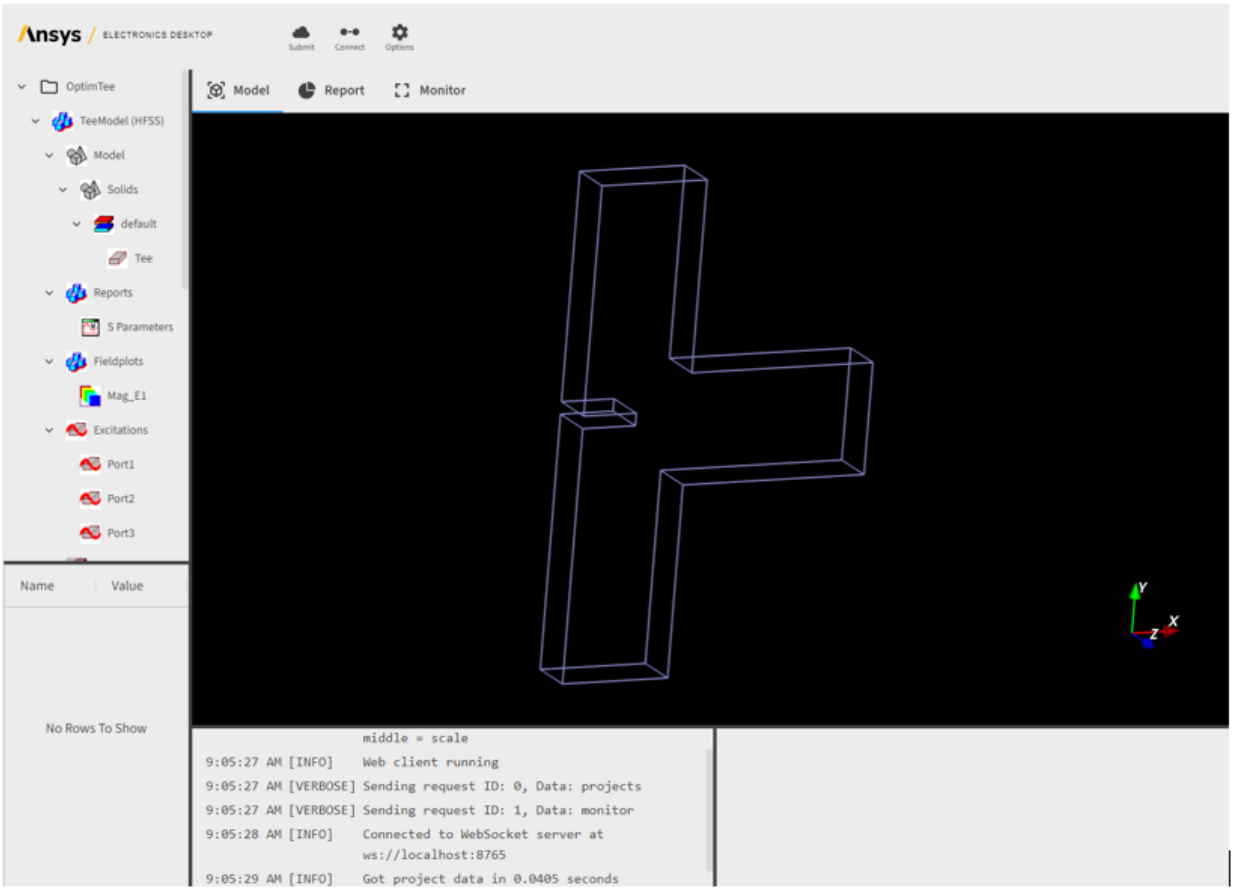
After specifying the project path and setting other parameters, you can then select **Submit Job**. This opens the Monitor Job dialog for your scheduler. Once the **Job Status** changes to Running, press **Open Web Client**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



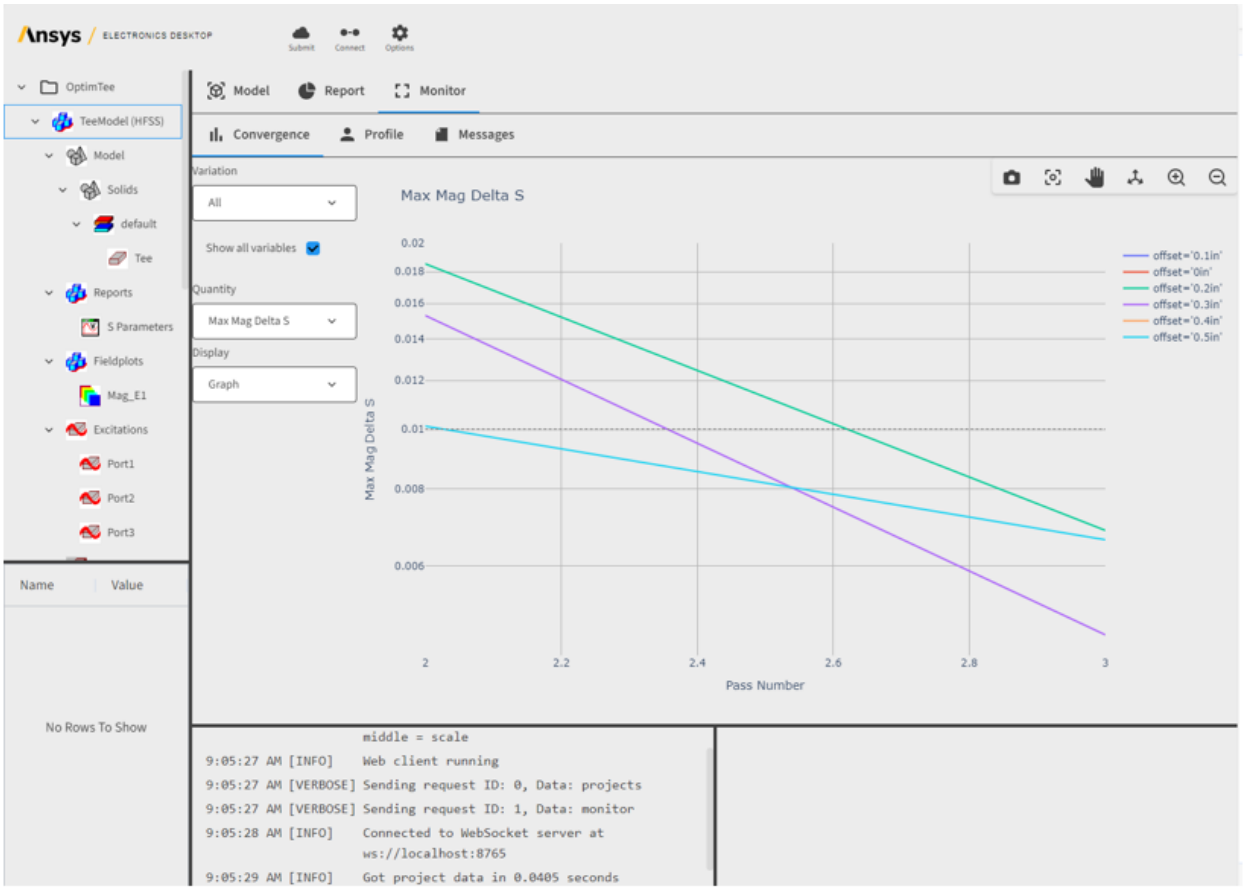
Click **Open Web Client** to open the Web Client in your default browser.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



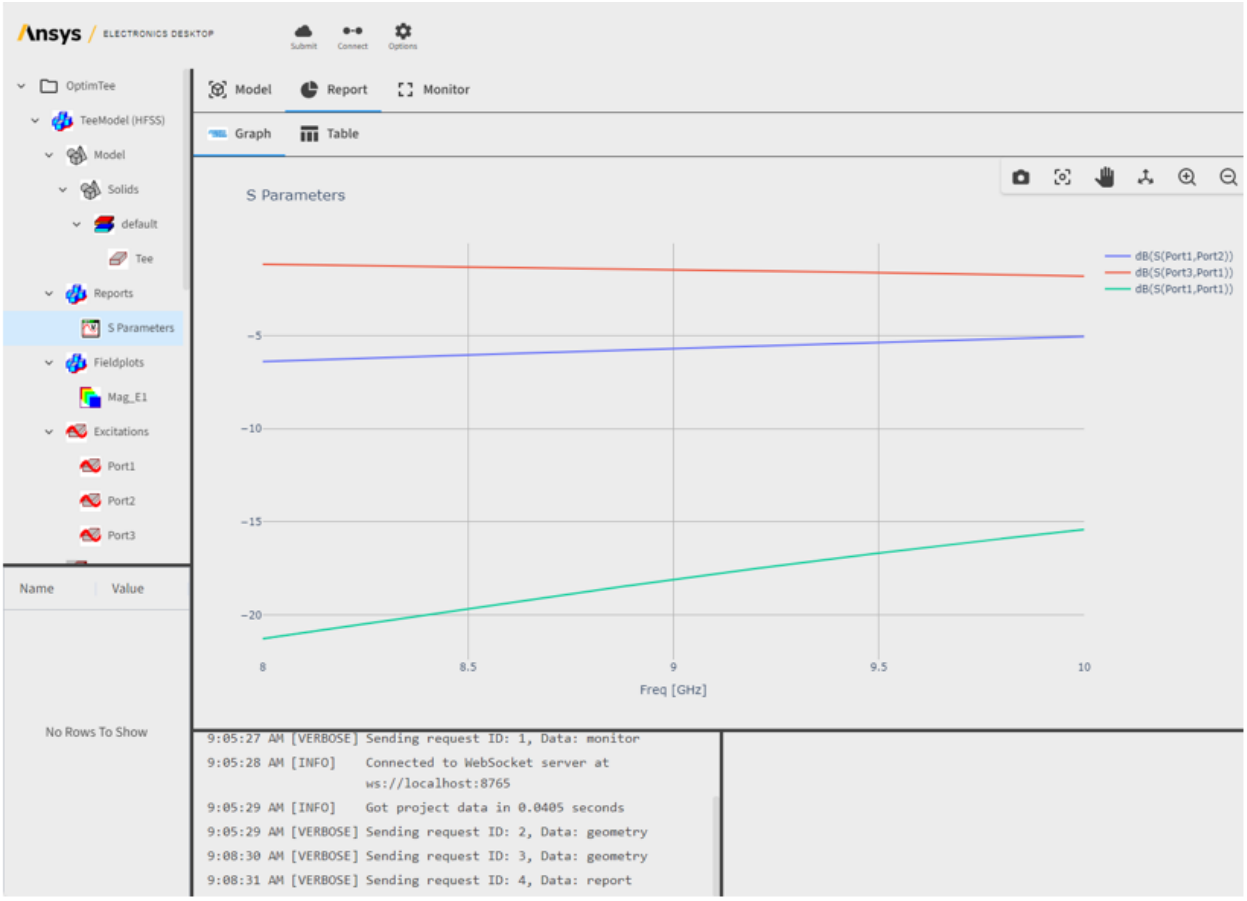
Switch to the **Monitor** tab to gain access to the subtabs for Convergence, Profile and Messages.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



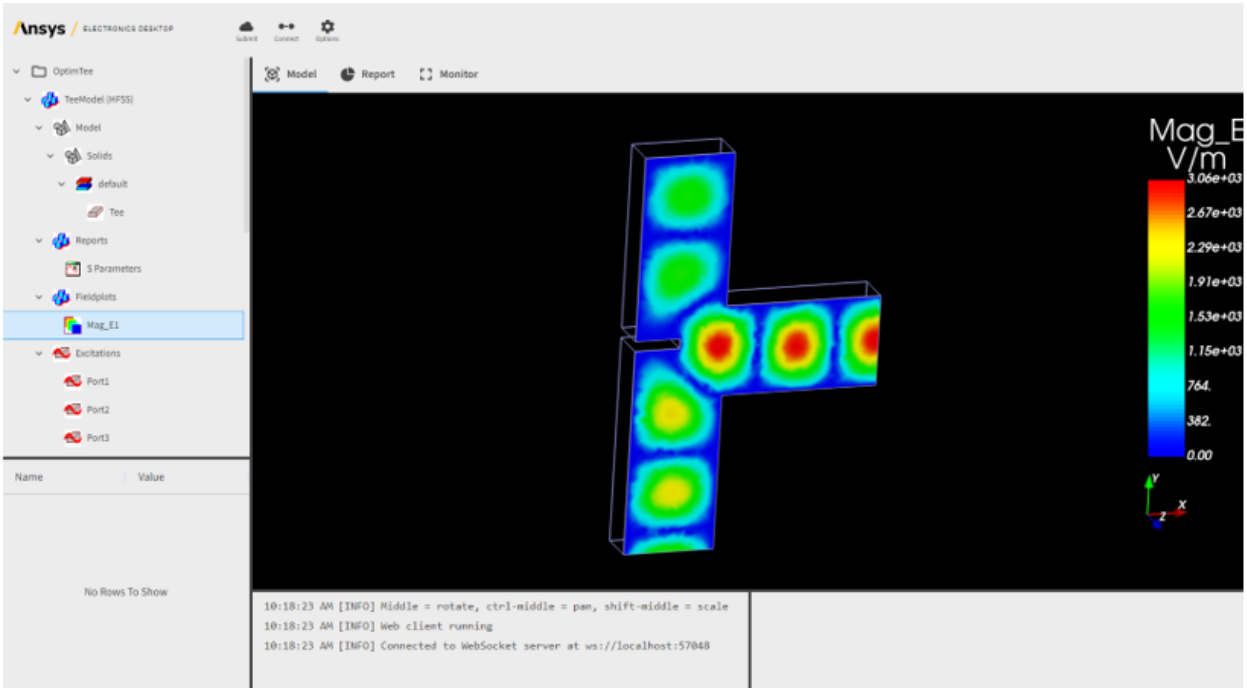
Select a Report in the **Project Tree** to view it.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Select a Field Plot in the **Project Tree** to view it.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Aborting an Analysis

To end the solution process before it is complete:

- Right-click In the **Progress** window and click **Abort** or click on drop-down button next to progress bar(s).

  The solver ends the analysis immediately.

The solver ends the analysis immediately. The data for the currently running adaptive pass, frequency point, time step, or variation is lost. Previously completed solutions may or may not be retained (depending on what process was interrupted by the **Abort** command). If you want to ensure that the results of all completed solutions are saved, use **Clean Stop** instead of **Abort**.

To abort the solution process after the current adaptive pass or solved frequency point is complete:

- Right-click the **Progress** window, or click on drop-down button next to progress bar(s) and click **Clean Stop** on the shortcut menu.

  The analysis ends after the currently running adaptive pass, frequency point, time step, or variation has been solved. Solutions completed before the stop request are retained.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you request a clean stop during the third adaptive pass, the solution for the third pass will be available once the third pass has finished solving, but the fourth pass will not run.

Ansys EM Application as an LSF Job

If you have an Ansys EM application running as an LSF job, you can use the command "bkill -s SIGTERM *jobid*" to terminate that application. Here *jobid* is the LSF job id. The response will be "Job <jobid> is being signaled". The response is the same whether the job is actually being signaled or not.

In cases where the SIGTERM parameter is ignored, the command kills the LSF job, but does not clean the lock files, and other files may not be in a consistent state.

Linux

For Linux, you can use TERM commands. Sigterm handling is done in the Desktop library. You can abort a running batchsolve by sending a TERM signal to hfss.exe.

GPU Acceleration

Several Ansys Electronics Desktop solvers can benefit from GPU acceleration:

- HFSS frequency domain, transient, and SBR+ solvers.
- HFSS 3D Layout for matrix solves.
- Maxwell 3D AC Magnetic (Eddy Current) matrix solves.
- EMIT

License Options

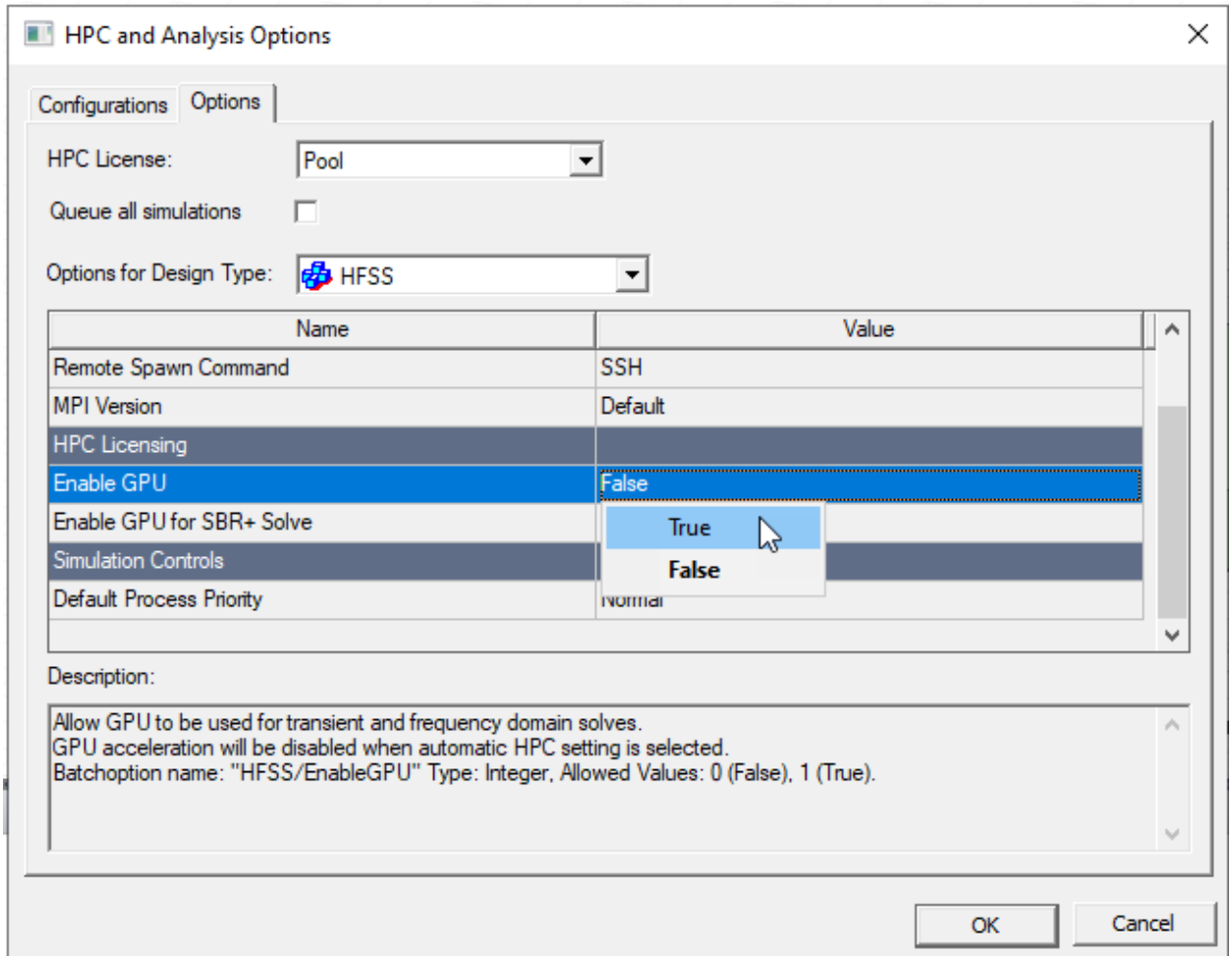
HPC licensing enables the use of cores and GPUs to accelerate simulations. In general, each core requires one unit of HPC, while each GPU requires eight units. The selected HPC license type determines which license is used, and how units of HPC are converted to license counts. For details, see the discussion of licensing under Options in Setting HPC and Analysis Options.

If the number of simulation jobs exceeds the number of GPUs in a system, the excessive jobs will fall back to CPUs and will be accelerated by up to 8 CPU cores for each job.

Enable/Disable GPU Acceleration from the Ansys Electronics Desktop Solver User Interface

To turn on or off GPU acceleration for supported solvers, click **Tools** > **Options** > **HPC and Analysis Options** and select the **Options** tab. Click on the value of Enable GPU and toggle it to either True or False. HFSS includes a separate toggle for Enable GPU for SBR+ Solve.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



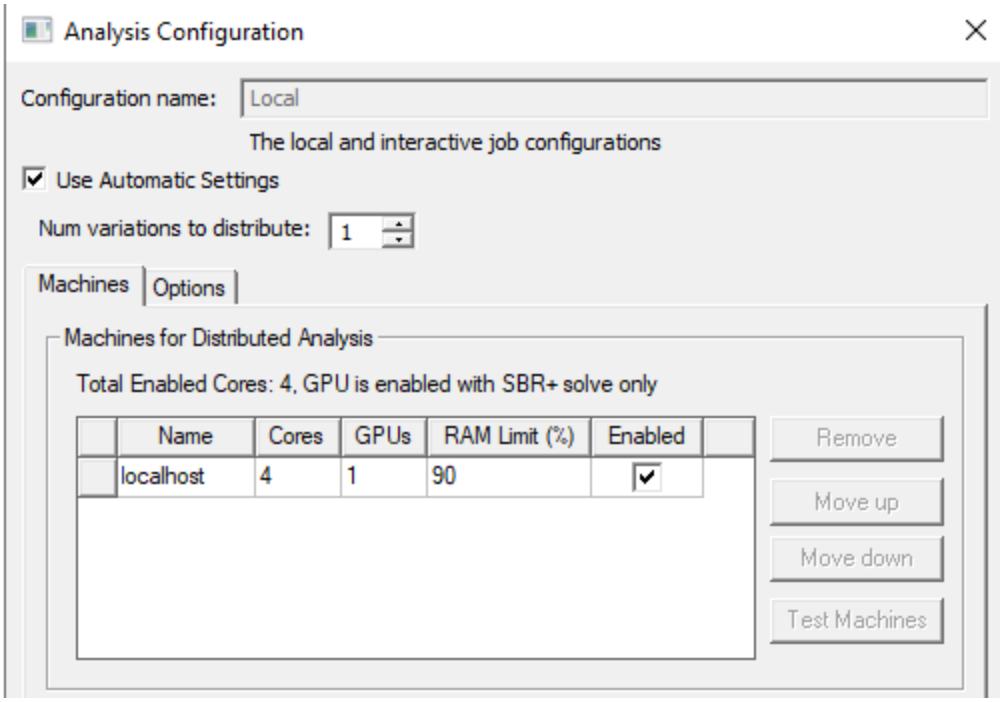
Enabling GPU-Distributed

There are two steps for setting up GPU-distributed.

1. Go to **Tools** > **Options** > **HPC and Analysis Options** and click **Edit** in the dialog box, or select the **Simulation** tab of the ribbon, and click **Analysis Config** to edit Analysis Configurations.

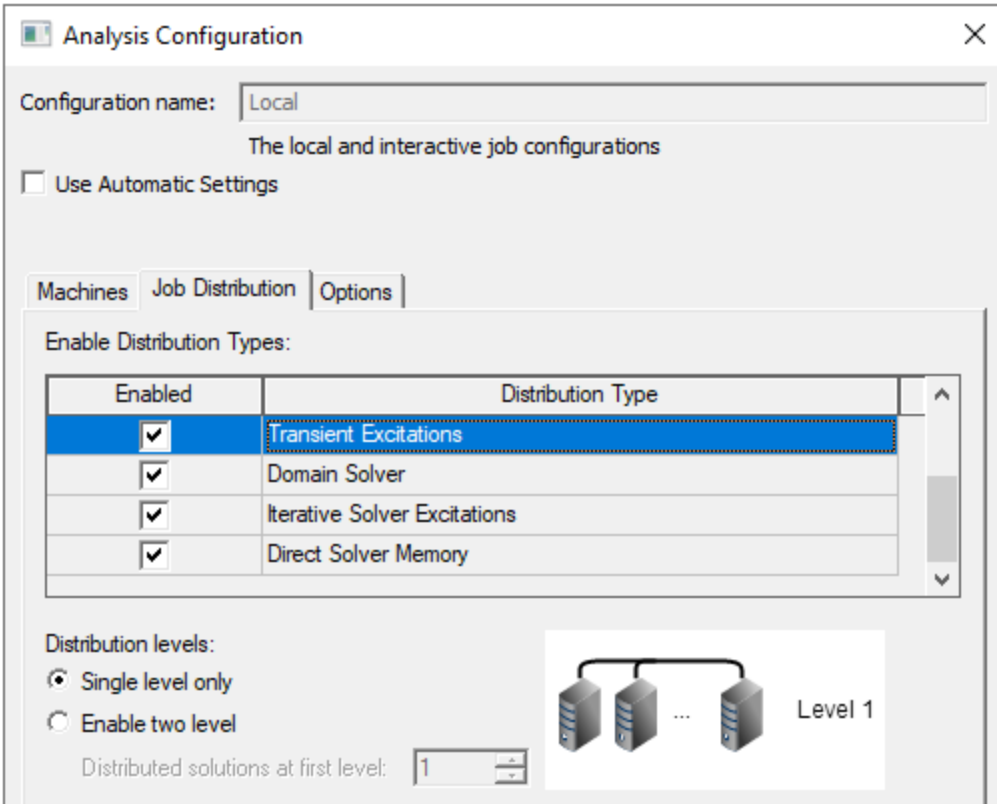
   If Use Automatic Settings is checked and the solver is HFSS, a message notes that only SBR+ has GPU Enabled.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Uncheck **Use Automatic Settings**, select the **Job DIstribution** tab, and select the appropriate Distribution Types for the solver. For example, GPU-DSO in HFSS Transient can be used either for parametric sweep (Variations) or Transient Network analysis of multiport networks (Transient Excitations).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

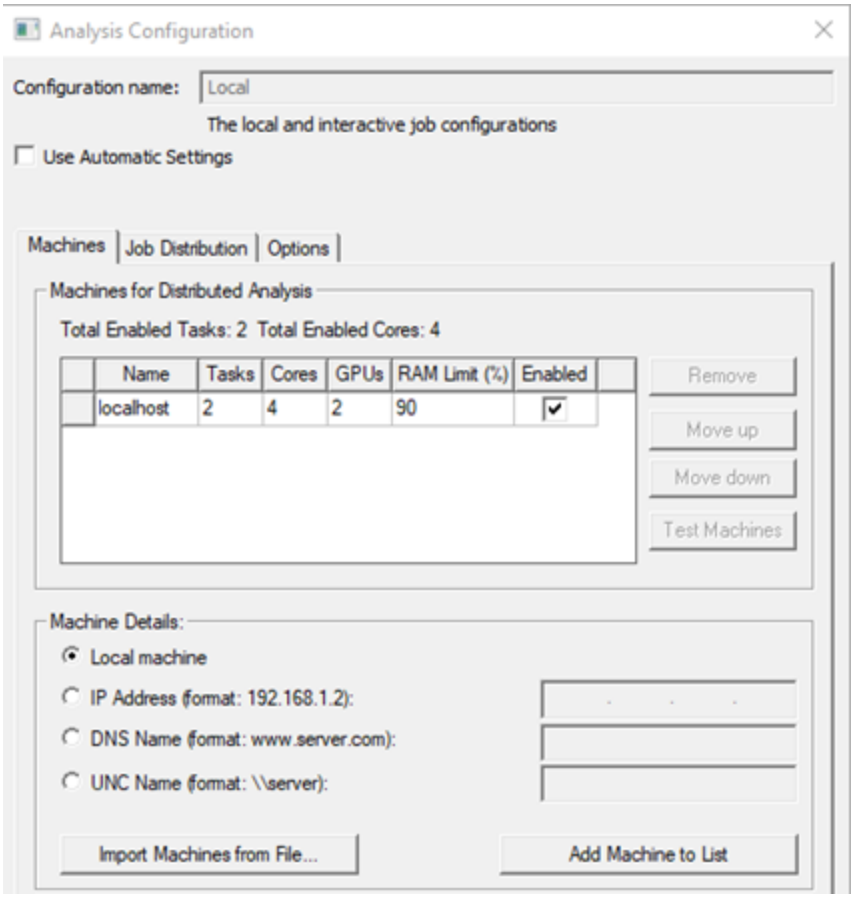


2. Set up the Machines for GPU-Distributed.

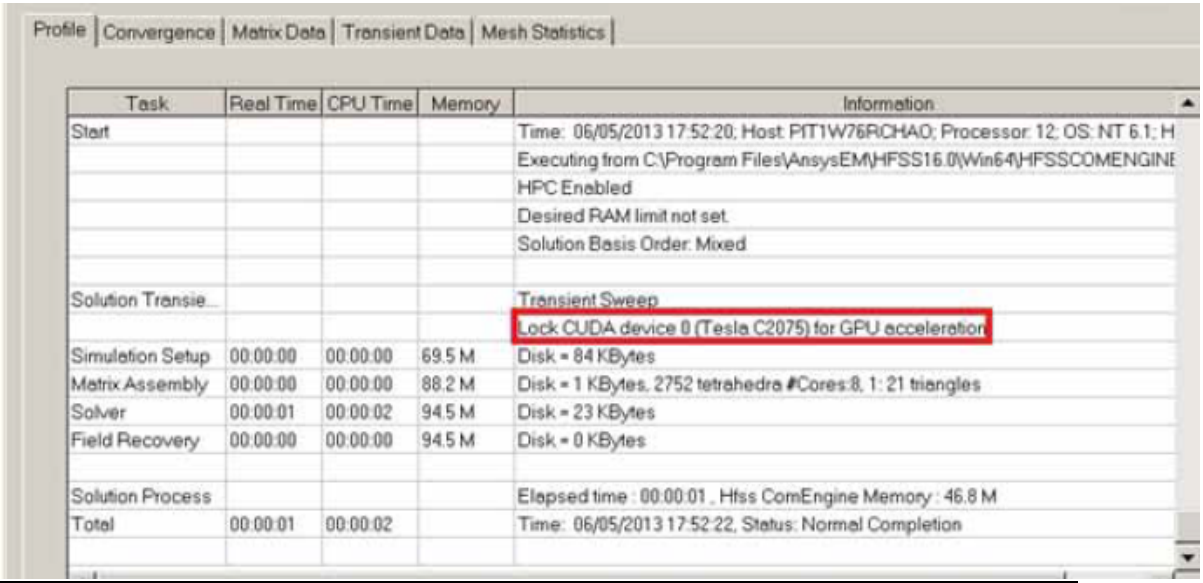
   In the following figure, two simulation jobs are allowed to run on the local host, or up to two GPUs can be used for acceleration depending on the availability of licenses and GPU

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

cards.

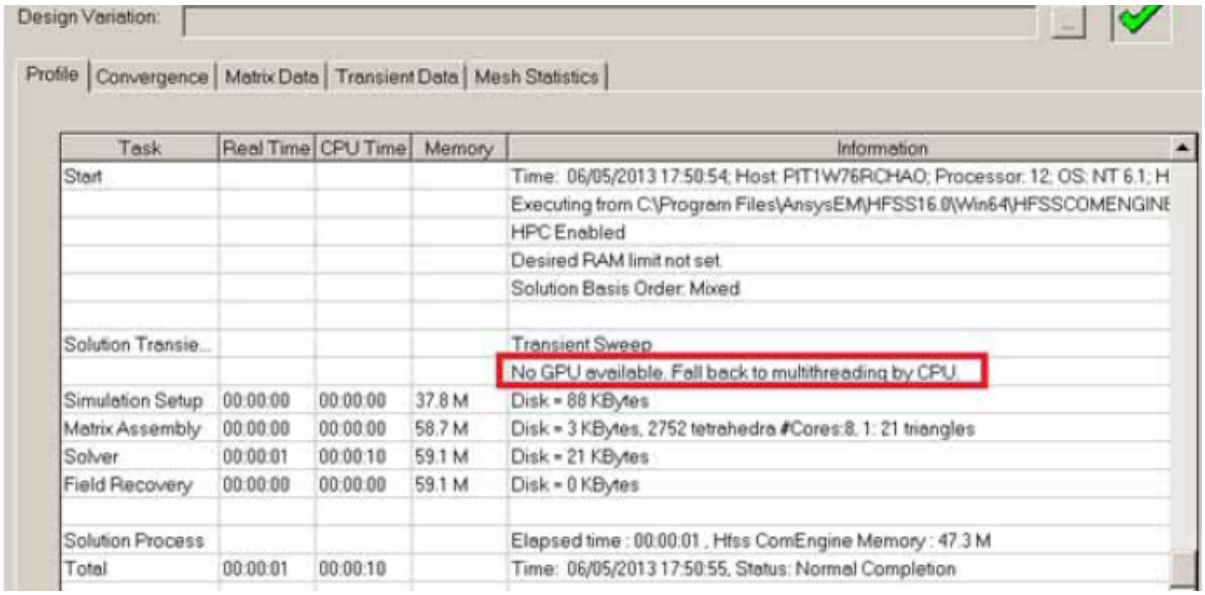


Whether GPU is used for acceleration can be checked by viewing the **Solutions** dialog box, **Profile** tab. If a GPU is successfully locked for the use by an hf3d process, the profile will show the GPU's CUDA device ID and its name.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Otherwise, the profile will indicate the fallback to CPUs. More information about why GPUs are not available for acceleration can be found in the log file. See further discussion under Step 5 of Setting HPC and Analysis Options.



Enable/Disable GPU Acceleration from the Command Line

GPU acceleration can be toggled by the `-batchoptions` command line argument:

`EnableGPU=[0/1].`

For example, the following command turns on GPU acceleration.

```
ansysedt.exe -batchsolve -batchoptions "'EnableGPU'=1"
projectname.aedt
```

Integration with Ansys Remote Simulation Management (RSM)

Ansys Electromagnetics supports its own Remote Simulation Management (RSM) software along with other High Performance Computing (HPC) software management programs (see High Performance Computing (HPC) Integration).

When do you need RSM?

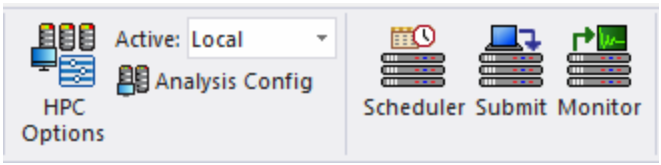
RSM is generally required if you want to run remote or distributed simulations. However, if you have a separate scheduling system that Ansys Electromagnetics supports, and you plan to run batchsolve simulations only, you may not need to install RSM. For details of installation and configuration of RSM, see the *Ansys Electromagnetics Installation Guides*.

Job Management UI for RSM

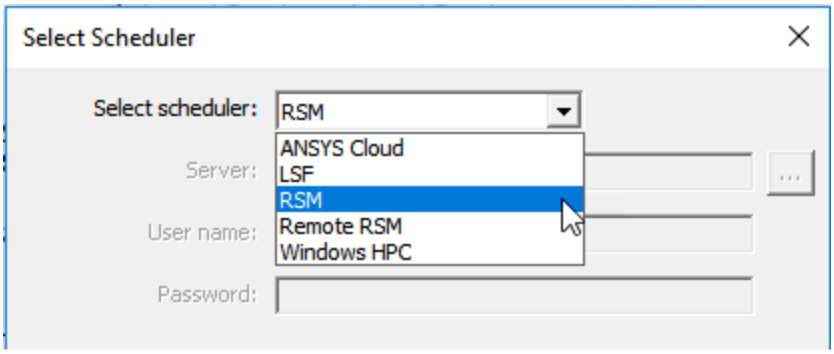
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can use the Job Management UI to submit batch jobs to RSM. The Job Management UI is accessed by running Ansys Electronics Desktop on the designated 'Postprocessing node' of the cluster. The Desktop provides UI commands for Scheduler selection, Job submission and Job monitoring/control. You access the Scheduler User Interface by clicking **Tools** > **Job Management** > **Select Scheduler...**

You can also select the **Simulation** tab of the ribbon, and click the **Scheduler** icon.



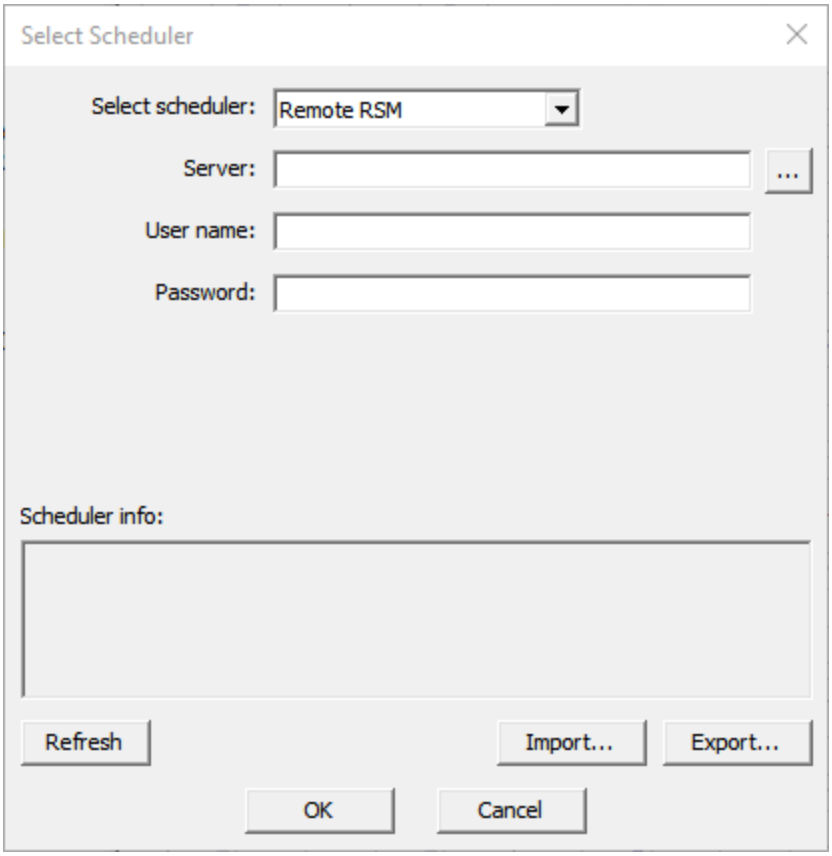
This displays the selection dialog box. The **Select scheduler** drop-down menu lists potential schedulers (which can include RSM, LSF, Windows HPC, or SGE, depending on the environment).



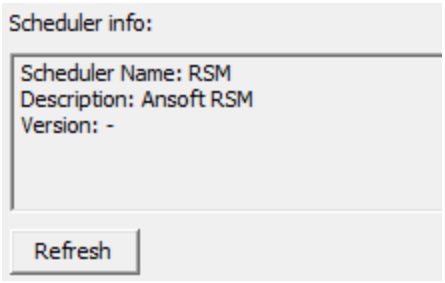
If you select a scheduler that is not supported in your environment, you receive a warning.

If you select Remote RSM and your environment has been configured , you can select a computer, user name, and password.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After selecting a scheduler, click **Refresh** to display information for that scheduler.



Once you have selected a scheduler supported in your environment, you can go through the following steps to submit a batch job.

1. Set up and prepare model on local workstation.
2. Copy the input project (or folder, if the project references external files) from a personal workstation to a shared-drive on cluster (say project is copied to /home/projects/spool/test.adsn).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- In the RSM environment, you are required to specify a machine-list. See: HPC and Analysis Options. For example, say the machine-list is: 3 cores from 'm1' and 3 cores from 'm2', for a total of 6 engines. You select the list on the **Compute Resources** tab of the **Submit Job to RSM** window, as described below.

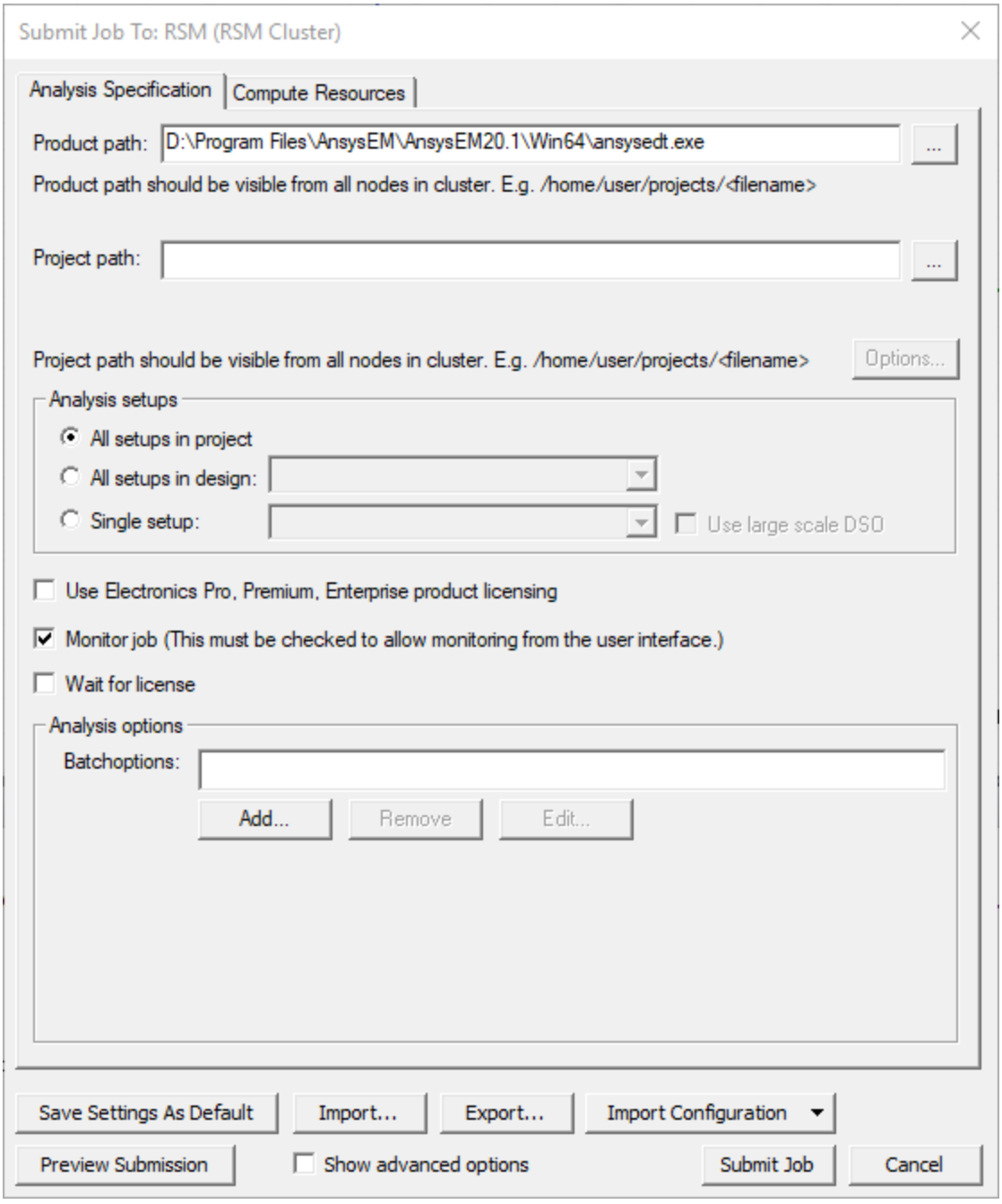
3. Open a remote-desktop session (or equivalent such as vnc session) on the node corresponding to the first machine of job's machine-list, 'm1' in this case. Launch Desktop graphically on 'm1'.
4. After setting the job submission node, select **Tools** > **Job Management** > **Submit Job...** or **Project** > **Submit Job...** or **[ProductName]** > **Submit Job...** to open the **Submit Job To:** window. You can also access **Submit Job** from the shortcut menus for the Project Name, Design name, Analysis Setup, or Optimetrics Setup.

   The **Submit Job To:** window contains two tabs:

   - **Analysis Specification**– specify the Product path, Project name, the setups, and analysis options such as batchoptions, or, for advanced users, Environment variables. If you select the Analysis or Optimetrics setup, the Analysis Specification is pre-populated.
   - **Compute Resources** – specify the amount of compute resources and how to select specific resources from those available, and automatic settings, if supported by the design types.

   The standard Job Submission window displays.

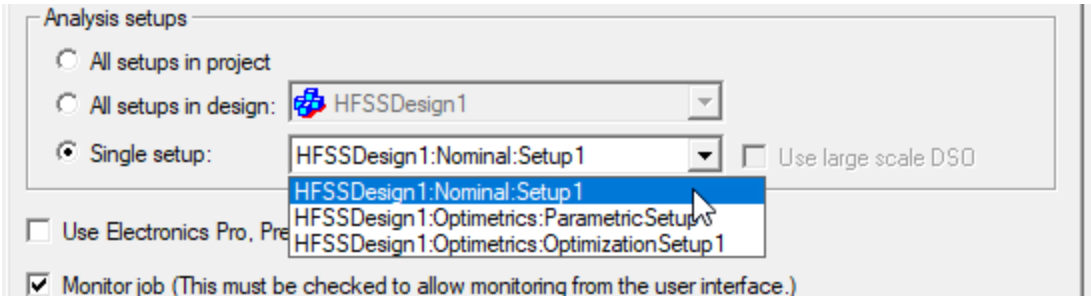
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Use the ellipsis button [**...**] to open a browser to select the project. The project can be an archive.
6. In the **Analysis setups** area, you can select radio buttons for **All setups in the project**, **All setups in the design**, or a **single setup**. For instance, the OptimTee example includes setups for Nominal, Parametric, and Optimization. If you accessed the **Submit Job** window from the right-click menus for Setup or Optimetrics Setup, this field can be

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

pre-populated.

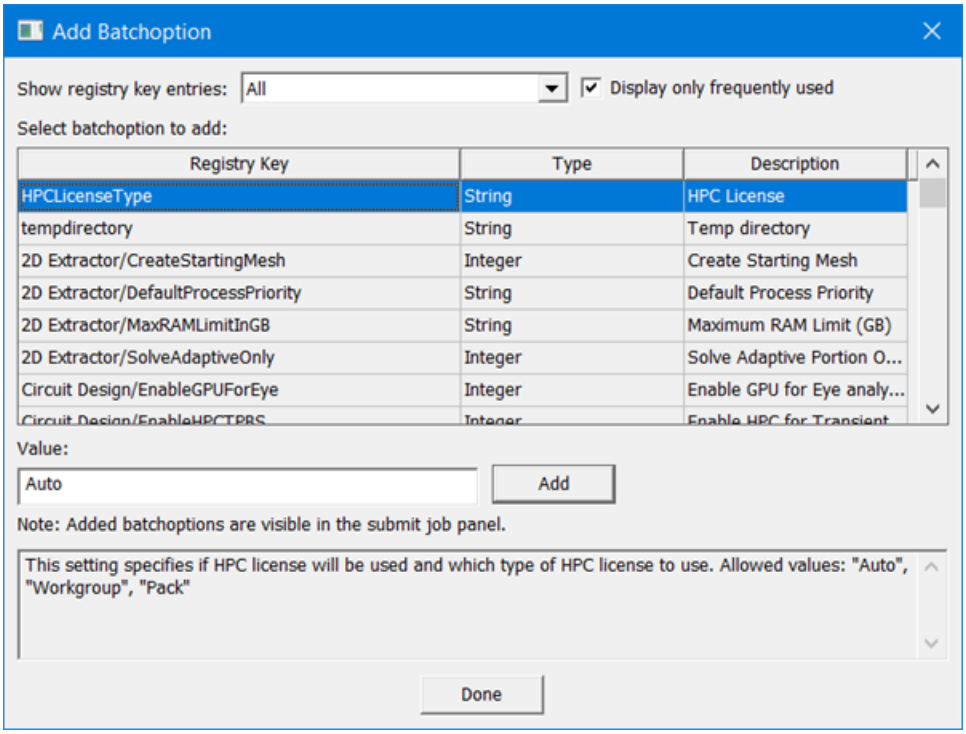


For Parametric setups, you have the option to select **Use Large Scale DSO**. For details on how and when you use this feature, see Job Management Interface for Large Scale DSO.

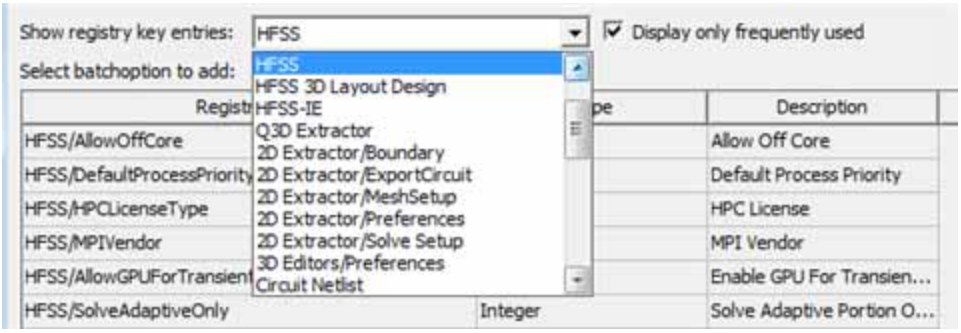
7. The Analysis options include check boxes for Monitoring the job, whether to wait for a license, and a field for adding Batchoptions via a graphical interface, or as text.
    - If you intend to monitor the job through a user interface, you must select **Monitor job**. You can then monitor this job through the **Tools** > **Job Management** > **Monitor Jobs...** command or by checking the dialog that opens when you submit the job.
    - The Batchoptions field allows you to add additional -batchoptions parameters, either as text, or by using a dialog with selection menus. Click the **Add** button to

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

view the **Add Batchoption** dialog box.



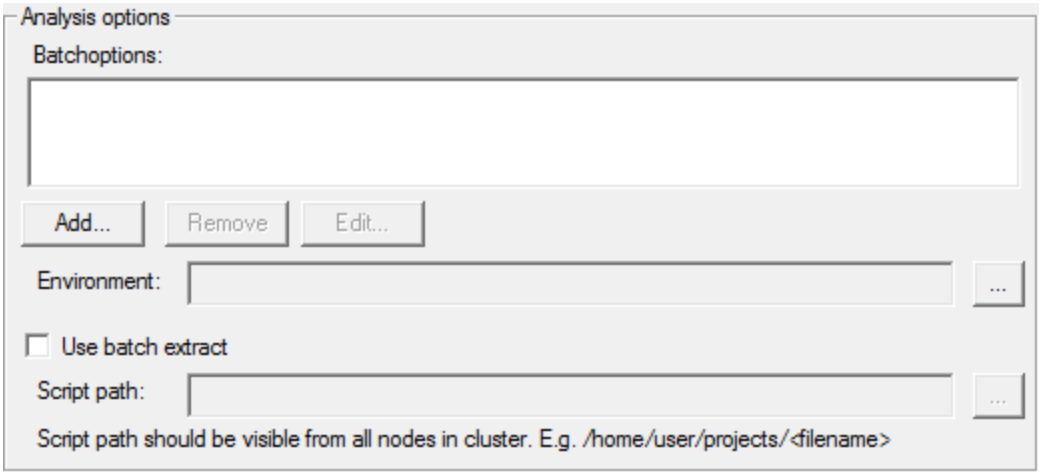
The **Show registry key entries** field lets you filter the entries displayed, by means of drop-down menu selection, and a check box to **Display only frequently used entries**.



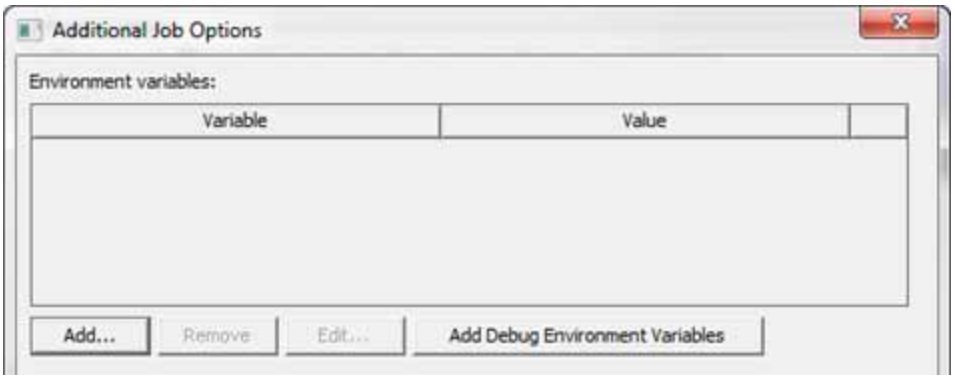
- When you have selected a batchoption, you can type the value in the field, and click the **Add** button to add the option to the batchcommand.
- In the **Submit Job To:** window, you can enable **Show advanced options** to display additional fields for environment variables, and whether to **Use batch**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

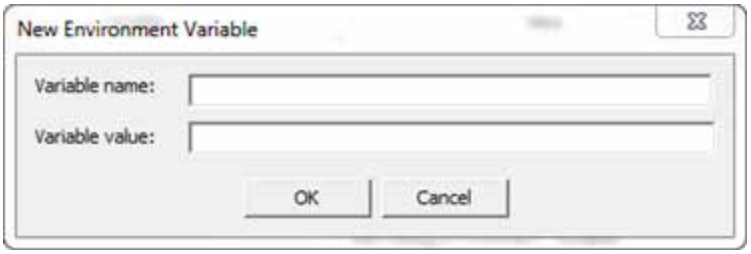
extract.



The Environment field is for environment variables, for instance, for debugging features or other variable controlled features. Click the ellipsis [**...**] button to open a dialog box for **Additional Job Options**.

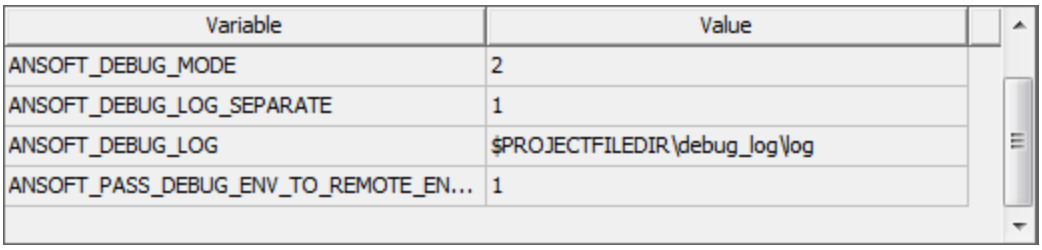


The **Add...** button opens a **New Environment Variable** dialog box in which you can include a variable name and value.



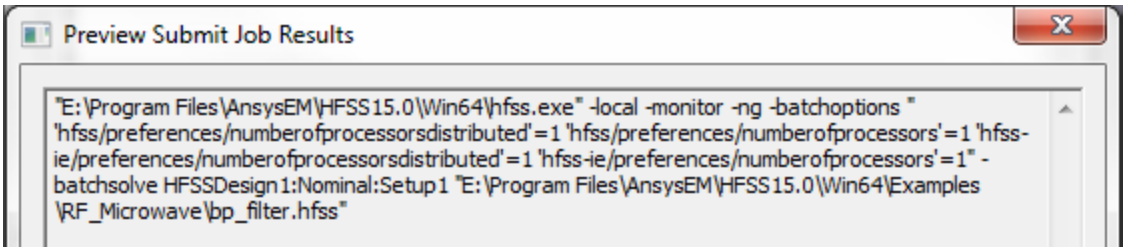
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clicking the **Add Debug Environment Variables** button automatically adds a set of debug variables. This can be useful in working with Ansys Application Engineering support.



Selecting a variable in the dialog enables the **Remove** and **Edit** buttons. The **Edit** button opens a dialog box where you can change the variable and value.

8. To see the command-line to be submitted to the scheduler, click **Preview Submission.** This opens a dialog box showing the command to be sent to the scheduler.

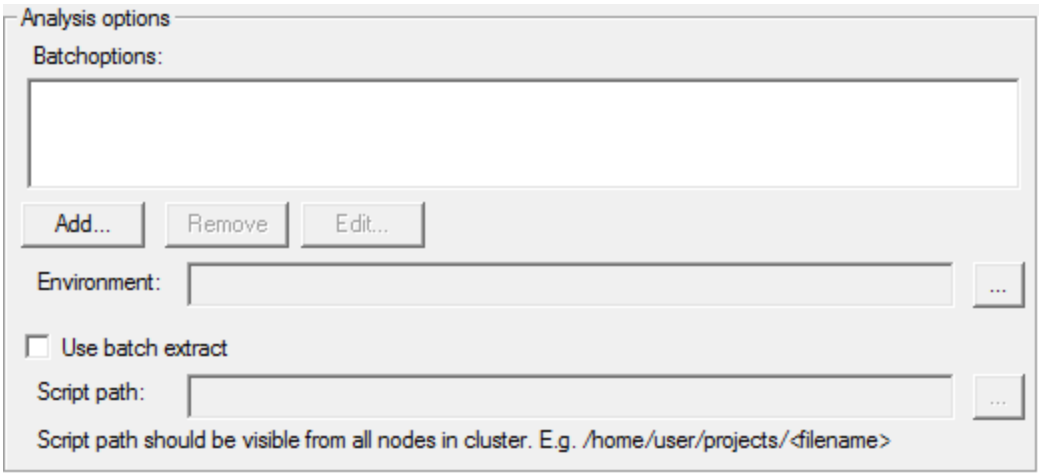


The text can be copied to the clipboard, if desired.

Use Batch Extract for RSM

Selecting **Show advanced options for RSM** also shows the Use batch extract fields.

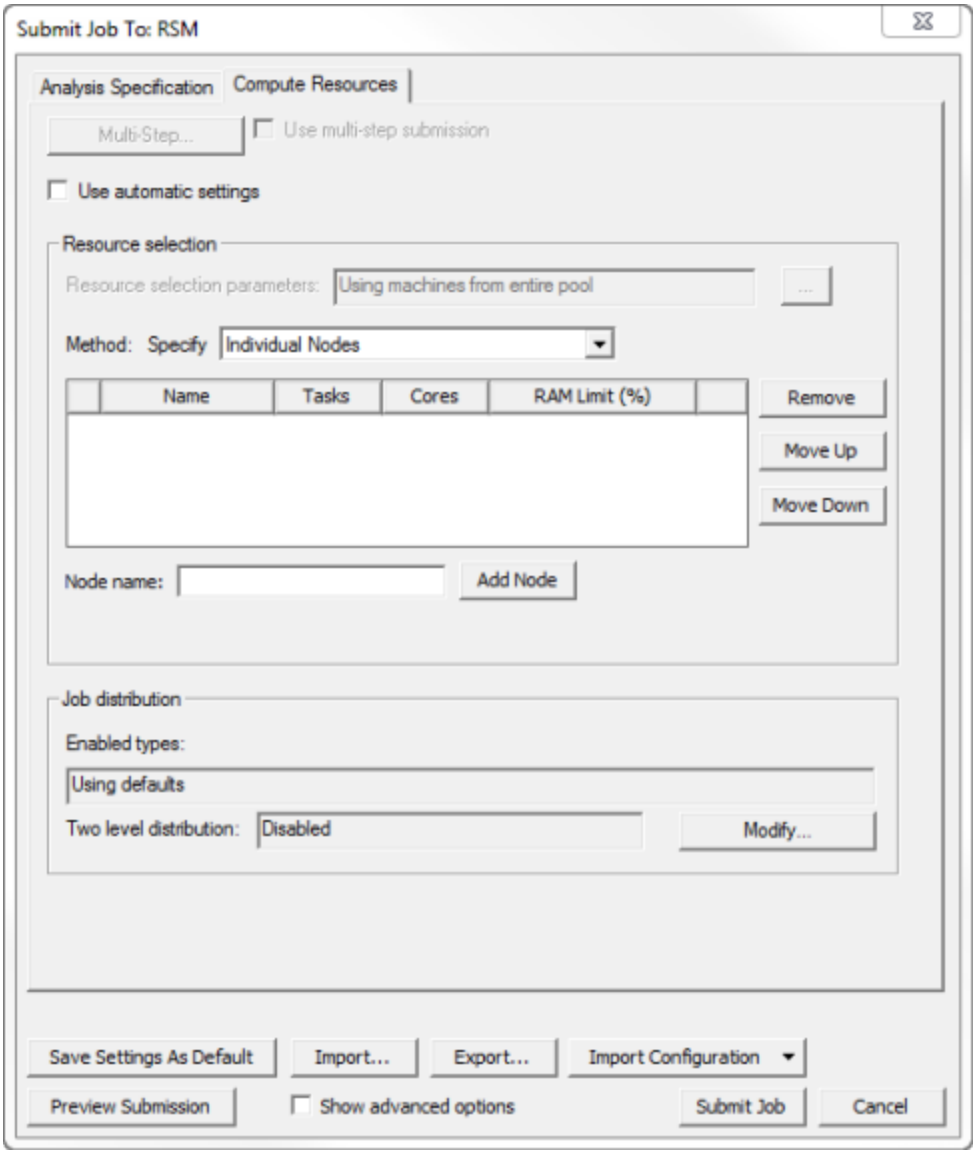
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



See the discussion on Running Ansys Electronics Desktop from a Command line for a discussion of the solve information available through batch extract.

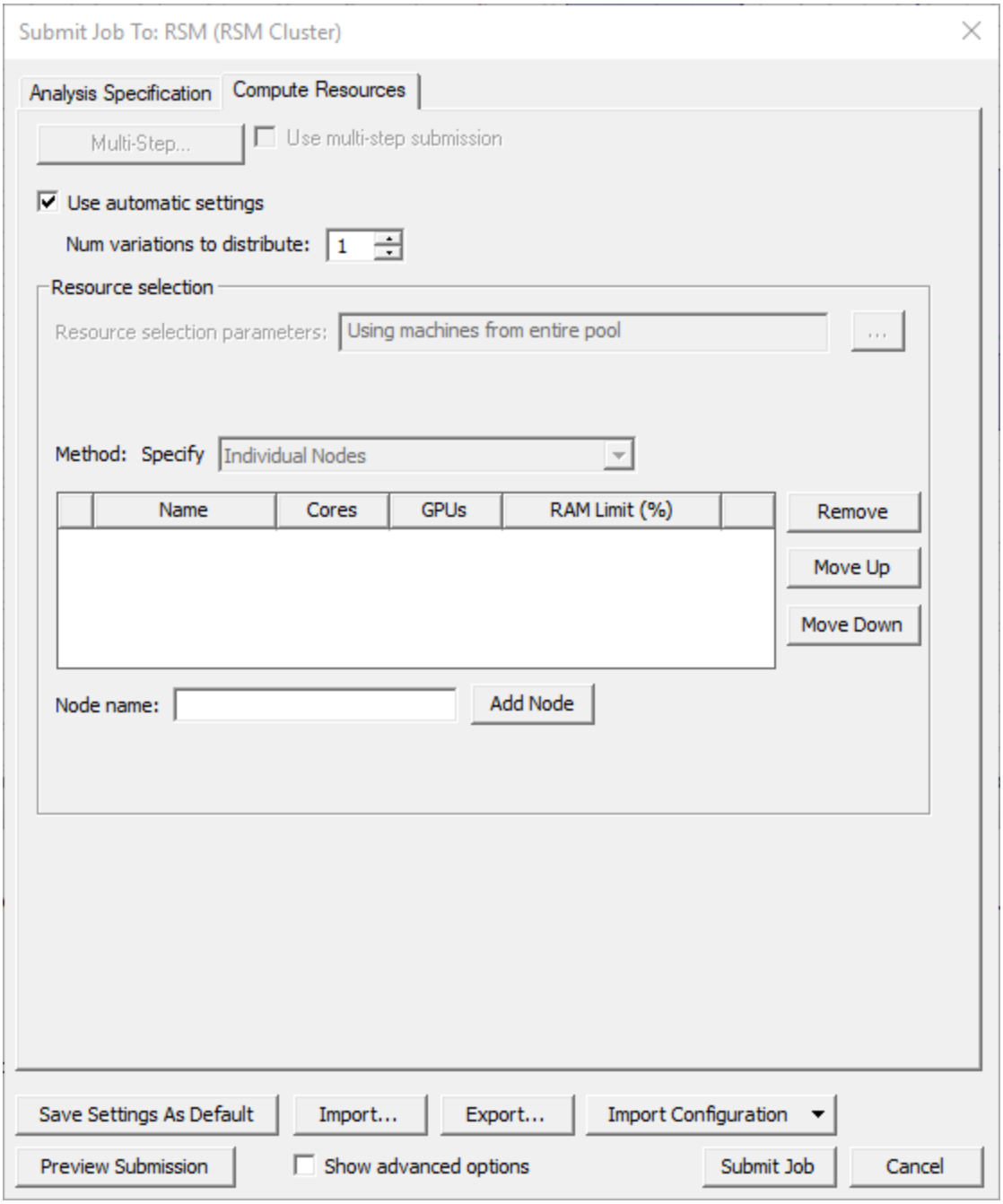
9. The **Compute Resources** tab of the **Submit Job to: RSM** window displays other parameters. Depending on the resources available for a scheduler environment, some of the fields may be disabled.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



With **Use automatic settings** selected, the **Job distribution field** is removed and the **Use automatic settings** check box and **Num variations to distribute** field appear.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



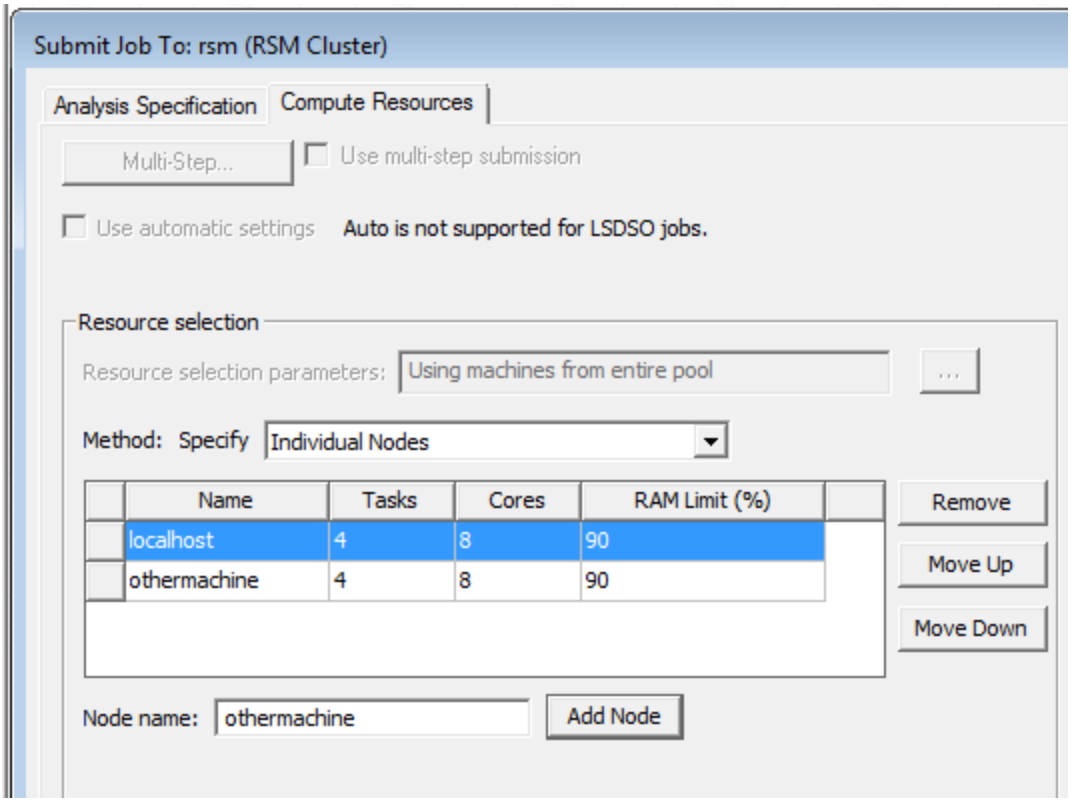
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> If you select **Use automatic settings** with **Num variations to distribute** set to 1, Optimetrics variations will be solved sequentially. Other distribution types will be distributed automatically. It does distribute frequencies, domains, and use of multiple level domains. If you set **Num variations to distribute** to 2 or more, Optimetrics variations will be solved in parallel. Other distribution types will be distributed automatically.

Otherwise:

- Specify node list

  Here you can specify a node list. In a computing environment where the available cores are not uniform, you can use this to control which resources your job will use. For use with Large Scale DSO for RSM, for jobs that are submitted from job submission panel, localhost must be the first node in the resource selection node list, other wise LSDSO solve with RSM will fail.

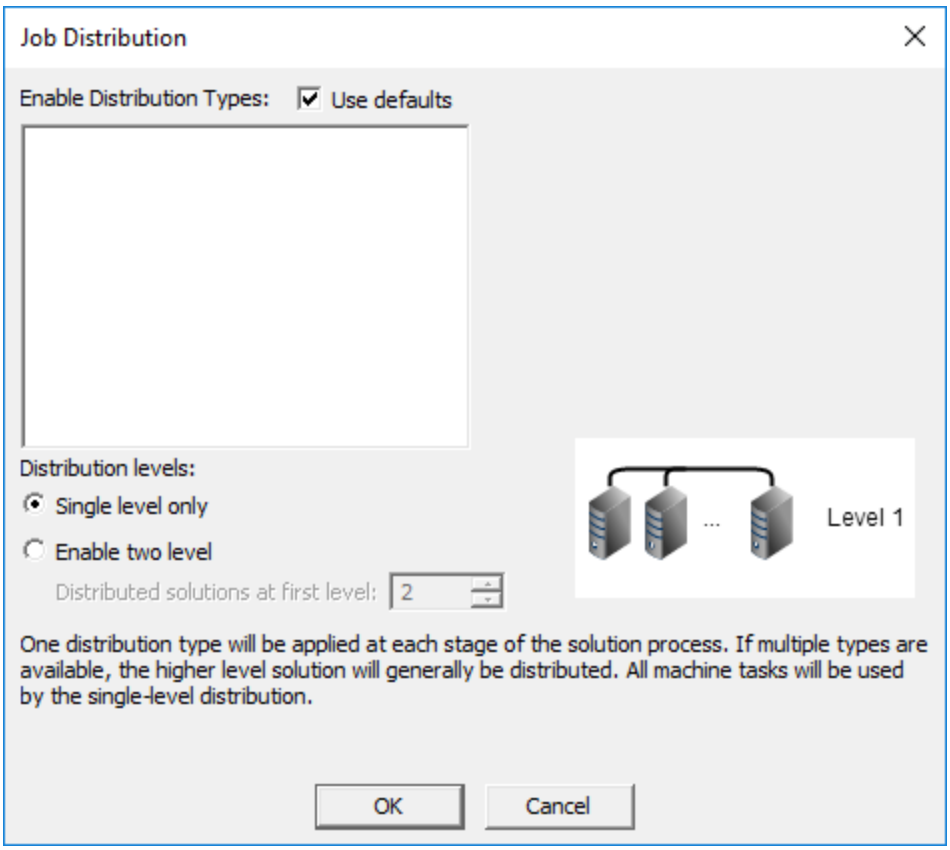


Job Distribution

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you disable User automatic selection, you can modify the Job distribution settings.

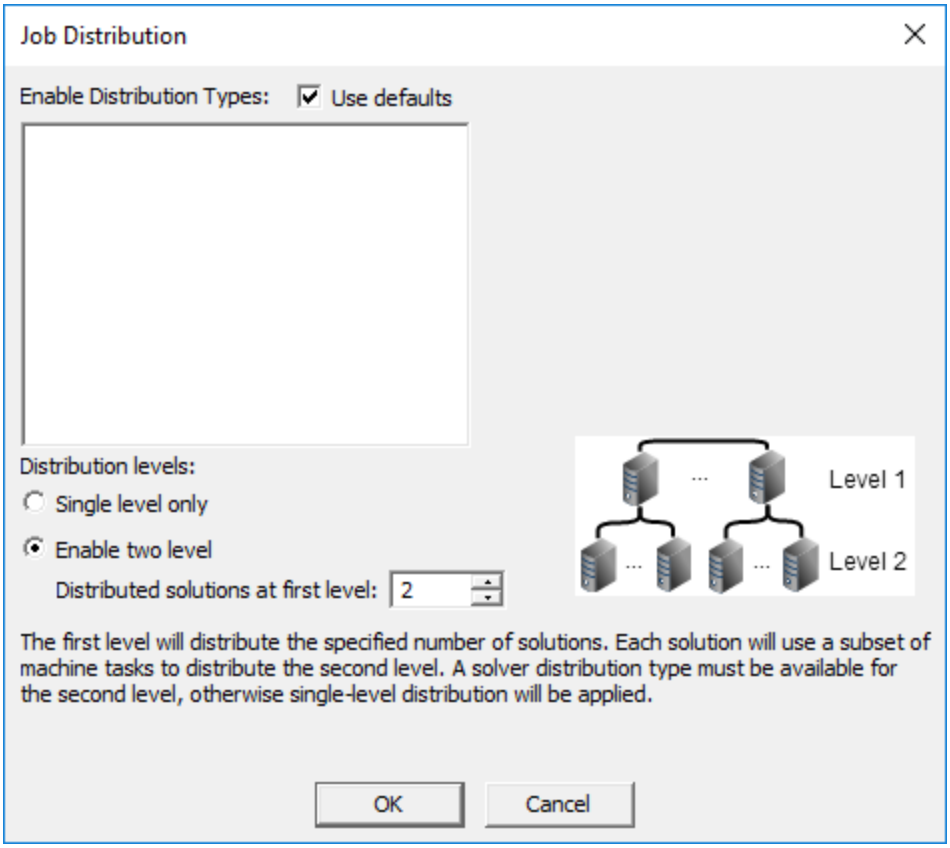
- Single level or two level distribution (*single level* is the default). Click **Modify** to display the *Job Distribution* dialog box and select the **Enable two level** option if appplicable and desired.



Enabled distribution types can be modified here.

Second-level distribution operates within DSO. If available and enabled you can specify a number of engines for level 1.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



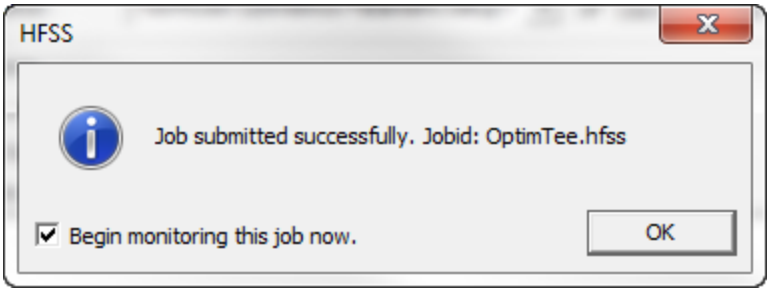
In response to a set of minimal constraints, the Scheduler may increase the resources assigned beyond the minimal values in order to meet the full set of requirements. For example, if you specify 7 distributed engines, with two processors per engine, and also limit the number of engines per node to 4, the scheduler may increase the number of cores used in order to meet the limit specified for engines per node. Notice that a preview of the Submit Job Results shows the number of resources assigned, and that the scheduler generated code includes an MPI specification.

1. To submit the command with the specified parameters, click **Submit Job**.

> **Note:**
>
> The RSM environment does not support for queuing, so **Submit Job** will immediately start running the job.

A dialog box displays. You can select **Begin monitoring this job now** and click **OK**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. You can monitor this job either automatically (by selecting the option above) or through the **Tools** > **Job Management** > **Monitor Jobs...** command. For more details, see Monitor Jobs window.

Process for Changing the Listening Port used by AnsoftRSM Service

To change the listening port used by the AnsoftRSMService, you need to change the configuration file, ansoftrsmservice.cfg, as follows:

You must specify the ListenPort within a 'CommDetails' block, which must be within a 'Default:CommDetails' block, which must be within the top level block of the file, the 'AnsoftCOMDaemon' block. The following example shows the listen port changed from 32958 to 32957, with these blocks at the beginning of the file:

```
$begin 'AnsoftCOMDaemon'
  $begin 'Default:CommDetails'
    $begin 'CommDetails'
      ListenPort='32957'
    $end 'CommDetails'
  $end 'Default:CommDetails'
  . . . .
$end 'AnsoftCOMDaemon'
```

For the second level block, ensure that there is a single colon character and no spaces or tabs separating the two parts of the block name 'Default:CommDetails'. The third level block, with name 'CommDetails' is also required. Use caution when editing this file by hand, because any typos in the block or value names may cause the data to be ignored.

Changing the AnsoftRSMService Listening Port

For Remote Analysis or Distributed Analsysis, processes may need to be started on multiple hosts. If the Ansys Electromagnetics Desktop needs to start a process on a remote host, the AnsoftRSMService is used to start these remote processes. By default, the AnsoftRSMService listens for socket connections from the Ansys Electromagnetics Desktop on port 32958.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This section describes how to change the port number used by the AnsoftRSMService.

To change the port number, both the AnsoftRSMService and the Ansys Electromagnetics Desktop must be configured to use the new port number. The same port number must be used for Ansys Electromagnetics Desktop and for the AnsoftRSMService process running on each host used for the analysis.

AnsoftRSMService Configuration

The AnsoftRSMService port number is configured in the ansoftrsmservice.cfg configuration file. This configuration file is located in the platform specific subdirectory of the RSM installation directory.

The default location of this directory is "C:\Program Files\ANSYS Inc\v252\EM_RSM" on Windows.

The default location of this directory is /opt/ansys_inc/v252/EM_RSM on Linux.

To modify the AnsoftRSMService configuration, first stop the ansoftrsmservice, then modify the ansoftrsmservice.cfg configuration file, then restart the ansoftrsmservice.

> **Note:**
>
> When using Linux, root maintains exclusive control over systemd-managed services. Once the startonboot command has been given, only root users can start and stop AnsoftRSMService, using the commands:
>
> ```
> sudo ansoftrsmservice start
>
> sudo ansoftrsmservice stop
> ```

The beginning of the configuration file should appear as follows:

```
$begin 'AnsoftCOMDaemon'
$begin 'Default:CommDetails'
$begin 'CommDetails'
'ListenPort'='32958'
$end 'CommDetails'
$end 'Default:CommDetails'
```

If there are additional lines between the following two lines, then they should not be modified:

```
$begin 'AnsoftCOMDaemon'
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$begin 'Default:CommDetails'

To change the port number, modify the ListenPort setting within the single quotes from 32958 to the desired port number. The single quotes should not be removed or changed.

For previous versions of the software, the ansoftrsmservice.cfg file may not contain the lines:

$begin 'Default:CommDetails'

$begin 'CommDetails'

'ListenPort'='32958'

$end 'CommDetails'

$end 'Default:CommDetails'

If these lines are not present, then add them to the ansoftrsmservice.cfg file after the first line of the file, and change the ListenPort to the desired port number. For example, to change the port number to 32000, the beginning of the ansoftrsmservice.cfg file should look like the following after the changes:

$begin 'AnsoftCOMDaemon'

$begin 'Default:CommDetails'

$begin 'CommDetails'

'ListenPort'='32000'

$end 'CommDetails'

$end 'Default:CommDetails'

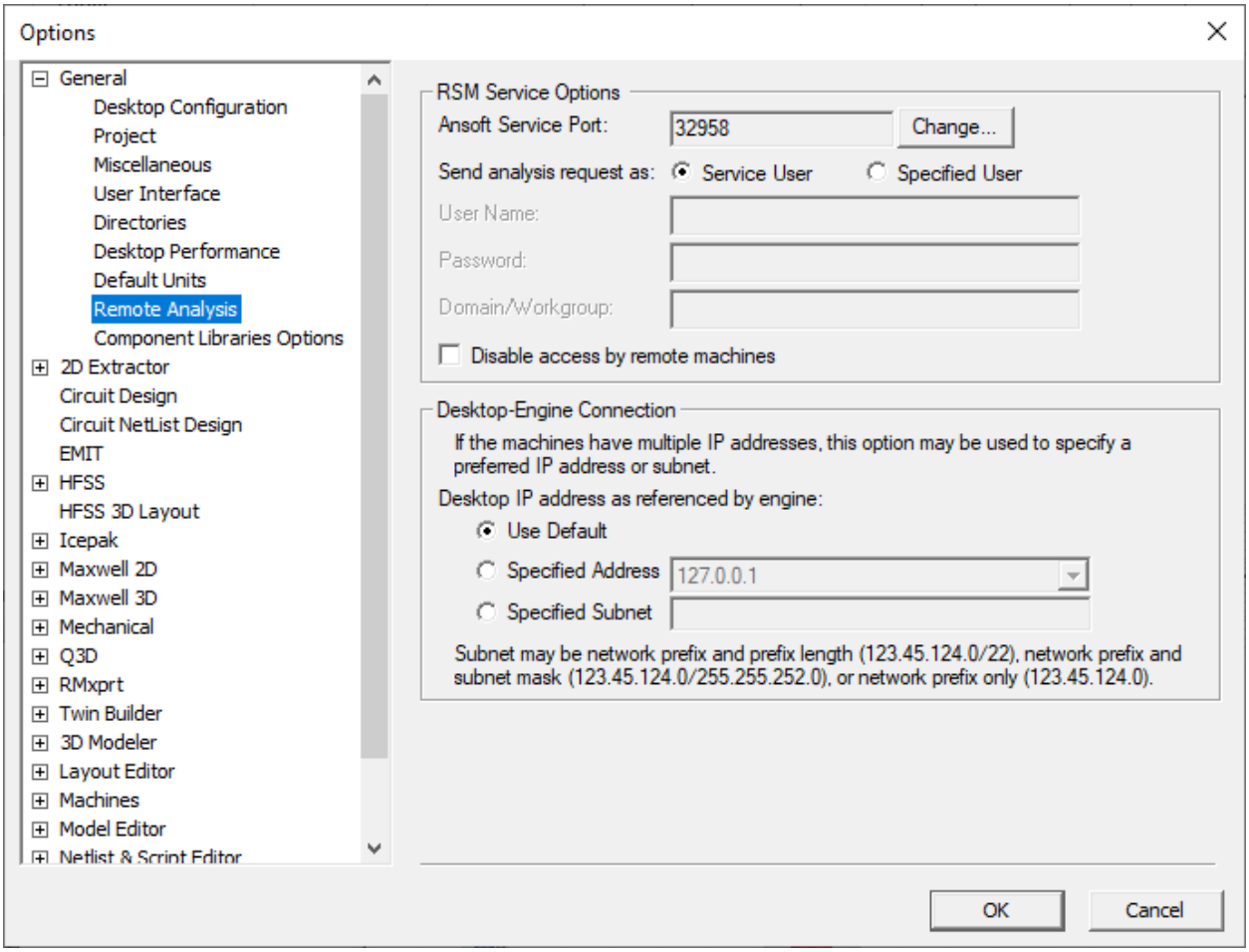
Ansoft Electromagnetics Desktop Configuration

The port number for connecting to the AnsoftRSMService is configured from the **General Options** window.

To access these options:

- Select **Tools** > **Options** > **General Options** to open the **General Options** window.
- If it is not already expanded, expand the **General** tree and click **Remote Analysis**.

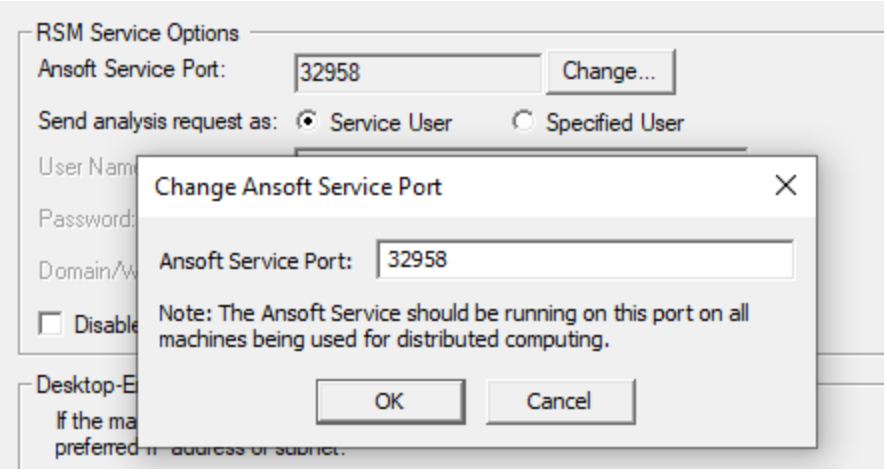
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Under **RSM Service Options**, the **Ansoft Service Port** field shows the current AnsoftRSMService port number.

1. Click **Change** to modify this setting. This opens the **Change Ansoft Service Port** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Enter the new port number in the **Ansoft Service Port** field.
3. Click **OK** to close the dialog box and accept the new port number.
4. Click **OK** to close the **General Options** window and accept the changes.

Running HPC Diagnostics

The Ansys EM HPC diagnostics tool simplifies HPC troubleshooting by automating diagnosis of routine issues. The diagnostics tool is run on the cluster as a scheduler managed job. Using its HTML based diagnostics report, cluster administrator or Ansys support staff can either resolve the issue, or guide the user with steps for further troubleshooting. In some cases, Ansys support staff may request to rerun the diagnostics with additional diagnostics tests. The user may extend the diagnostic scripts to suite their HPC environment.

This note describes how to use the diagnostics tool.

- Supported schedulers
- Running the diagnostics job
- Standard diagnostic job
- Using diagnostics scripts on Linux clusters
- Using Windows HPC job file
- Diagnostic report
- Site-specific diagnostics job
- Environment variables
- ANSYSEM_DIAG_PROD_DIR contents
- ANSYSEM_DIAG_RESULTS_DIR contents
- How does the diagnostic tool work

Supported Schedulers

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The tool supports diagnosis of issues on Linux and windows clusters managed by the following schedulers:

- LSF
- SGE
- PBS/Torque
- Windows HPC

For the above schedulers (see High Performance Computing (HPC) Integration), the tool includes basic diagnostic scripts. Further, if password-ssh has been enabled, it also supports generic Linux clusters using ssh. Please note that currently diagnostics tool does not support PBSPro and LSF/Windows.

Running the Diagnostics Job

The diagnostics are run as a scheduler managed job. Once the job finishes, you locate the resulting HTML file and provide it to the cluster administrator or to Ansys support staff. In case, there are any job or test failures, please also provide the networking*.json files from the Hosts subdirectory as well.

Basic Diagnostic Job

To run the basic diagnostics, submit a diagnostic job to the scheduler using a provided job submission script. Each basic diagnostic job is a 12 core job with 4 cores per host. On Linux, running this script submits a scheduler job to run the diagnostic tool on the cluster. On Windows, you need to submit a job using a job file.

Basic scripts for each supported scheduler are available in diagnostics subdirectory of schedulers directory.

Linux:

```
opt/ansys_inc/v252/AnsysEM/schedulers/diagnostics
```

Windows:

```
"C:\Program Files\ANSYS Inc\v252\AnsysEM\schedulers\diagnostics
```

Using Diagnostics Scripts on Linux Clusters

The following basic scripts are provided in the diagnostics directory (.../ansys_inc/v252/AnsysEM/schedulers/diagnostics):

These job submission scripts are scheduler specific.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scheduler	Basic job submission script	Comment
LSF	test_lsf	Supports both lsrun and blaunch
SGE	test_sge	Supports both qrsh and rsh
PBS/Torque	test_torque	Requires changing the PATH and PBS_BINARY_PATH environment variable
Generic Linux cluster	test_ssh	Supports only ssh. Requires password-less ssh. Requires creating a file with the names of hosts and saving it in ${HOME}/ansysem_hostfile

Using Windows HPC Job File

A sample job file winhpctest.xml is available in the diagnostics directory:

`...\schedulers\diagnostics.`

To submit this diagnostic job, you must change the job description to suite your environment as following:

1. Select a directory for saving the diagnostic results. This directory must be accessible at the same path from all the hosts of the cluster.
2. Locate the directory for Ansys EM installation. This directory also must be accessible at the same path from all the hosts of the cluster.
3. Locate the winhpctest.xml in the diagnostics subdirectory of schedulers directory in Ansys EM installation.
4. Start Windows HPC job manager, and choose "New job from XML File…" action.
5. Select the winhpctest.xml job file.
6. Change the value of both the following environment variables with the directories located in the first two steps:

   ANSYSEM_DIAG_PROD_DIR

   ...

   ANSYSEM_DIAG_RESULTS_DIR
7. Now submit the job.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> After making the above changes, you can also save the resulting XML file using "Submit Job XML File…". Then you can submit the job using the job command as following:
>
> job submit /jobfile: XMLfile name

Diagnostic Report

The diagnostic report is an HTML file that (along with other related diagnostics results) is placed in the following directory

Linux:

```
${HOME}/Ansoft/HPCDiag/Results/JOBID
```

Windows:

```
%ANSYSEM_DIAG_DIR%\Results\JOBID
```

Report file:

```
.../HTML/report.html
```

where JOBID is the job ID assigned by the scheduler. On Windows, the user must specify ANSSEM_DIAG_DIR directory.

Site-Specific Diagnostics Job

To run a diagnostic job with job submission parameters of your choice, you need to create your own job submission script. For example, you may want to specify a different LSF queue, or select a different SGE parallel environment. To run such a job, you need to create your own job submission script starting from the basic diagnostic scripts with the following steps:

1. Locate the relevant basic diagnostic script in the diagnostics subdirectory of schedulers directory in Ansys EM installation.
2. Make a copy of the diagnostics script into a directory that is accessible from a submit host for the cluster.
3. Edit the script file to change the value of ANSYSEM_DIAG_PROD_DIR environment variable to point it to the installation directory (See below).
4. Modify the job submission parameters as needed.
5. Optionally, copy any site-specific diagnostic tests provided by Ansys support staff in the *../Custom* subfolder of the *ANSYSEM_DIAG_RESULTS_DIR* directory.
6. Run the diagnostics script from a submit host for the cluster.

Environment Variables

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following environment variables are applicable for both Linux and Windows environment.

ANSYSEM_DIAG_PROD_DIR

Environment variable	**ANSYSEM_DIAG_PROD_DIR**
Description	Location of the Ansys EM installation. This must be available at the same path from all the hosts of the cluster.
Windows example	"\\filer\AnsyEM\v252\AnsysEM"
Linux example	/shared/ansys_inc/v252/AnsysEM
Comments	Windows: Required. Linux: Optional. Export this environment variable if you make a copy of the diagnostic script.

ANSYSEM_DIAG_RESULTS_DIR

Environment variable	**ANSYSEM_DIAG_RESULTS_DIR**
Description	Location of the diagnostic report and other results on a shared drive. This must be available at the same path from all the hosts of the cluster
Example	\\filer\Home\User\Ansoft\HPCDiag
Linux example	/shared/home/user/Ansoft/HPCDiag
Comments	Windows: Required. Linux: Optional. Export this environment variable if the home directory for the user is not accessible from the cluster.

ANSYSEM_DIAG_CUSTOM_DIR

Environment variable	**ANSYSEM_DIAG_CUSTOM_DIR**
Description	Location of the configuration of product tests and other custom site-specific tests. This location must be on a shared drive that is available at the same path from all the hosts of the cluster
Example	\\filer\Home\User\Ansoft\HPCDiag\Custom
Linux example	/shared/home/user/Ansoft/HPCDiag/Custom

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Environment variable	**ANSYSEM_DIAG_CUSTOM_DIR**
Comments	Windows: Optional. You may want to specify it if the path %ANSYSEM_DIAG_RESULTS_DIR%\..\Custom is not suitable Linux: Optional. Export this environment variable if the home directory for the user is not accessible from the cluster.

How the Diagnostic Tool Works

The diagnostics are run as a scheduler managed job. Running the diagnostic script submits a scheduler job that runs the diagnostic tool on the hosts allocated to the job. Once the diagnostic job starts, the tool executes a set of diagnostic tests. These tests run on each host allocated to the job, and collect diagnostic information relevant for running HPC jobs. The tool combines the diagnostic information to produce an HTML report. The tool saves HTML diagnostic report and other results in a shared drive, which must be available at the same path from all the hosts of the cluster. On Linux, the default is Ansoft/HPCDiag subdirectory under user's home directory. On Windows, the user must specify this location using ANSYSEM_DIAG_RESULTS_DIR environment variable.

Changing Solution Priority for System Resources

You can modify the priority of Ansys Electronics Desktop simulations so that system resources are allocated to other computer processes before the solver. If you reduce the priority of Ansys Electronics Desktop simulations, your other software tools will respond as they normally would, but Ansys Electronics Desktop simulations may take longer.

> **Note:**
>
> The Windows Task Manager does not indicate a reduced priority for the Ansys Electronics Desktop solvers. It only lists the priority of the engine manager, which appears normal, not the actual engine. The actual engine is in a separate thread, whose priority is not visible in the Windows Task Manager.

To change the priority of simulations for the system's resources:

1. While a solution is running, right-click the **Progress** window, and click **Change Priority** on the shortcut menu.
    - To affect priority for future simulation runs, click **Tools** > **Options** > **HPC and Analysis** to access the **HPC and Analysis** dialog box, and click the **Options** tab.
2. From the **Change Priority** menu (or the **Default Process Priority** drop-down menu), select one of the following priorities:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lowest Priority	
Below Normal	
Normal	The default.
Above Normal	
Highest	

3. Click **OK**.

desktopproxy Utility to Abort or Cleanstop a BatchSolve

The desktopproxy Utility to abort or cleanstop a batch solve is available in the installation directory: ("...\ANSYS Inc\v252/AnsysEM" on Windows, ".../ansys_inc/v252/AnsysEM" on Linux).

To abort a job, run it on a host which has network access to the first execution host of the job, as well as has access to the directory of the project file. This utility does not abort Large scale DSO jobs for which you need to use desktopjob utility itself.

Usage:

```
desktopproxy -abort <projectpath>
-cleanstop <projectpath>
```

Where the <projectpath> is the full path to the project file.

The behavior (e.g., saving the results computed so far) should be same as if you interactively aborted AnsysEDT solve. if AnsysEDT is 'busy' (e.g., doing Optimetrics calculation, computing a report), abort (using desktopproxy) might take a long time.

The utility exits silently after it requests ansysedt process to abort the job cleanly. After the job ends, the batch log file may contain a message such as:

```
[error] Project:TestProject, Design:HFSSDesign1 (DrivenModal),
Simulation was aborted by user on server: sjohpc-lsf-0.ansys.com.
(3:52:30 PM Oct 11, 2013)
```

The lock file will be automatically deleted.

Example (Linux):

```
desktopproxy -abort /home/user/test/Diff_Via.aedt
```

Example (Windows):

```
desktopproxy -abort \\server\share\projectfolder\Diff_Via.aedt
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Limitations:

- Aborting a job makes a best effort to delete all the temp directories on all the hosts
- For a hung job, a clean shutdown may not be possible. In that case, the user must use OS/scheduler's facilities to kill the job.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19 - Ansys Workbench Integration Overview

Ansys Workbench combines the strength of its core product solvers with the project management tools necessary to manage project workflow. In Ansys Workbench, analyses are built as *systems*, which can then be combined into a *project*. The project is driven by a schematic workflow that manages the connections between the systems.

From the schematic, you can interact with applications (called workspaces) that are native to Ansys Workbench and that display within the Ansys Workbench interface. Native workspaces include: Project Schematic, Engineering Data, and Design Exploration (Parameters and Design Points).

You can also launch applications that are data-integrated with Ansys Workbench, meaning the application's interface remains separate, but the data from the application communicates with the native Ansys Workbench data. Thus, data can be passed back and forth between any Ansys Electromagnetics product on a Workbench Project Schematic and any supported Ansys or Ansys Electromagnetics desktop product. Depending on the application, data integration can include basic actions such as saving projects, as well as more complex actions such as the coupling of Ansys Electromagnetics product variables to Workbench Design Exploration parameters.

Data-integrated applications include the following Ansys Electromagnetics products:

Circuit, HFSS, Icepak, Maxwell/RMxprt, Mechanical, Q3D Extractor, and Twin Builder.

> **Note:**
>
> For detailed information on working with Ansys Workbench, please refer to the Workbench documentation.

Integrating Ansys EM Suite with Ansys Workbench

You can integrate the Ansys Electromagnetics Suite with the Ansys Workbench application after both have been installed on your system. For successful integration, both preinstalled applications must be of the same version. A single operation completes the integration process for all supported design types.

The same procedure is used to integrate the applications or to decouple them (if they had previously been integrated). The procedure is as follows:

1. From the Windows Start Menu, select **All Programs** > **Ansys EM Suite 20*yy* R*n*** > **Modify Integration with Ansys 20*yy* R*n***.

   A *Modify Integration with Ansys 20yy Rn* command window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

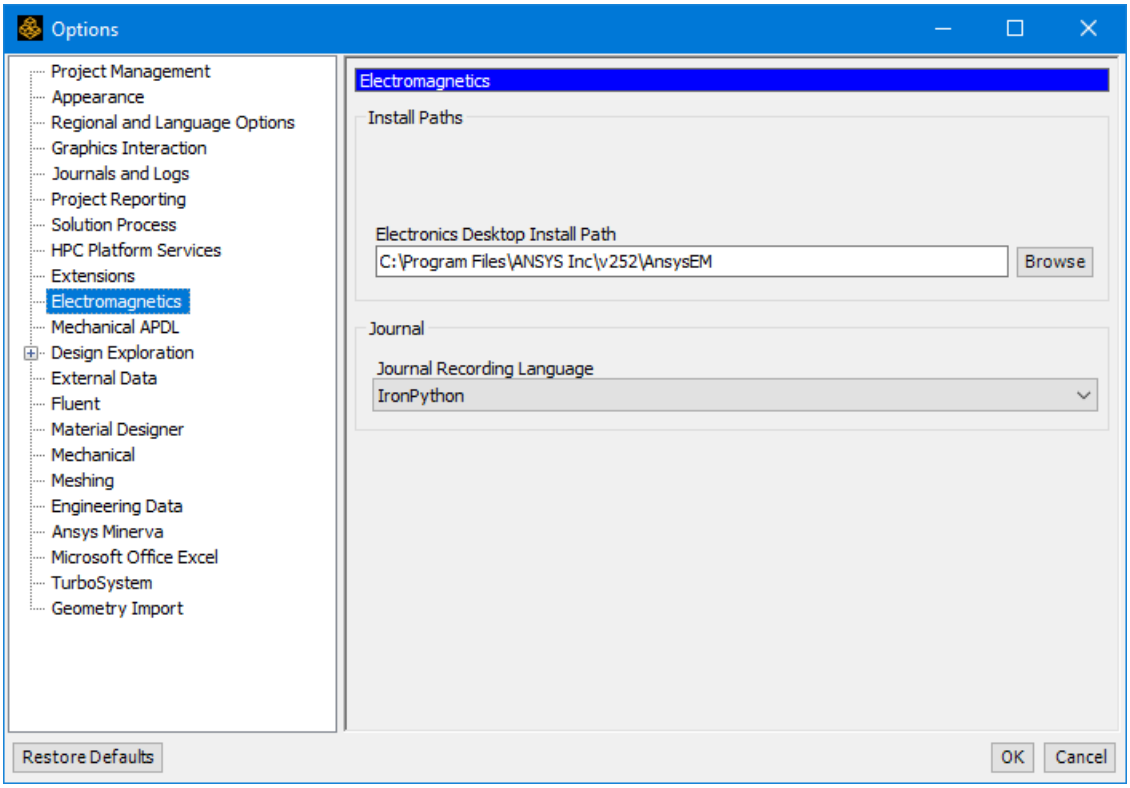
- If both applications are present on the system, their installation paths will be reported, and you will be asked if you wish to integrate them.
- Alternatively, if integration has already been performed, you will be asked if you wish to decouple them.

2. Type **yes** to perform the prompted action or **no** to abort the operation without making any changes. Then press **Enter**
   - If you enter *"yes,"* a confirmation line appears in the command window indicating that integration (or decoupling) has been completed, and the window remains open.

     Press **Enter** again to close the command window.
   - If you enter *"no,"* the operation aborts and the command window closes immediately.

Note:

You can confirm that the Ansys 20*yy* R*n* application is “aware” of the integrated application via the *Options* dialog box in Workbench. The path to the integrated Electronics Desktop application is listed in the *Electromagnetics* group of settings.



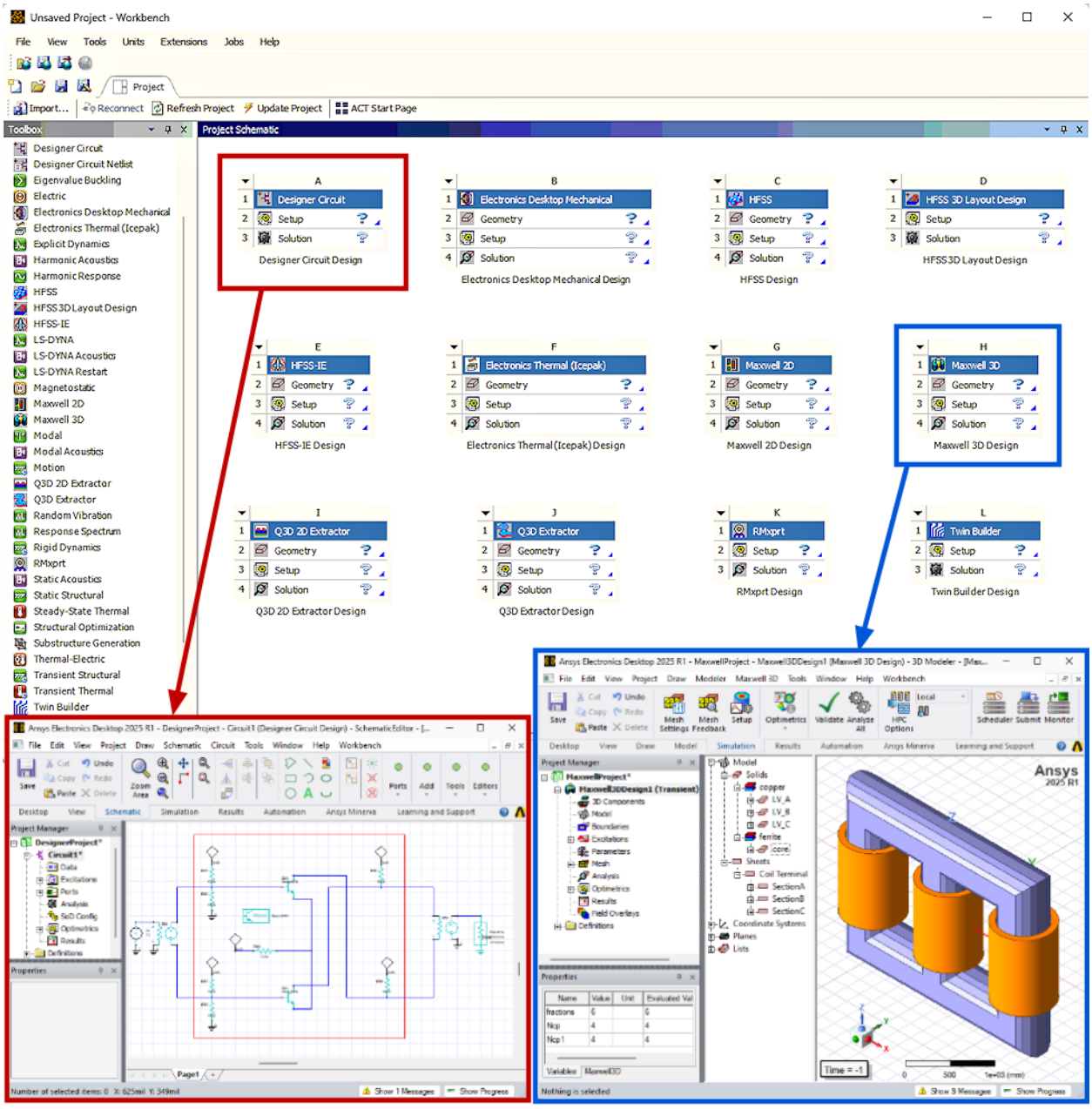
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this group of options, you can also specify Workbench's default **Journal Recording Language** for use when interoperating with the Ansys Electromagnetics products. **Iron Python** is the default language, and the alternative choice is **Visual Basic** (VB). Changes applied here affect any new Workbench or Ansys Electronics Desktop session launched after the change. Any open instance will continue recording in the language that was in effect when the program was launched.

Workbench Data Integration Overview

Ansys EM data-integrated applications can reside on a Workbench **Project Schematic**, as shown.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



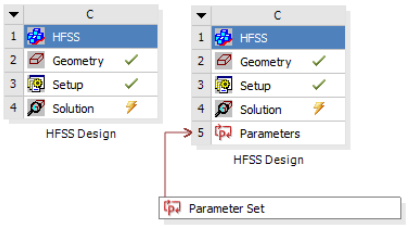
Objects placed on a Workbench Project Schematic (such as instances of Ansys Electromagnetics projects) are referred to as *systems*. The number of cells in the system depend on the desigh type, as follows:

- **RMxprt, Circuit, HFSS 3D Layout,** and **Twin Builder:** Appear on the Workbench Project Schematic as systems with three cells: **<*System_Type_Name*>, Setup,** and **Solution**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **HFSS, Maxwell, Q3D Extractor, Icepak,** and **Mechanical:** Include an additional **Geometry** cell along with the **<*System_Type_Name*>, Setup,** and **Solution** cells, for a total of four cells.

If you invoke Ansys **DesignXplorer** to use variables for refining a design, a **Parameters** cell is added, with a link to the associated Workbench **Parameter Set**. Refer to the Ansys Workbench help for details on working with systems, cells, and parameter sets.



All supported Ansys Electromagnetics Suite products integrate with Workbench commands, services, and DesignXplorer in a similar manner. Some capabilities include:

- Adding New Analysis Systems
- Importing Existing Ansys EM Projects into Workbench
- Editing Ansys EM Models in Workbench
- Ansys EM to Workbench Geometry Transfer
- Ansys EM to Workbench Material Data Transfer
- Analyzing Ansys EM Models in Workbench
- Performing Parameter Studies in Workbench
- Scripting in Workbench

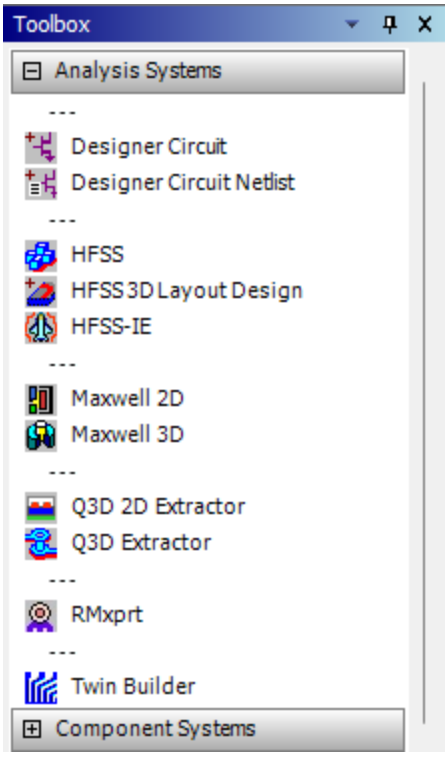
In addition to these major features, Workbench can archive, save, back up, duplicate, and delete Ansys Electromagnetics projects used in a Workbench project. Progress information and messages from integrated Ansys EM projects are also displayed in Workbench.

> **Note:**
>
> Look in the *Ansys Workbench Help* to find detailed information about performing the above-listed operations.

Adding New Analysis Systems

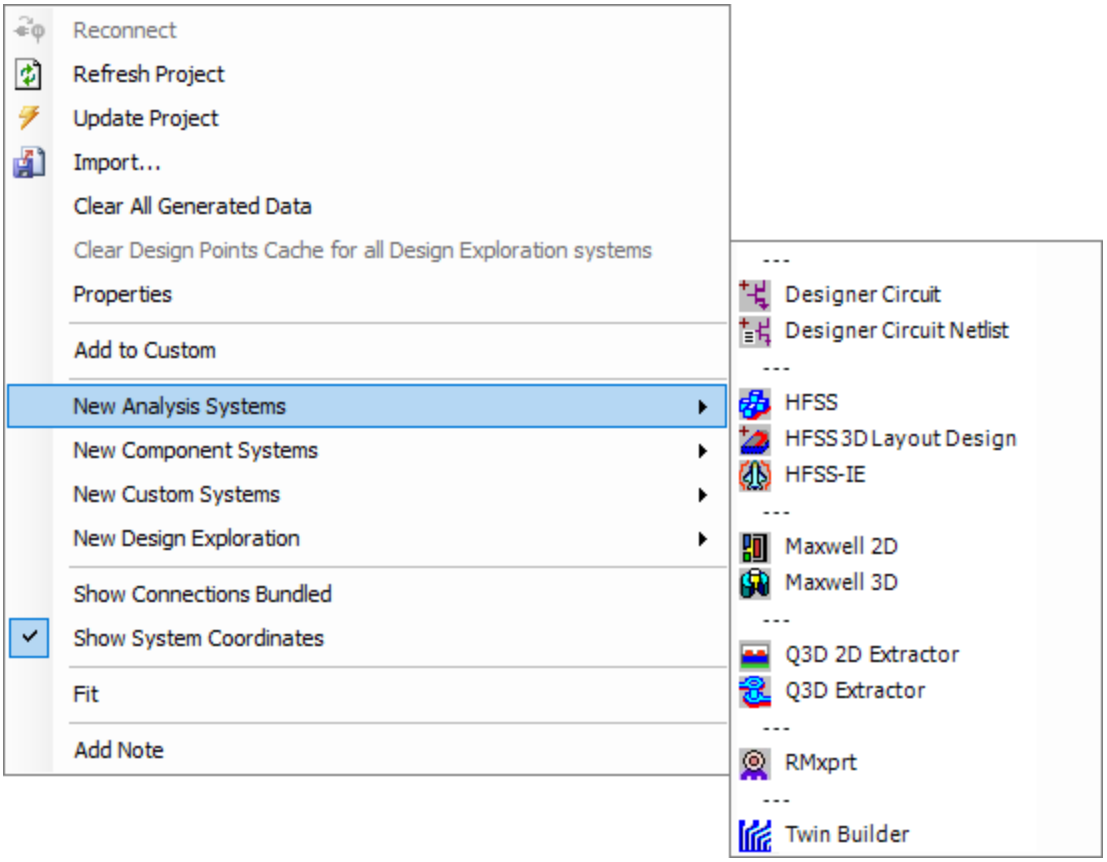
A new Ansys Electromagnetics Analysis System can be added to a Workbench Project Schematic by dragging and dropping it from the Toolbox:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Alternately, you can add a system by right-clicking in the Workbench Project Schematic area and selecting **New Analysis Systems** > **[System Type]**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

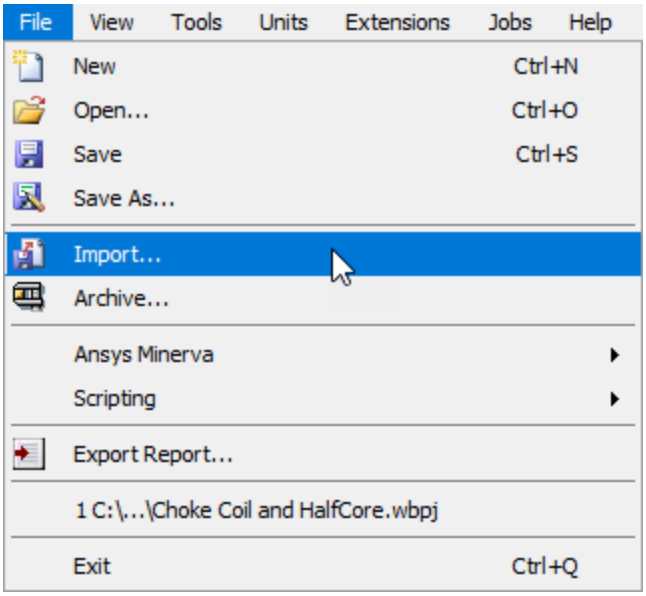


Importing Existing Ansys EM Projects into Workbench

You can import existing Ansys Electromagnetics Suite projects into a Workbench project schematic using **File** > **Import**. When you do, a copy of the Ansys Electromagnetics project is placed in the Workbench project folder. The original Ansys Electromagnetics project remains intact.

> **Note:**
>
> Object, material, and parameter names with non-ASCII characters are not allowed for data transfer. Such transfers fail and produce an error message.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

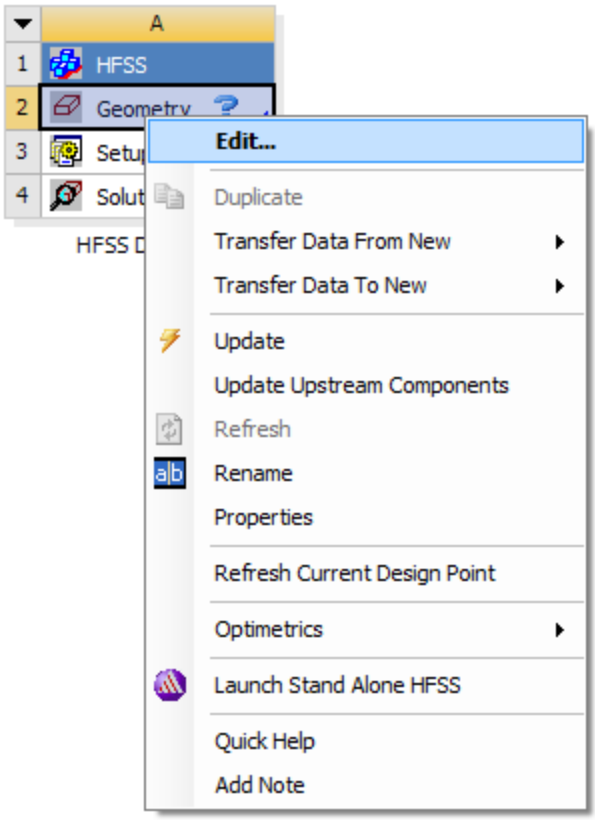


Editing Ansys EM Models in Workbench

You can edit various properties and parameters of the Ansys Electromagnetics project (geometry, setup, solution, etc.) either by right-clicking the project in Workbench and selecting **Edit**, or by double-clicking the project.

Doing so launches the Ansys Electromagnetics desktop application and loads the project so that you can set up your project in a familiar desktop environment. Changes made to the Ansys Electromagnetics project are saved to the project instance in the Workbench project folder.

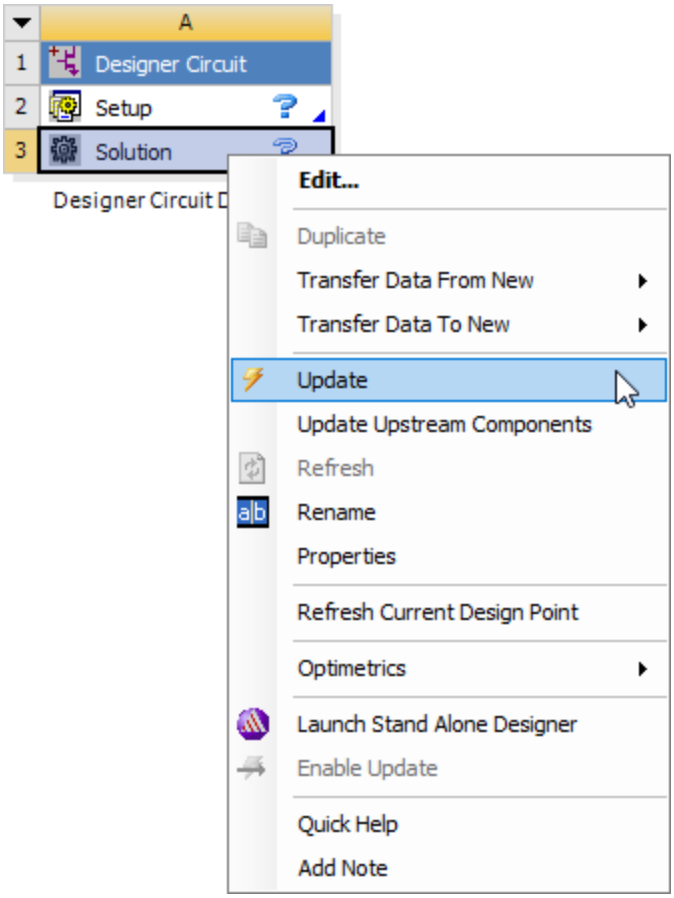
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



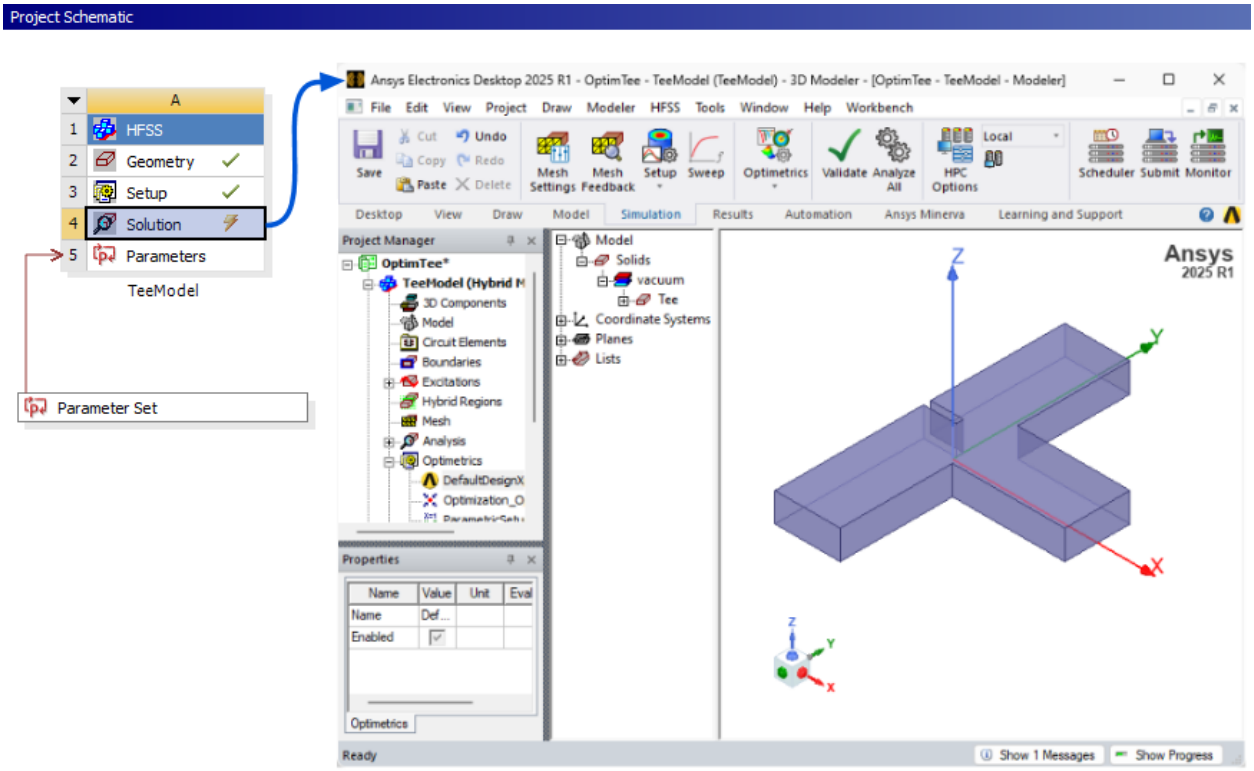
Analyzing Ansys EM Models in Workbench

You can use Workbench's **Update** command to run analyses in the integrated Ansys Electromagnetics project. Progress information is also shown in Workbench.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



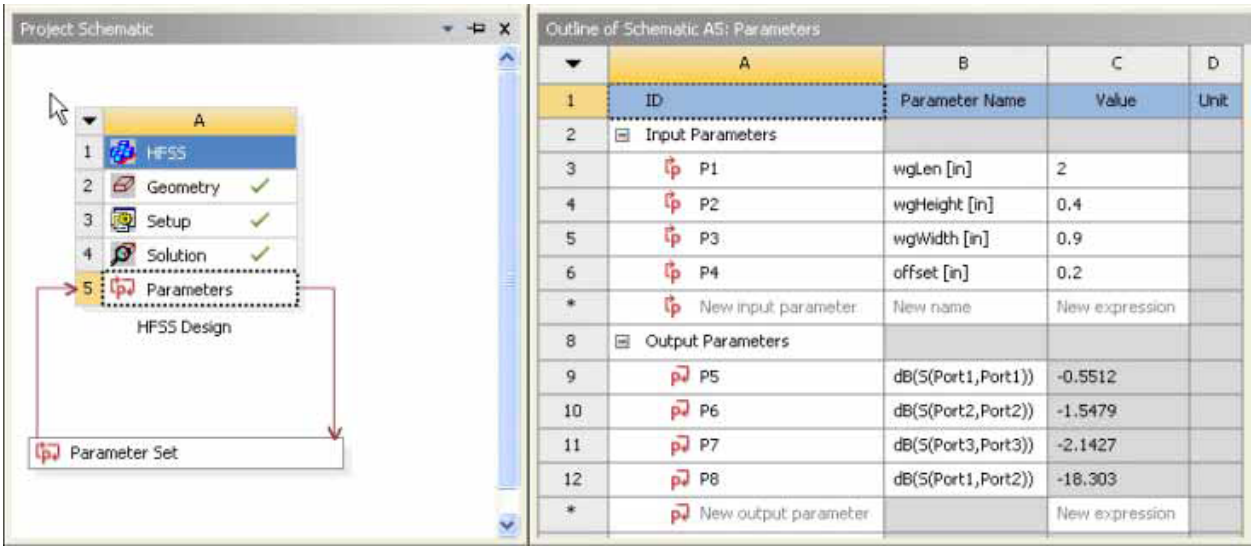
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Performing Parameter Studies in Workbench

Workbench **Parameter Sets** allow you to change parameter values and units of measure, or add new parameters. Parameter data is passed back to the Ansys Electromagnetics application for updated analyses. After analysis, the Workbench Parameter table should be updated correctly for all design points.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

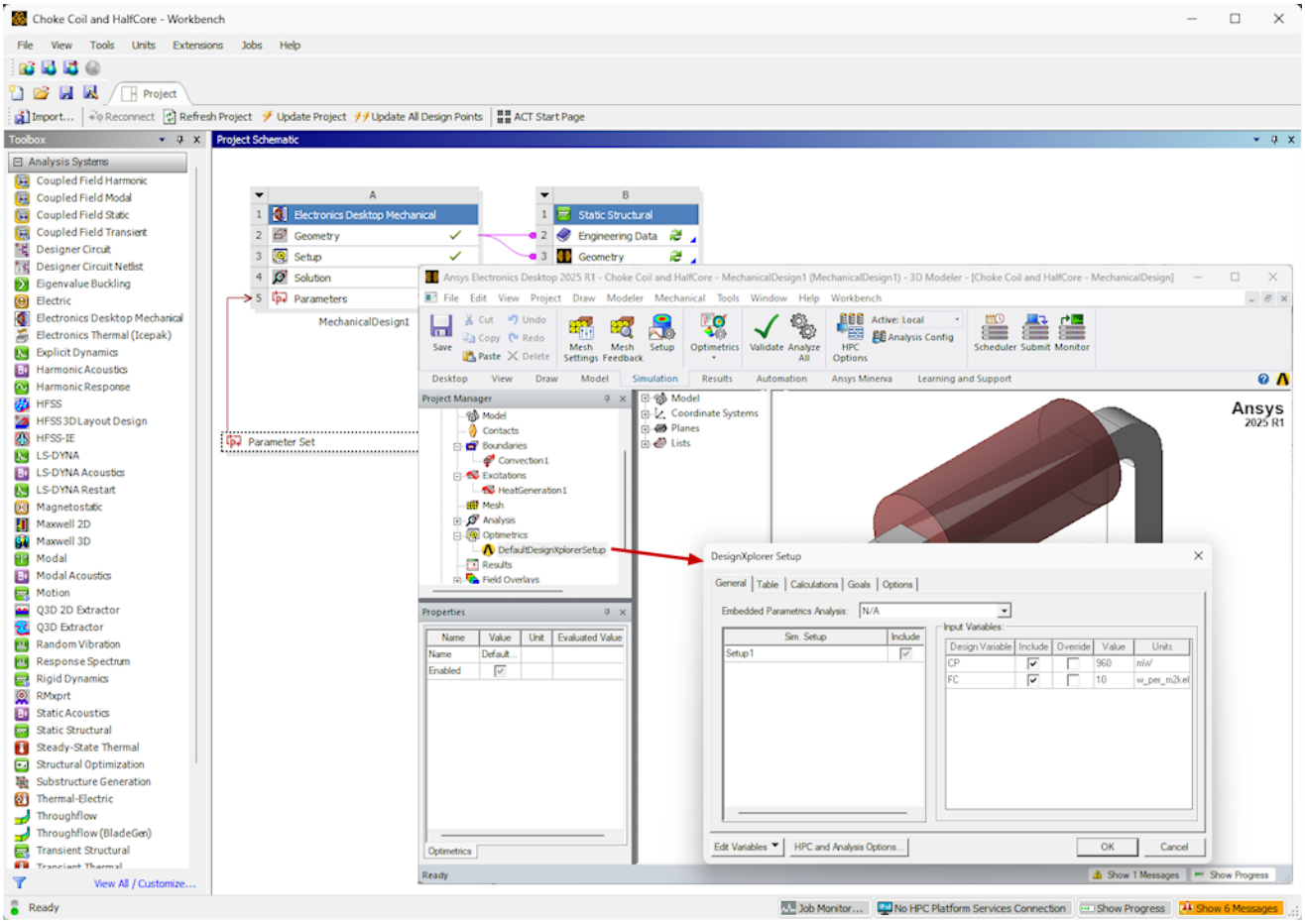


Note:

Because Workbench and Ansys Electromagnetics Desktop have different unit systems, units defined in Desktop are Dimensionless in the Workbench Parameters table and thus cannot be changed. Refer to the *Ansys Design Modeler User's Guide* > *Project Schematic Operations* > *Parameters in Project Schematic* > *Parameter Units* topic for additional information.

Parameters from the Ansys Electromagnetics project are exposed to Workbench through the **DesignXplorer** setup. The Ansys Electromagnetics system's cell status on the Workbench project is updated as changes are made in the Ansys Electromagnetics application. The following example is a Mechanical, Steady-State Thermal design in which two variables have been defined. One variable (CP) is the coil power (960 mW heat generation) and the other (FC) is the film coefficient of the assigned convection boundary:

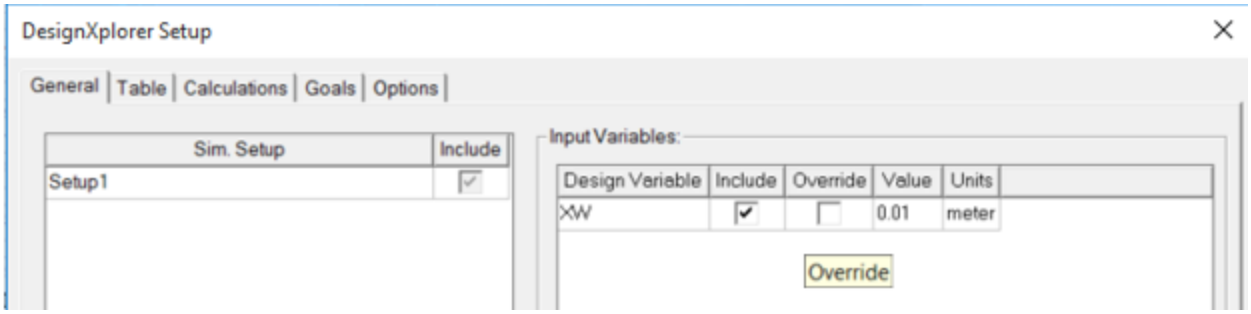
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



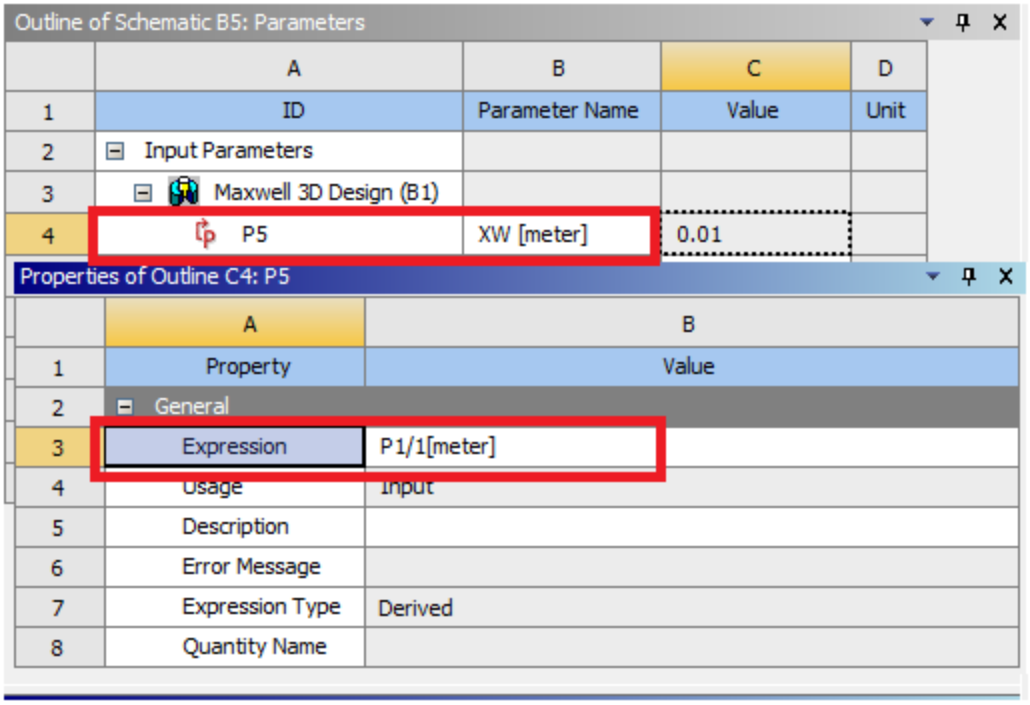
The workflow for using Electronics Desktop systems with the Workbench to take advantage of Distributed Analysis is as follows:

1. In the Ansys Electronics Desktop, specify a variable an as input parameter. The following figure shows a DesignXplorer Setup that includes a geometry variable called XW This variable is mapped to a variable in the corresponding Workbench Design, in this example, P5.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

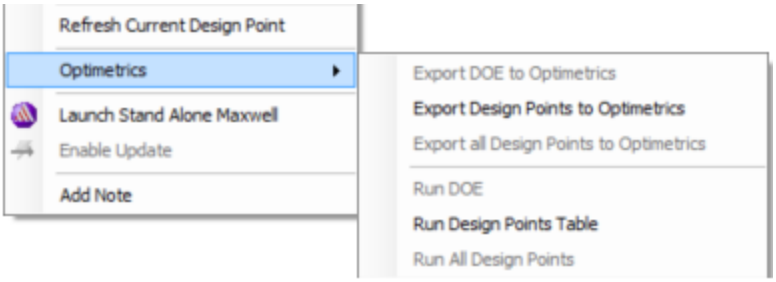


2. In the Workbench Design, change the P5 expression to a new value (in this example figure, the P5 parameter is assigned an expression of P1/1[meter] to tie it to the geometry system's variable.



In Workbench, the **Export Design Points to Optimetrics** and **Run Design Points Table** commands are enabled.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

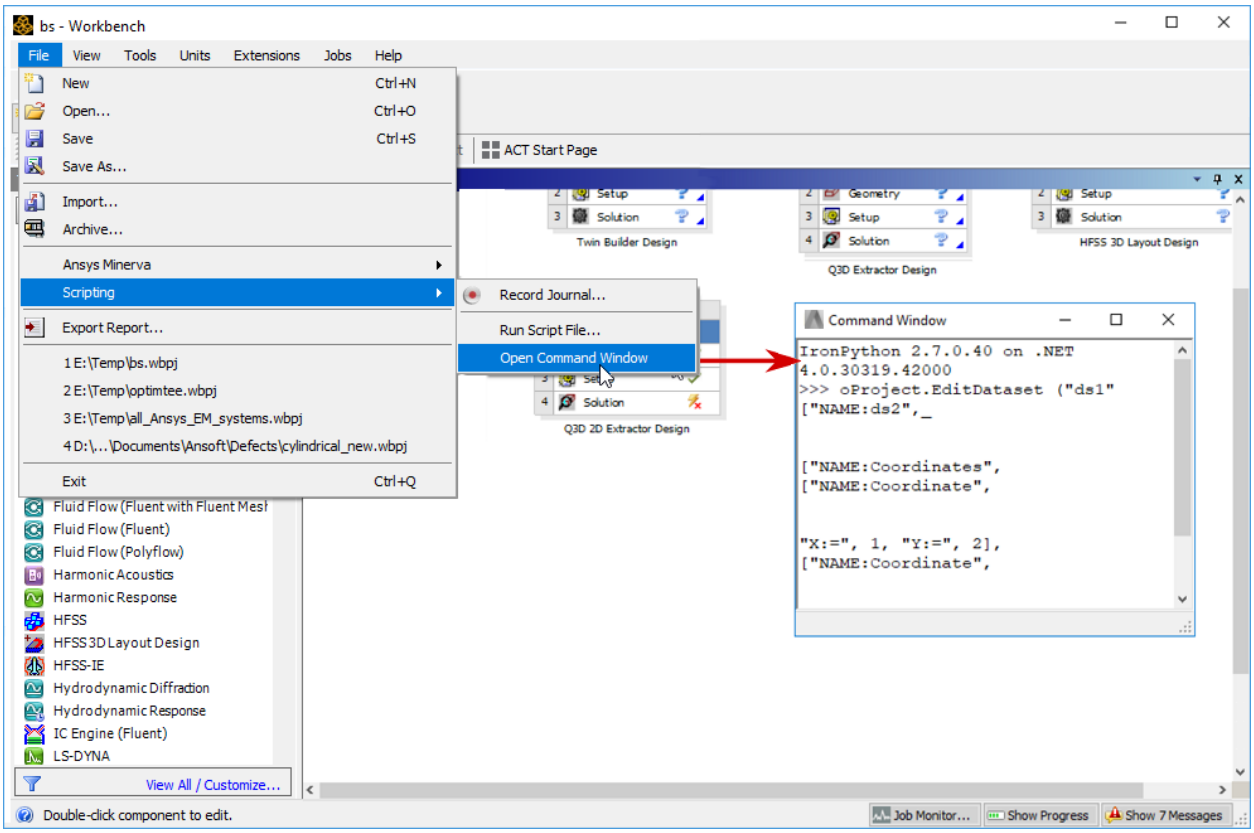


3. In Electronics Desktop, you can click on the default DesignXplorer Setup, and choose **Generate Variation Data**.

If you have configured your Analysis Configuration for multiple machines, the Project is ready for Distributed Analysis. After analysis, the Workbench Parameter table should be updated correctly for all design points.

Scripting in Workbench

Scripts that include Ansys Electromagnetics projects can be recorded and run in Workbench.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When a Workbench journal is run from the command line, Electronics Desktop allows the Ansys EDT portion to also run in batch mode (that is, when using RunWB2 -b).

Example: From a Windows Command prompt, enter the following command to replay the journal named "myReplayJournal.wbjn":

```
RunWB2 -b -r myReplayJournal.wbjn
```

Ansys EM - Ansys Multiphysics Coupling

Data integration provides an improved multiphysics workflow between Ansys Electromagnetics designs and Ansys applications such as Mechanical and Thermal. Coupling is provided through project schematic links. Heat losses and force data are automatically transferred to Ansys Mechanical, and there is no need to export or import transfer XML files. Edits made in Ansys Electromagnetics applications are automatically transferred to the Ansys application through the Workbench **Refresh** command. Workbench commands also enable easier automation of iterative coupling of thermal feedback.

The following sections provide some examples of multiphysics coupling.

Multiphysics Coupling on Workbench with Ansys Thermal

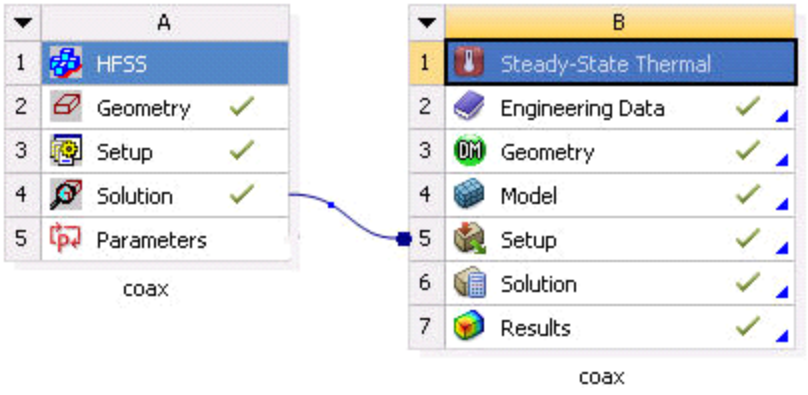
Multiphysics Coupling on Workbench with Ansys Structural

Multiphysics Coupling between Ansys EM Field Systems on Workbench

Multiphysics Coupling on Workbench with Ansys Thermal

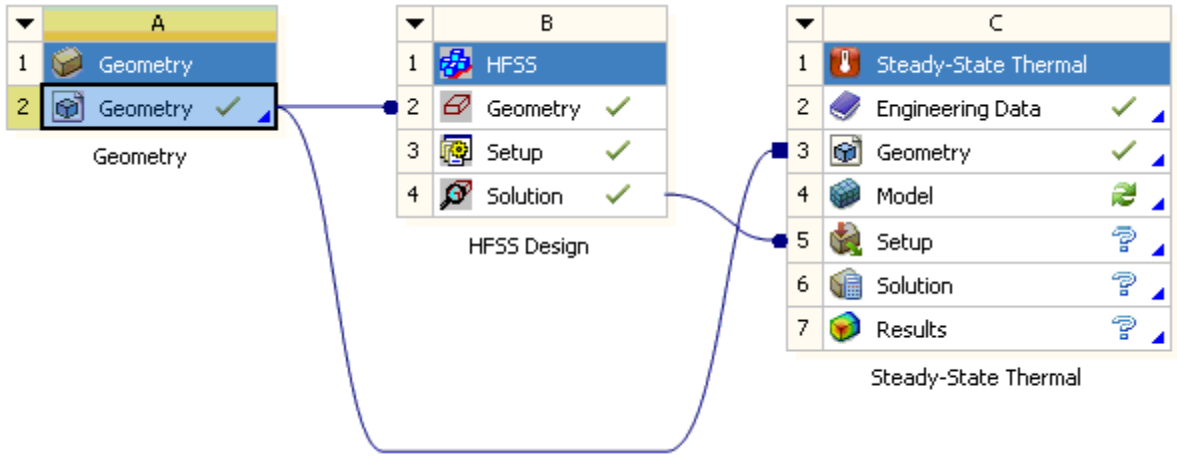
Using data integration, HFSS, Maxwell, and Q3D Extractor provide heat losses (heat generation and heat flux) to Ansys Thermal. In HFSS, both Driven and Eigenmode projects can be coupled for multiphysics. You first need to enable feedback as described in Setting the Temperature of Objects.

Note how the HFSS design is linked visually to Ansys Thermal on the Workbench project schematic.

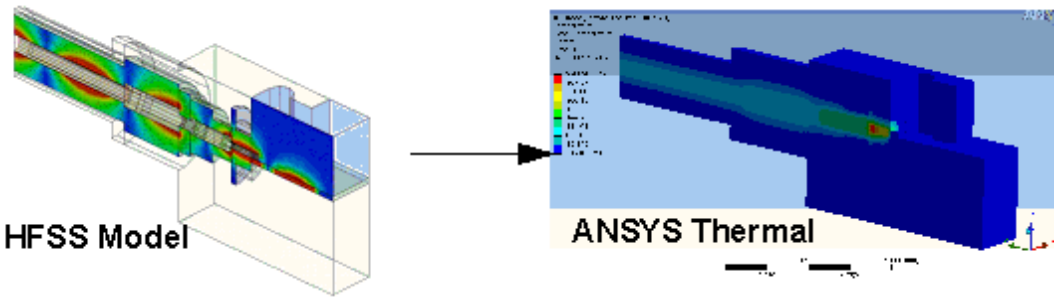


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Geometry sharing is possible, provided you activate Beta Options in Workbench. A geometry from the Component Systems in the Toolbox in Workbench Schematic can be shared as shown in the image below.



In this example, HFSS coax model Solution provides heat loss data as a thermal load to the Ansys Thermal Setup. The resulting analysis shows a thermal "hotspot", providing the user with the information needed to adjust the design's material to fix the problem.

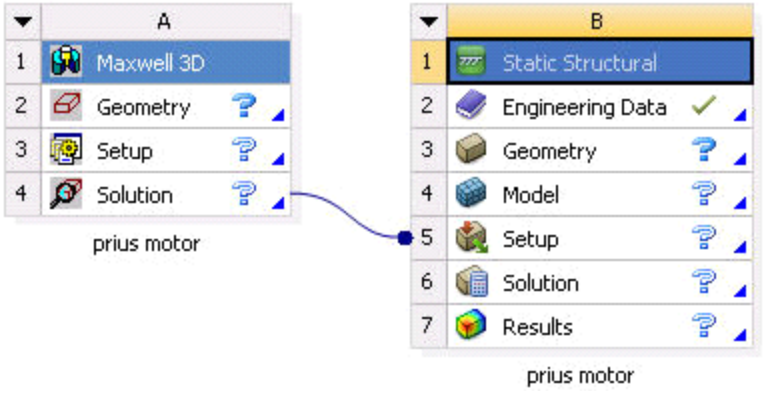


You enable this feature by checking Enable Feedback in the dialog for Setting the Temperature of Objects.

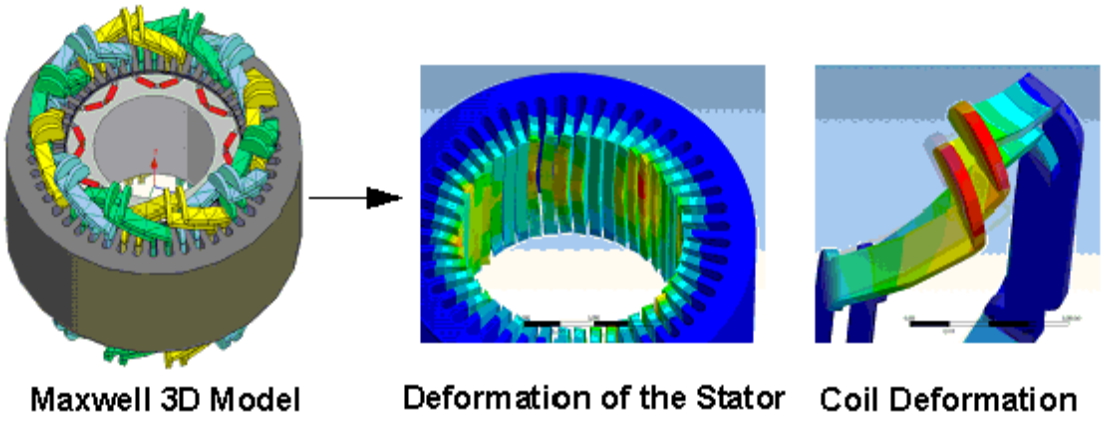
Multiphysics Coupling on Workbench with Ansys Structural

Using data integration, Maxwell 2D and Maxwell 3D can provide forces to Ansys Structural.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In this example, the Maxwell 3D electromagnetic force density **Solution** is used as the load in Ansys Structural to determine how these forces deform the motor's stator and coils.

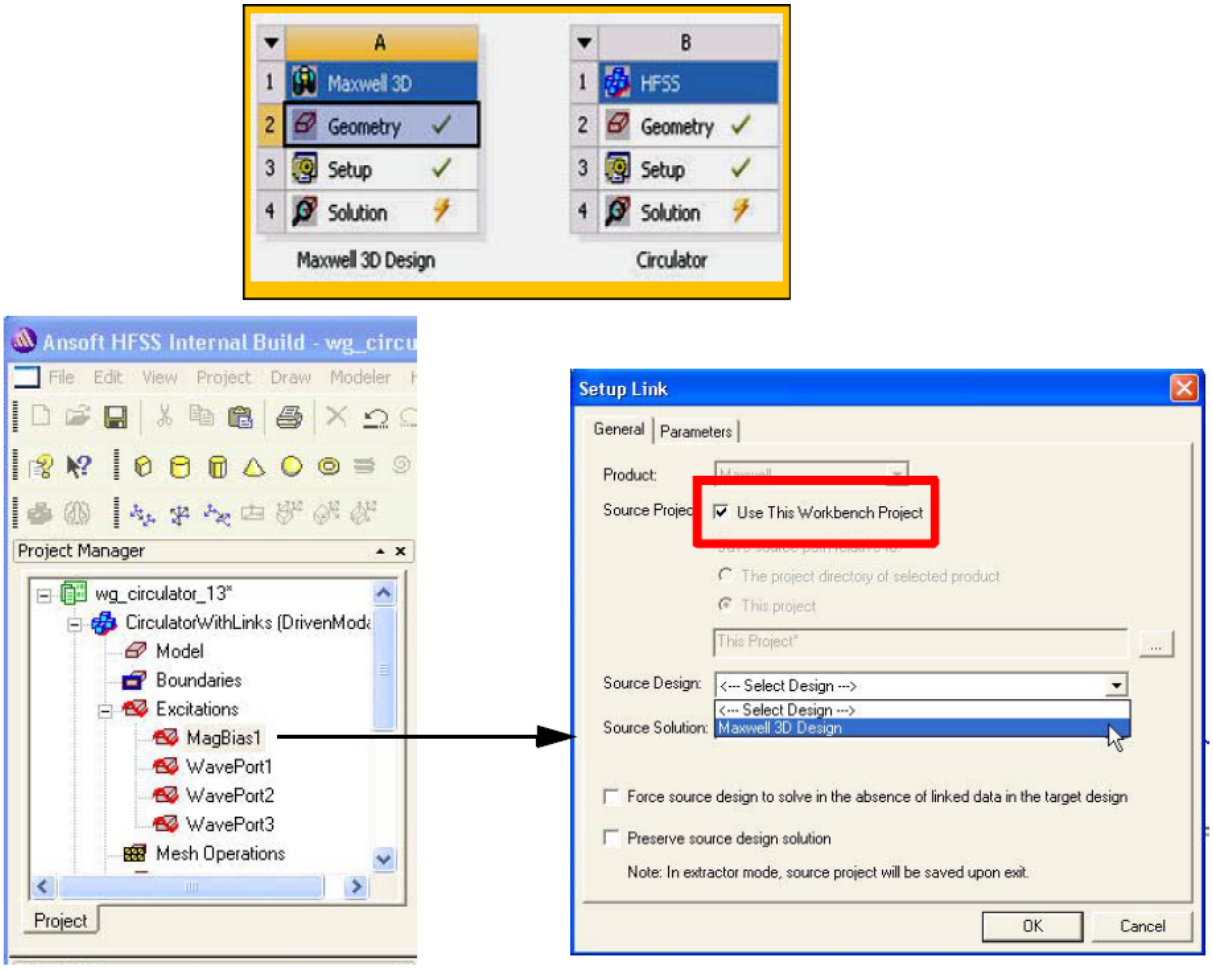


Multiphysics Coupling between Ansys EM Field Systems on Workbench

You can set up links between Ansys Electromagnetics field systems that reside on a Workbench project schematic. Linking is set up in the Ansys Electromagnetics application as shown in the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

following example.

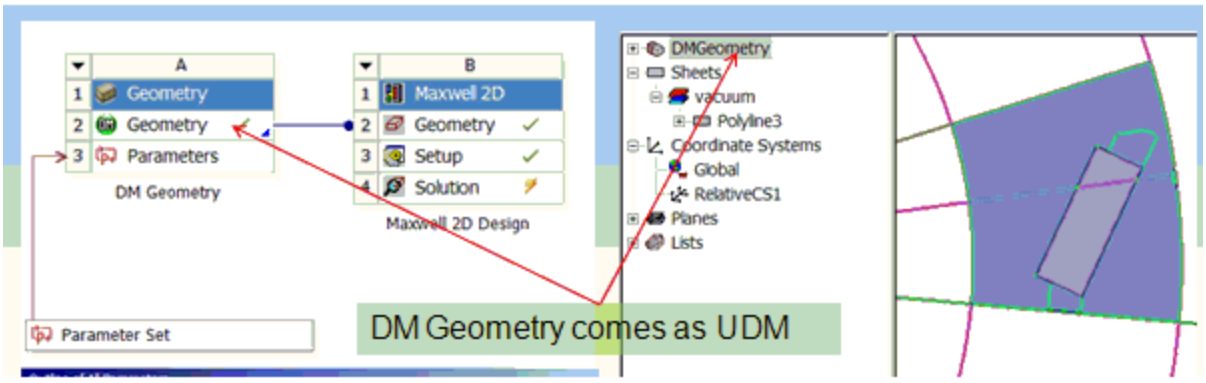


Ansys EM CAD Integration through Workbench

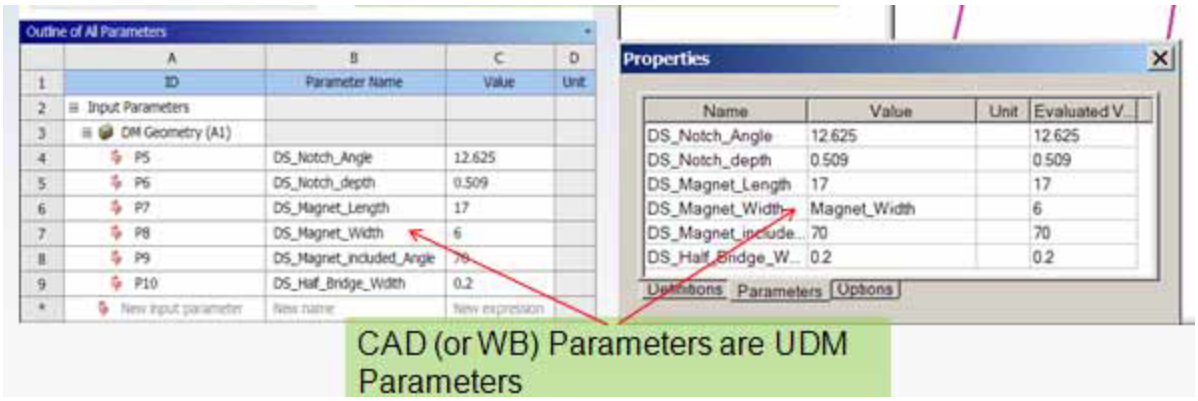
Ansys Electromagnetics CAD integration is a Workbench feature available for Ansys Electromagnetics 3D Products - HFSS, Maxwell, Q3D, Icepak, and Mechanical, as the Ansys Framework. The feature is available only through Workbench, and is not available via standalone Ansys Electromagnetics products.

Ansys CAD integration provides a bi-directional dynamic link through Workbench. This makes it possible to get updated geometry from CAD, modify CAD parameters in Ansys Electromagnetics products, and return an updated geometry. The feature is non-associative due to a need to reassign boundaries if a modified CAD model is used. The process creates a User Defined Model (UDM) for each geometry source.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



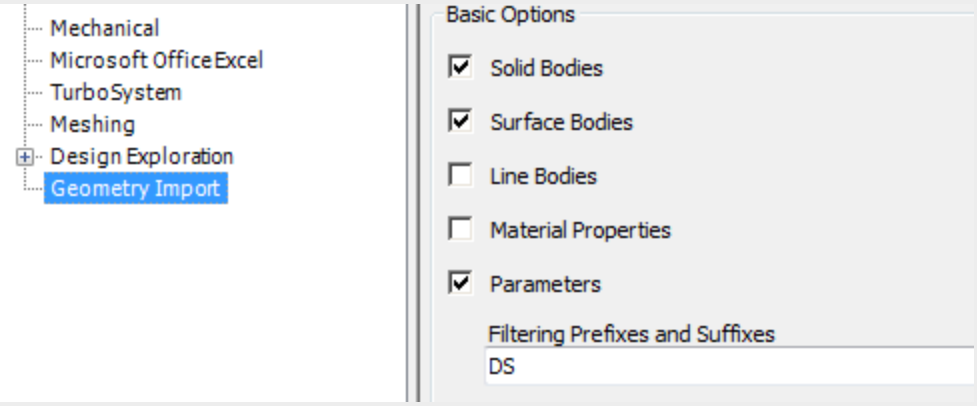
The UDM format makes it possible to exchange parameters. Ansys online help contains further description of the UDM feature and function.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The parameters shown in the previous example all have a DS prefix. This is the default for the Workbench **Tools>Options** for **Geometry Import**. To import parameters with different prefixes or names, assign an appropriate prefix or clear the **Filtering Prefixes and Suffixes** field.



Ansys Electromagnetics CAD integration makes it possible to consume geometry from multiple upstream sources (any CAD or Ansys Electromagnetics product).

This feature supports direct interfaces with major CAD systems, including

- Creo Parametric
- UG NX
- CATIA V5
- SOLIDWORKS
- Autodesk Inventor
- Ansys Design Modeler (DM)
- Discovery UDM model

Note:

The CAD software must be installed on the user machine (not required for solve nodes).

The following sections contain additional information:

CAD Integration Functionality

CAD Integration and Geometry Sharing

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Bi-Directional CAD Integration

CAD Integration Model Edits

Multiple Geometry Links for CAD Integration

Creating Dynamic Links to EDT Designs

CAD Integration Functionality

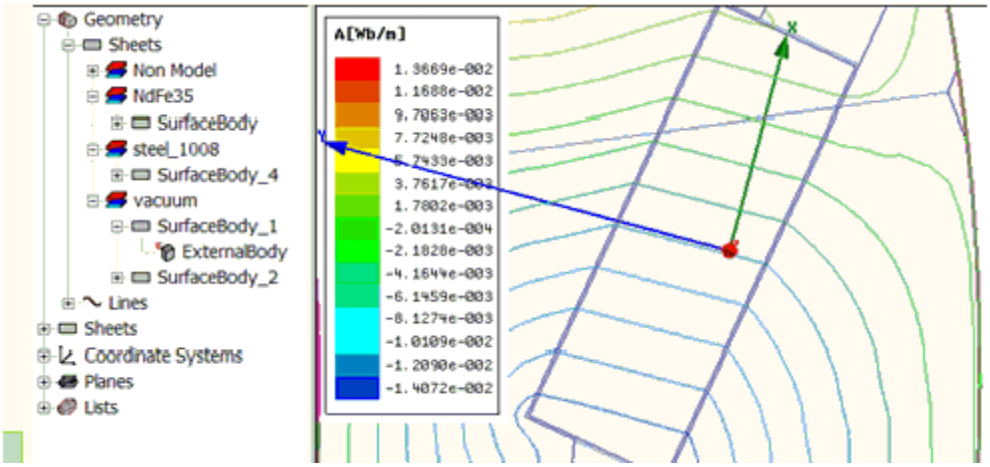
Healing with CAD Integration

Important Geometry Options for CAD Integration

CAD Integration Functionality

CAD Integration includes the following functionality:

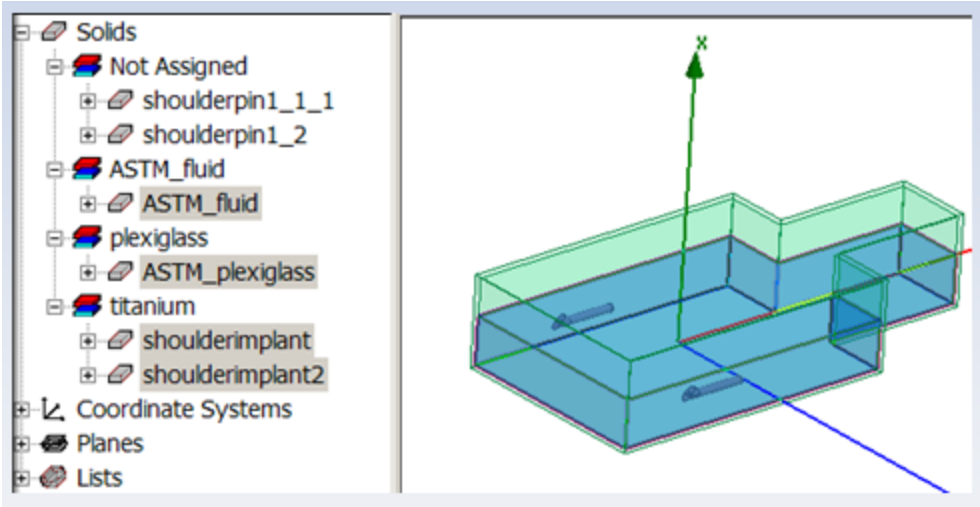
- DX analysis
- WB Update Project
- WB Update All Design Points
    - Parametric analysis with DSO
    - Animation
    - Geometry
    - Field plots



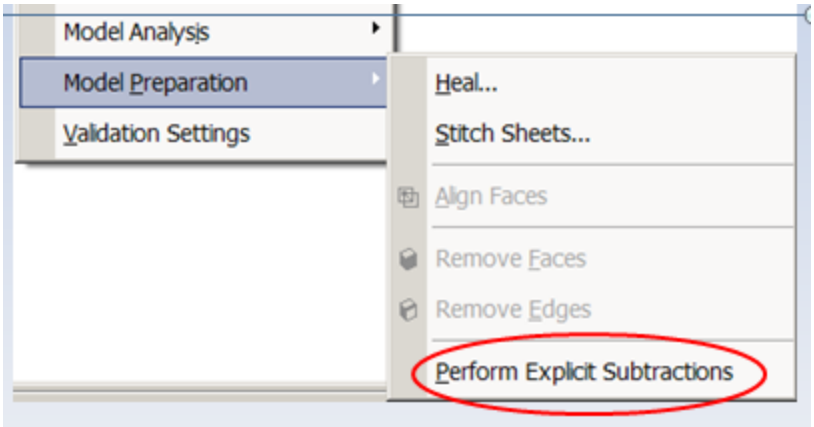
Ansys Mechanical does not do implicit subtraction. Ansys Electromagnetics products can do explicit subtractions before sending geometry to Ansys using **Modeler > Model Preparation > Perform Explicit Subtraction**.

For example, consider the following model.

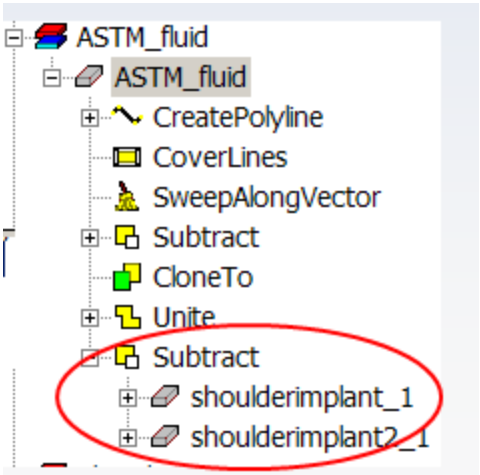
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Perform Explicit Subtractions can be performed.

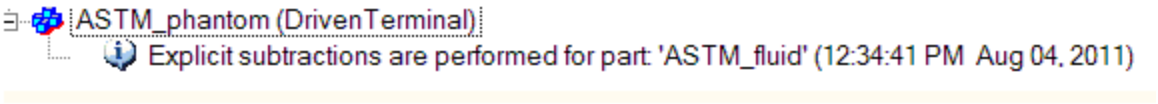


Results appear in the History tree, as shown:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Messages** window reports this action.

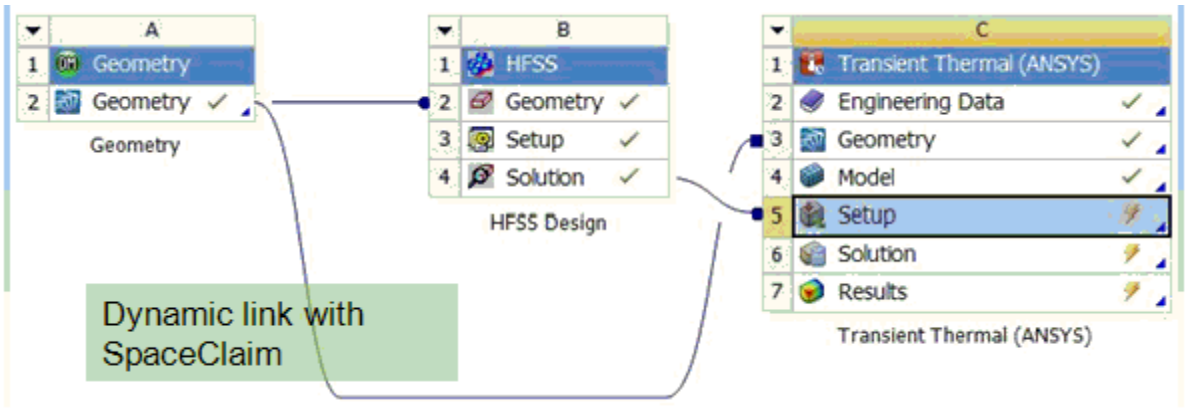


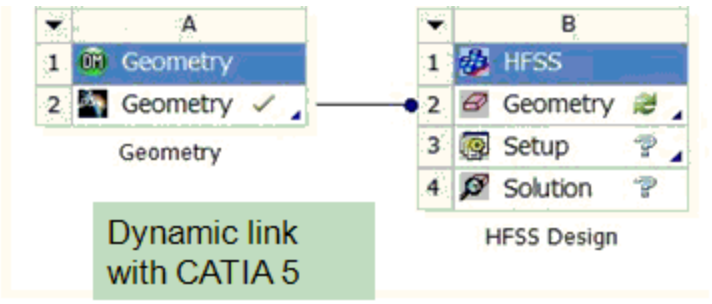
CAD Integration and Geometry Sharing

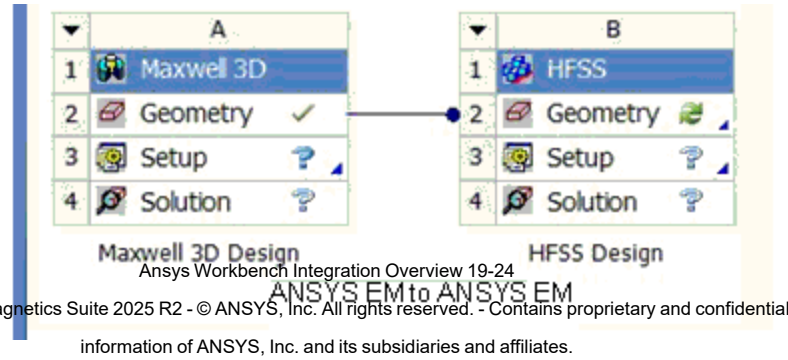
A CAD model comes into Ansys Electromagnetics as a User Defined Model (UDM).

The inputs to Ansys Electromagnetics from CAD are

- Geometry/Topology with persistent IDs
- CAD parameters
- Material assignment
- Attributes, such as name and color

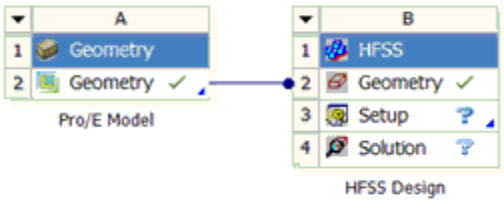




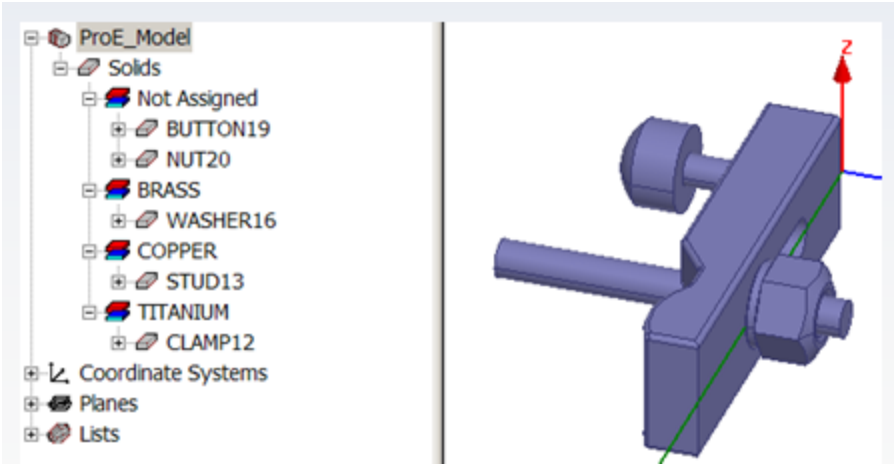


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, in Workbench, a Pro/E Model can be linked to Ansys:



The geometry can then be viewed as a UDM.



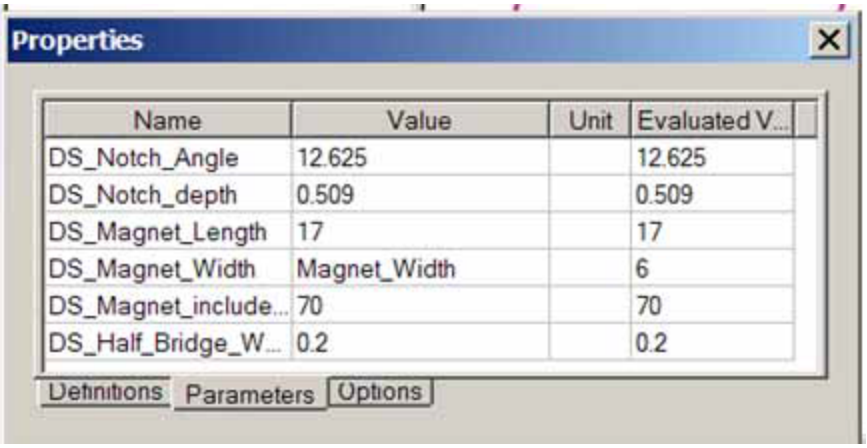
The CAD or WB model parameters appear in the Workbench:

Outline of All Parameters

	A	B	C	D
1	ID	Parameter Name	Value	Unit
2	Input Parameters			
3	DM Geometry (A1)			
4	P5	DS_Notch_Angle	12.625	
5	P6	DS_Notch_depth	0.509	
6	P7	DS_Magnet_Length	17	
7	P8	DS_Magnet_Width	6	
8	P9	DS_Magnet_included_Angle	70	
9	P10	DS_Half_Bridge_Width	0.2	
*	New input parameter	New name	New expression	

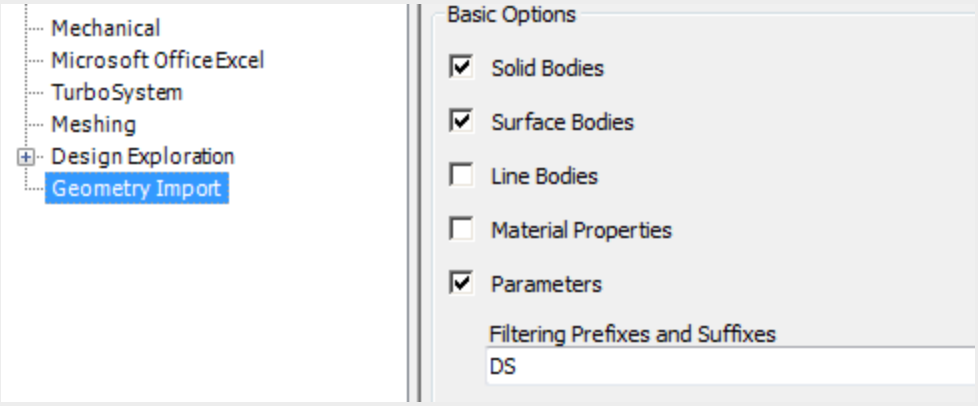
Through the Ansys Electromagnetics CAD integration, the linked UDM includes the same parameters.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



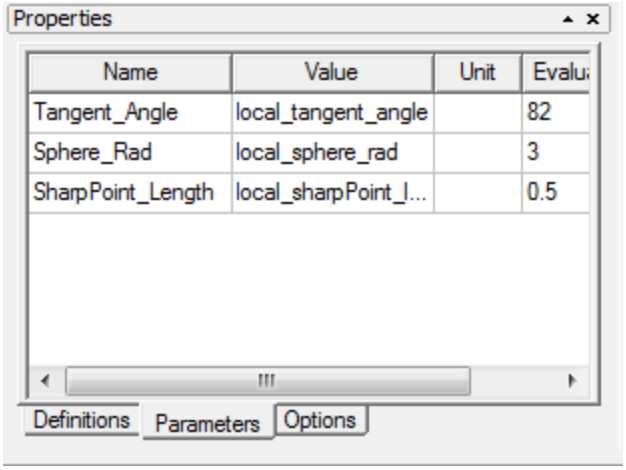
Note:

The parameters shown in these examples all have a DS prefix. This is the default for the Workbench **Tools** > **Options** for **Geometry Import**. To import parameters with different prefixes or names, assign an appropriate prefix or clear the **Filtering Prefixes and Suffixes** field.



Once you import a geometry with parameters to an Electromagnetics application in the desktop, you need to map to local variables. In the target Electromagnetics application, select the geometry associated with the parameters. Update the geometry and view the **Parameters** tab in the **Properties** window. The **Value** column shows the values of the imported parameters. Type names for local variables in the **Name** column.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Once you have local variables for the imported Design Modeler parameters, you are ready to use Design Xplorer for multiple design parameters.

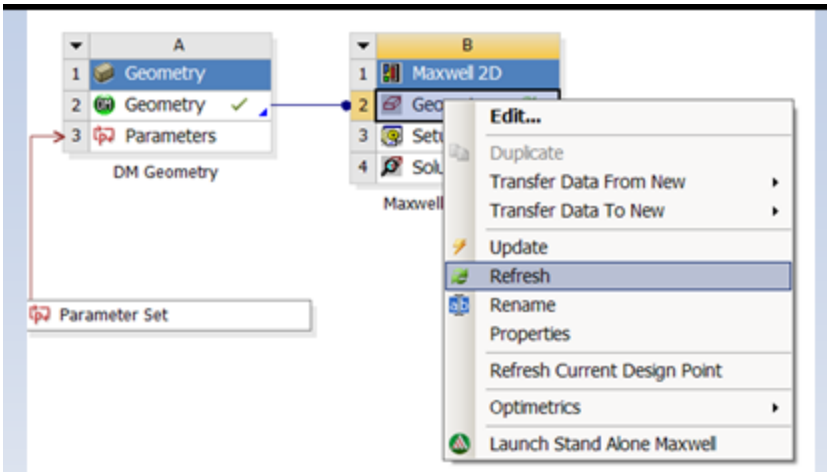
Table of Schematic D2: Design of Experiments (Custom)

	A	B	C
1	Name	P3 - Tangent_Angle (degree)	P4 - Sphere_Rad (in)
2	1	80	3
3	2	75	3
4	3	85	3
5	4	80	2.7

Bi-Directional CAD Integration

Ansys Electromagnetics CAD Integrations uses **Refresh** or **Generate** to pass updates.

You make an edit in a CAD application and run **Refresh** on an Ansys Electromagnetics Geometry Cell.

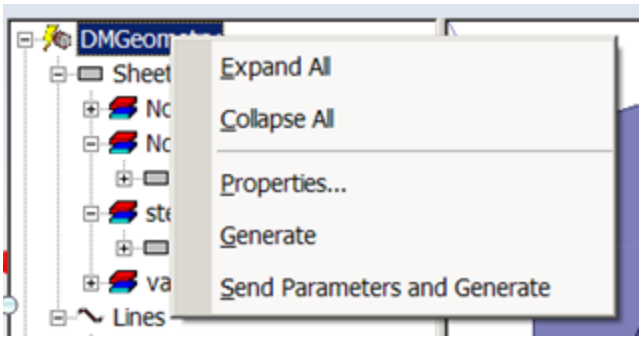


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Refresh pulls the current state of CAD model (geometry, parameters, materials etc) and updates the corresponding data in the Ansys Electromagnetics application.

Alternatively, you can run **Generate** on the UDM in the Ansys Electromagnetics window.

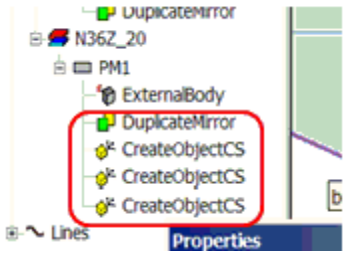
If you edit UDM (CAD) parameters in the Ansys Electromagnetics modeler window you can run the **Send Parameters and Generate** command.



The command passes the edited parameters to the linked CAD application and then pulls corresponding CAD geometry.

CAD Integration Model Edits

Several modeling operations are allowed on a CAD model in the Ansys Electromagnetics Modeler window. Operations are included in the History tree and retained during model **Refresh**.

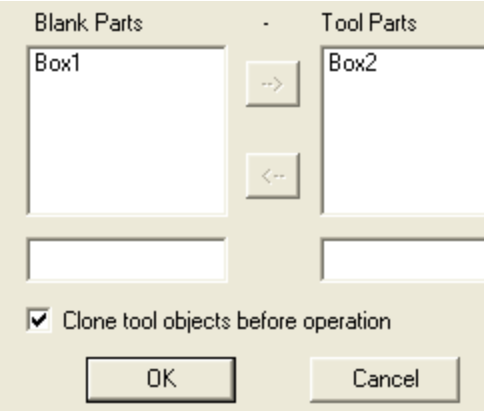


The following operations are not allowed:

- Non-history tree operations like Heal or defeature.
- Operations that use UDM parts as tools, such as Sweep

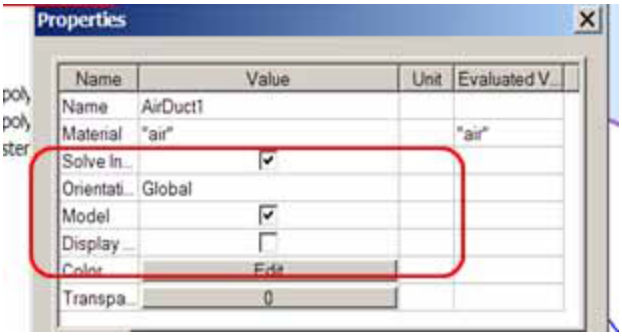
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Boolean operations like Split or Unite (allowed when you select the clone tool option)



The following part attributes can be modified for UDM parts:

- Model/Non Model flag
- Solve Inside flag
- Part orientation
- Color
- Display Wireframe



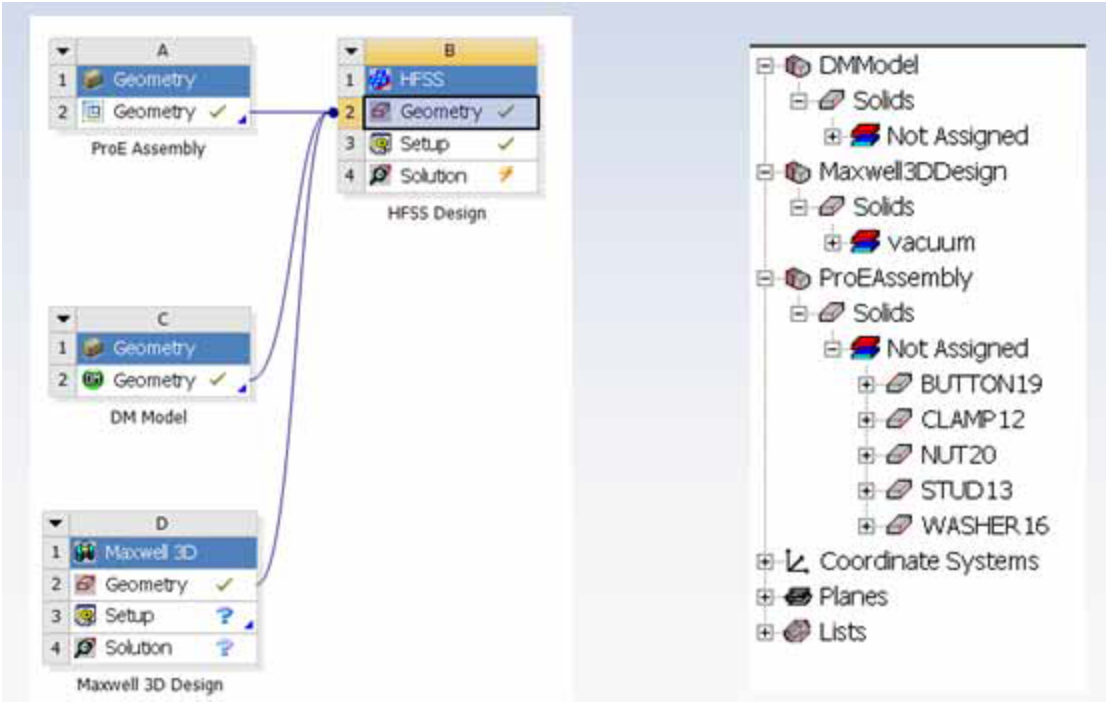
It is not possible to delete individual parts of UDM.

Multiple Geometry Links for CAD Integration

With CAD Integration you can consume geometry from multiple upstream sources. The source can be any of supported CAD or Ansys Electromagnetics products. This creates a UDM for each geometry source.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

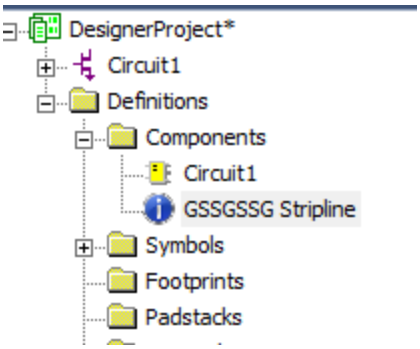
For example, the following figure shows a DM Model, a Maxwell model, and ProE model linked to HFSS in Workbench, and displayed in the HFSS History tree as three UDMs.



Creating Dynamic Links to EDT Designs

If you have a Circuit Design and one or more HFSS, 2D Extractor, Q3D, Icepak, or Mechanical designs open in a Workbench project, you can create dynamic links to those designs.

Update the Circuit Design to open the Electronics Desktop. In the menu bar, select **Workbench** > **Update Dynamic Link Components**. This command will create components in the project tree for each design in the Workbench project.

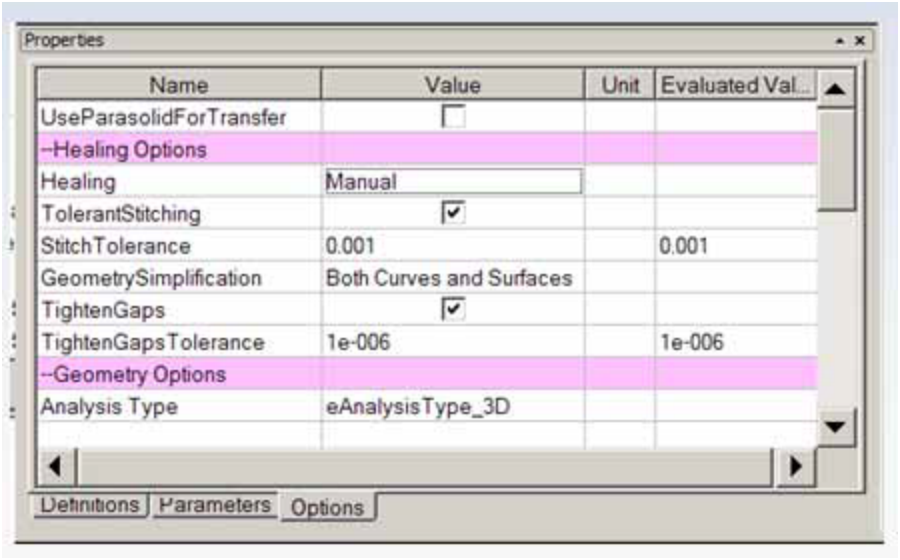


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that in this example, a GSSGSSG Stripline component was created corresponding to the GSSGSSG Stripline design in WorkBench. Once created, you can drag the component into a schematic to place it.

Healing with CAD Integration

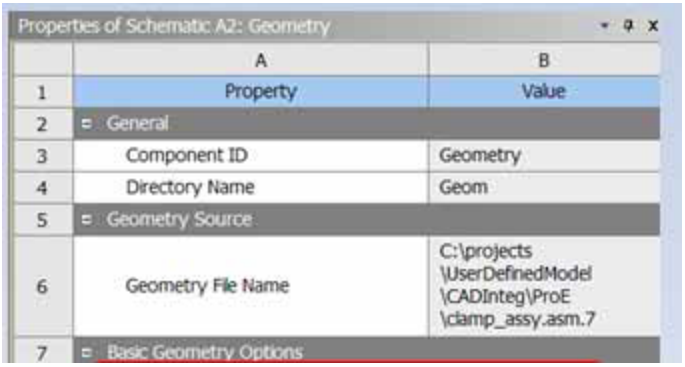
It is not possible to use the Heal command in Ansys Electromagnetics Modeler. Instead similar healing options are available under UDM properties option tab.



Healing options are None, Auto and Manual. By default, healing is off (None) and should be turned on only if required.

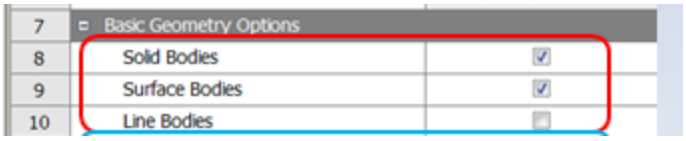
Important Geometry Options for CAD Integration

Select a Geometry Cell in Workbench to see options in the Properties window.



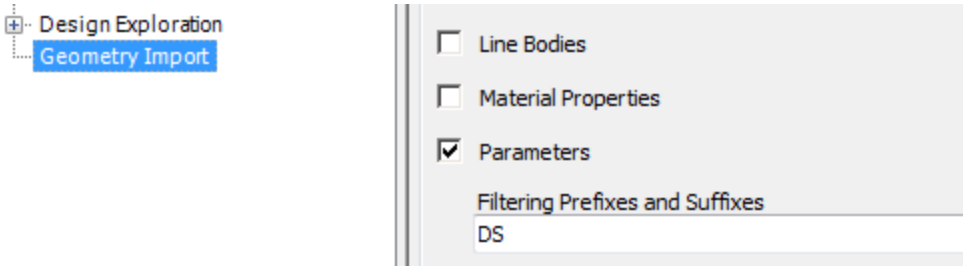
Control dimension of bodies coming from CAD

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Make sure **Parameters** is checked and that the parameter key (filter) is appropriate for CAD parameters.

The default parameter key filtering means that only parameters whose names start with DS will come through. This is the default for the Workbench **Tools>Options** for **Geometry Import**. To import parameters with different prefixes or names, assign an appropriate prefix or clear the **Filtering Prefixes and Suffixes** field.



The **Attribute Key** should be empty or "Color" to bring in CAD colors.

Material Properties must be checked to bring in the material assignment.

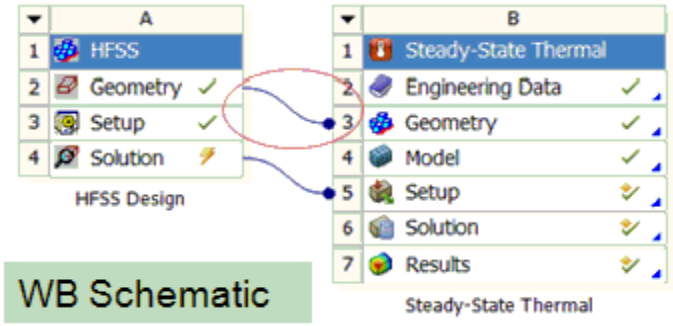
The **Mixed Import Resolution** option is used to resolve parts with mixed dimension (typically from Pro/E)

Refer to Ansys Help for details.

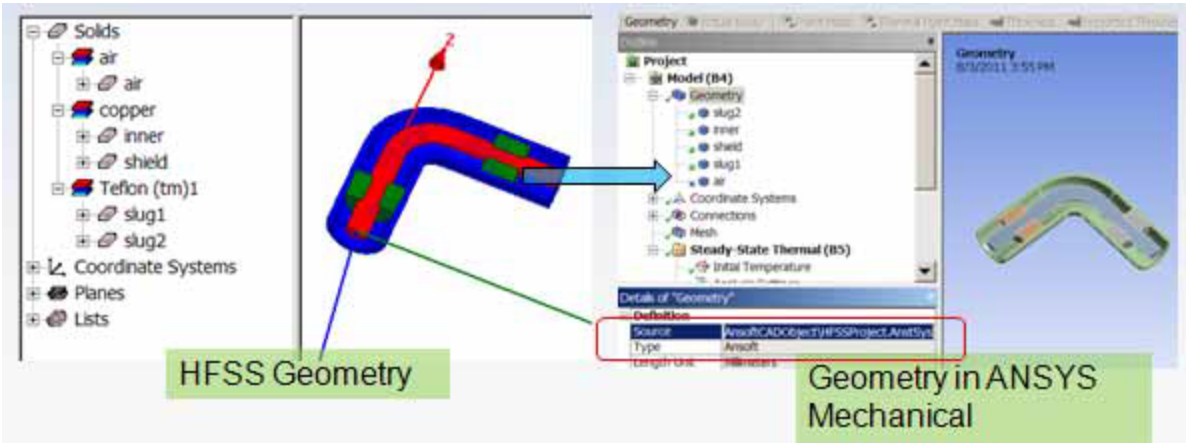
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys EM to Ansys Geometry Transfer

Ansys Electromagnetics CAD Integration transfers model information based on Workbench links.

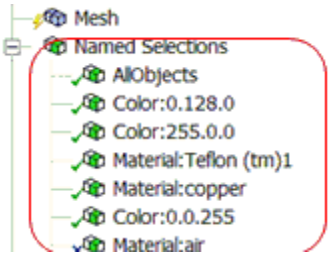


The following figure shows how the information is transferred between simulators.



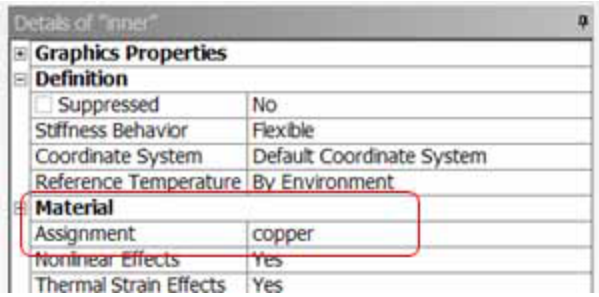
Information transferred includes

- Geometry
- Ansys Electromagnetics lists and material assignment as Named Selection



- Material assignment

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The CAD Integration geometry link is dynamic (you can get updated geometry from Ansys Electromagnetics) and associative (IDs persist between Ansys Electromagnetics model and Ansys model during model refresh).

Boundary conditions in Ansys are preserved. Object, material, and parameter names with non-ASCII characters are not allowed for data transfer. Such transfers fail and produce an error message.

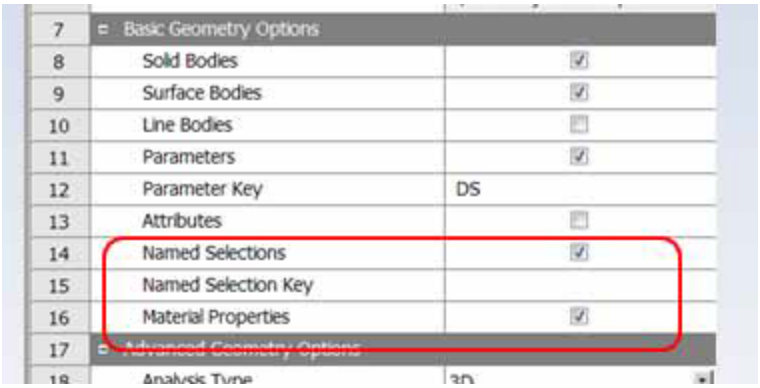
Material Assignment Transfer

There are two prerequisites for material assignment transfer:

- Engineering Data should have materials used in Ansys Electromagnetics model and material names should match (including case)
- Material Properties in Geometry Options must be checked

Before transferring as a named selection, **Named Selections** must be checked in **Basic Geometry Options**.

The Named Selection Key should either be empty or contain "Material."

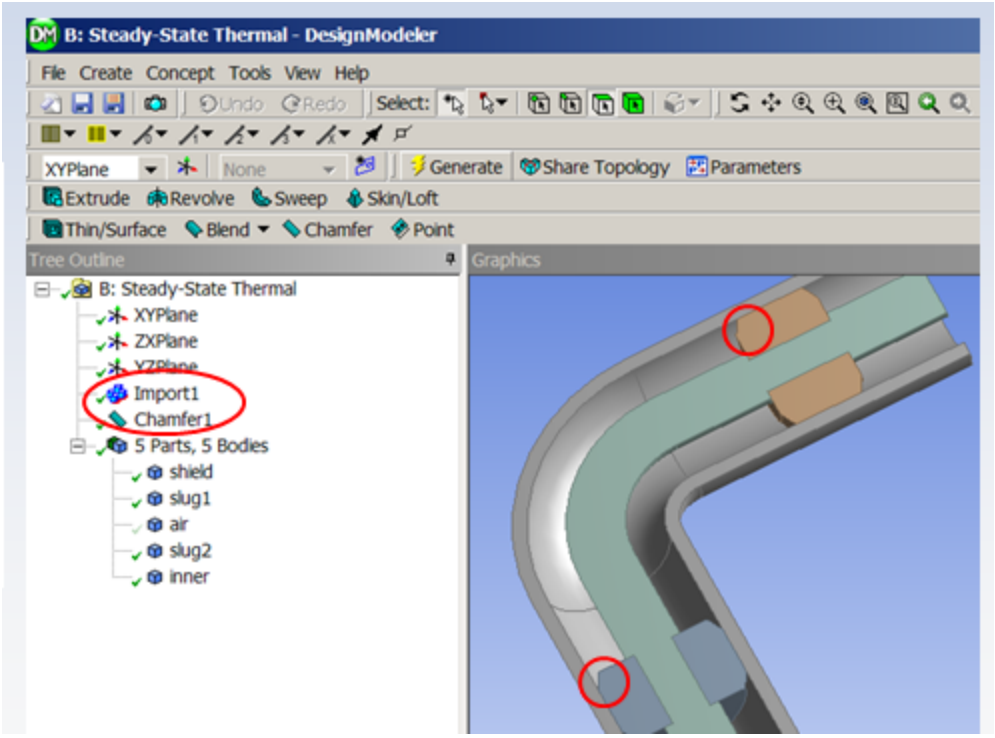


Geometry Transfer through Ansys DesignModeler (DM)

It is possible to edit geometry in Ansys DesignModeler (DM) before consuming it in Mechanical. This is useful when geometry requires preprocessing for Ansys simulation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

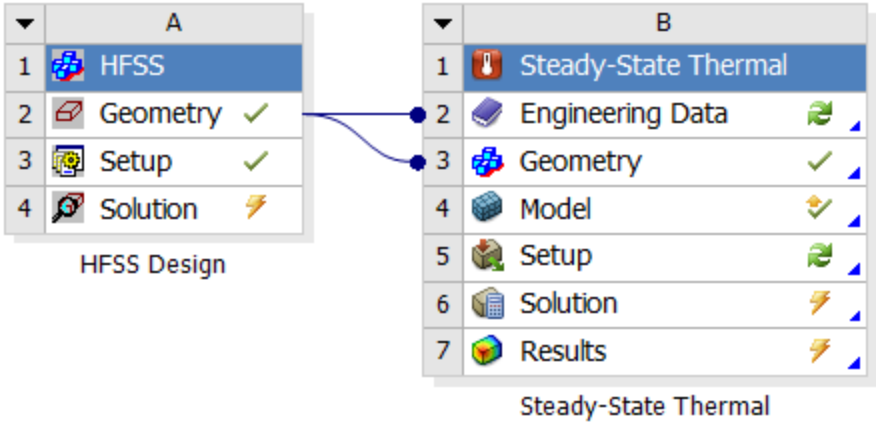
Consider an HFSS model linked to DM through the Workbench. In the following example, a chamfer operation on geometries is being imported.



Workbench Material Data Transfer

A link can be created from an Ansys Electromagnetics Desktop system (e.g., HFSS) geometry cell to a downstream Engineering Data (ED) cell. When the downstream ED cell is refreshed, it updates with materials (names and properties) used by the upstream project. This ensures that the geometry-to-geometry link connecting the same two systems, which specifies the material assignments for various parts, refers to the correct materials and their properties.

Detailed Behavior



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The geometry-to-ED link can be created by dragging the Ansys Electromagnetics Desktop system geometry cell to the ED cell of a downstream system. When the downstream system's ED cell refreshes, the upstream system generates a MatML format XML file, which is consumed by Workbench to update the downstream system. This file is stored in the workbench project files according to design point. Note that Workbench ED cell material names are case-sensitive, as are material names within Ansys Electromagnetics Desktop products.

For material assignments to work with the ED in the downstream model, the downstream geometry cell must have the **Material Properties** check box selected. A convenient way to do this is to have this property checked by default, by setting this in **Tools** > **Options** > **Geometry Import**. Systems created after this default option is set will have **Material Properties** automatically checked for the geometry cell.

When the downstream model cell is refreshed or updated, material properties from the ED in the same system are used in the model's materials, based on material assignments set during the geometry cell refresh. Note that if material assignment fails, the Workbench default material (Structural Steel) is used.

Workbench resolves name conflicts by changing the name of any materials being transferred over from the upstream system. For example, if the upstream Ansys Electromagnetics Desktop system uses a material named "Structural Steel", this will conflict with the existing downstream default material of the same name. The update will result in the Ansys Electromagnetics Desktop material being named "Structural Steel 2" in the downstream ED, but the downstream model will still use the properties supplied by Ansys Electromagnetics Desktop for the material.

Note that some materials may not satisfy all requirements for downstream system physics. For example, vacuum has no thermal conductivity, but downstream steady state thermal systems require a non-zero value for this property. Material properties in the workbench ED cell of the downstream system can be modified after refresh to ensure that validity criteria can be met. Any property added to a material in the downstream ED is preserved in subsequent ED cell updates. However, any existing property that is edited in the downstream ED will be overwritten by a subsequent ED cell update.

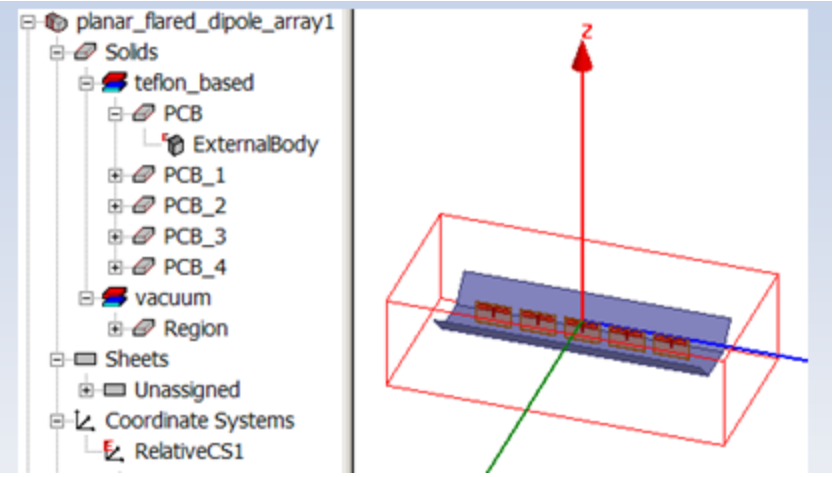
User-Defined Model (UDM)

A User-Defined Model (UDM) is collection of externally defined parts imported into an Ansys EM 3D Modeler or created using a C or Python script. It has the following attributes:

- UDM includes part attributes (like name, color etc) and material assignment
- UDM can also have external coordinate systems and corresponding planes
- UDM parts can be parameterized and manipulated in Ansys Electromagnetics modeler

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

just like any other part

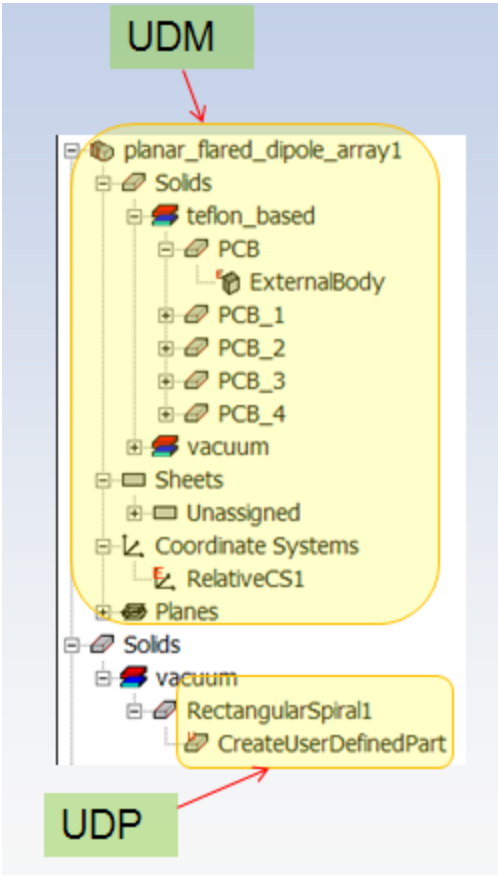


- UDM can either represent Static geometry models or Dynamic links to models of external geometry editors.
- It can be used for supporting CAD integration in WorkBench.
- UDM uses same plugin technology as User Defined Part (UDP)

UDM Compared to User-Defined Primitives

User-Defined Models (UDM) resemble User-Defined Primitives (UDP):

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics products can be extended by users through new UDMs.

UDM plugins are discovered by searching standard directory paths. Plugins for static UDMs can build a model using ‘callback interfaces’ (like create-box, create-cylinder, subtract etc) similar to UDP.

UDMs run inside Ansys Electromagnetics application and provide geometry, topology, persistence, and parameters.

In contrast to UDP:

- A UDM provides multiple parts/CS/etc.

A UDP provides primitive operation only for a single part.

- A UDM provides part attributes and material assignment.

A UDP does not define part attributes or material.

UDM Properties have four tabs – Definitions, Parameters, Options and Info.

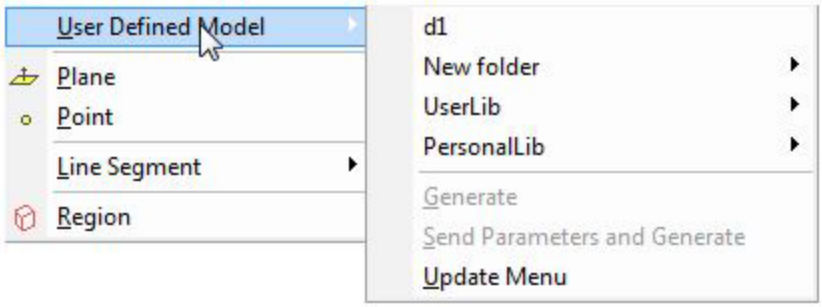
- Definition tab has UDM name, and the coordinate system used to position UDM. It may have external reference to file.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Info tab has UDM dll name, dll location, version, *etc*.
- Option tab may have options.

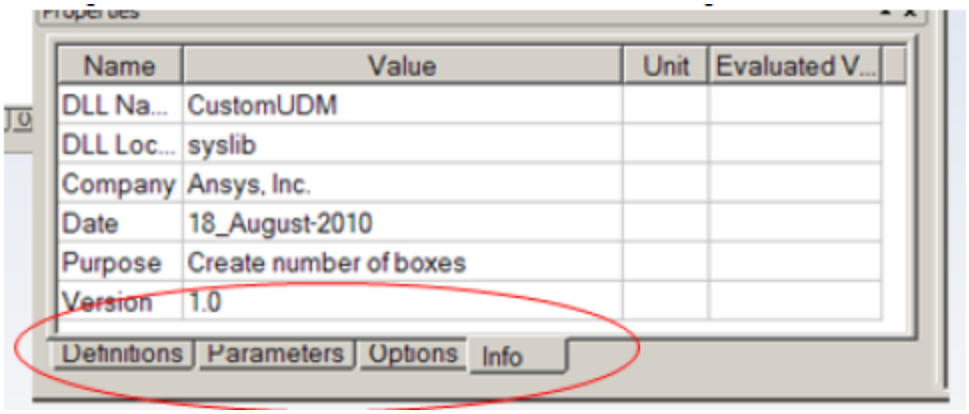
Insert UDM Command on Draw Menu

To insert a UDM into a design, use the **User Defined Model** command on the **Draw** menu for the Modeler window.



UDM Properties

UDM Properties have four tabs – **Definitions**, **Parameters**, **Options** and **Info**.



Definition tab has

- the UDM name
- Coordinate system used to position UDM
- May have external reference to file

Info tab has the UDM dll name, dll location, version, *etc*.

Option tab may or may not have any options.

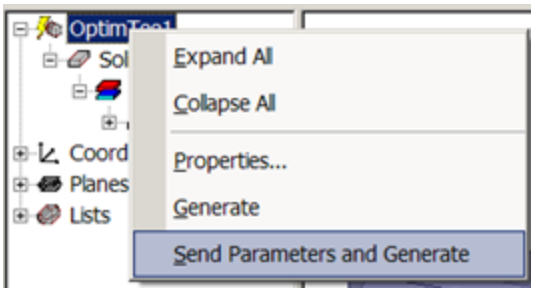
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UDM Parameters

UDM geometry can be manipulated through its parameters. It can be mapped to a design or project variable for animation or parametric analysis.

IDs are persisted (allowing to retain boundaries) during a parameter edit. Note that UDM geometry is not dynamically updated upon parameter edits. The UDM shows a lightning bolt icon by the model name when parameters are edited:

You must run the **Send Parameters and Generate** command to synchronize parameters with geometry:



UDM Part Edits

Several modeling operations are allowed on UDM parts. Operations will be part of the history tree and will be retained during a model refresh.

The following operations are not allowed:

- Non-history tree operations like healing, defeature.
- Operations that use UDM parts as tool, such as sweep or boolean (but allowed when clone tool option is selected)

The following part attributes can be modified for UDM parts:

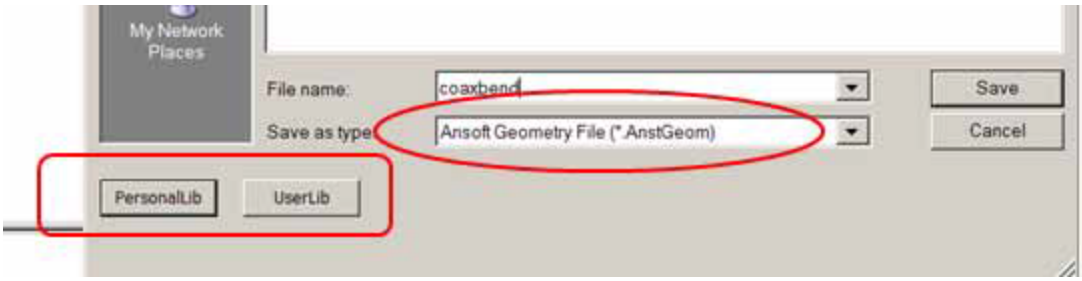
- Model/Non-model flag
- Solve Inside flag
- Part orientation

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

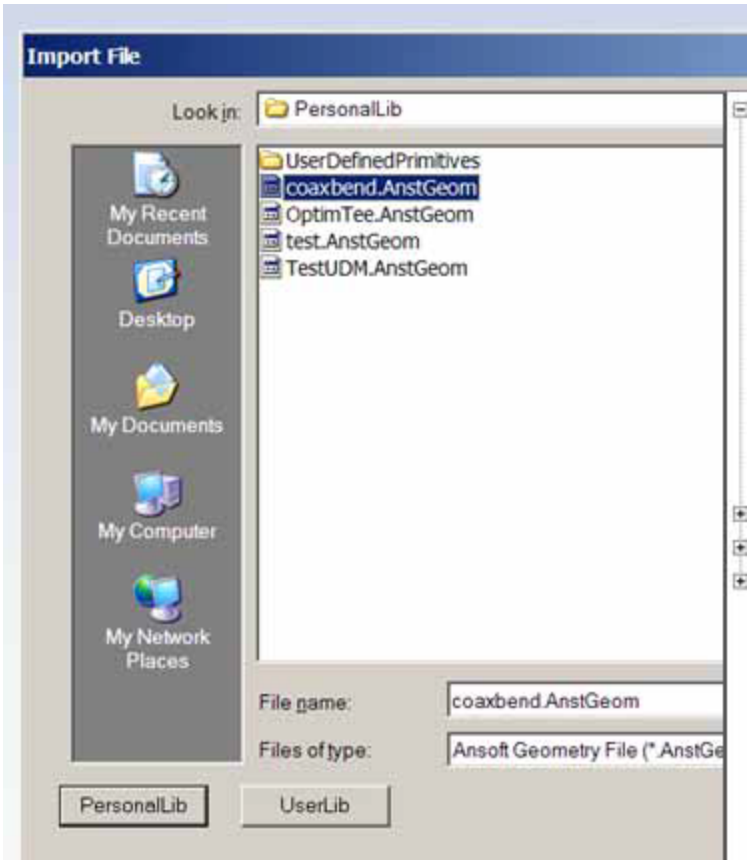
Library of Models for CAD Integration

UDM technology allows a library of models

- Any Ansys EM model can be exported as 'Ansoft Geometry File'

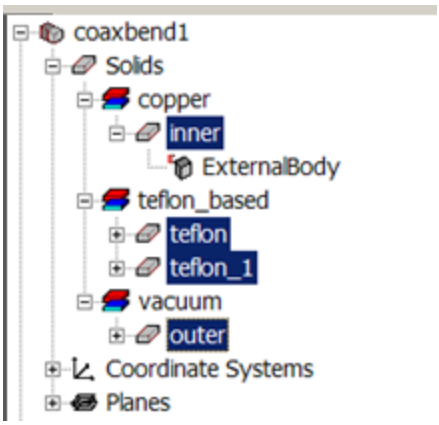


- An Ansoft Geometry File can be imported back as a UDM.



- Any Design/project variables associated with model are brought in as UDM parameters.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Geometry computation for UDM (and also UDP) can be specified in the Modeler Options as either done on.

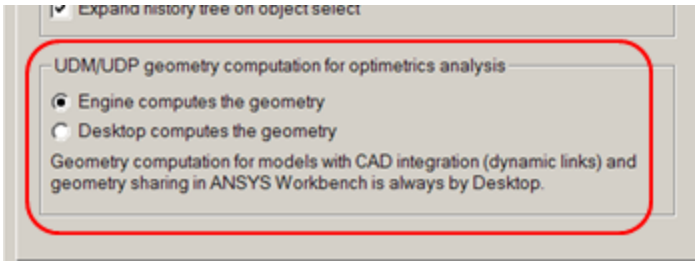
- Engine side (default):

  Requires deployment of UDM on each node

- Desktop side:

  UDM need not be deployed on each engine

  Desktop will be busy during parametric analysis



Feedback Iterator

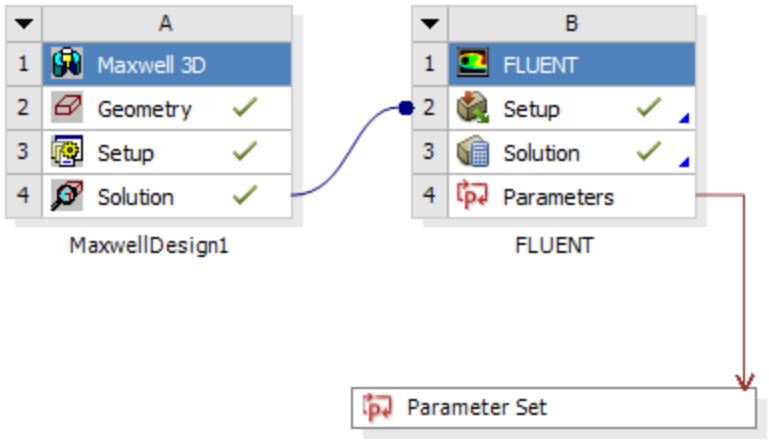
Background Information

Prior to the introduction of the Feedback Iterator, Ansys Workbench supported a two-way loose-coupling protocol with Ansys Electromagnetics products.

System Coupling already uses the word coupling to mean low-level solver coupling. Existing electromagnetics product coupling is loose compared to system coupling and is limited to file transfers at the end of a complete solve in a standalone system/product. No communication occurs during a solve. The coupling is two-way.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The one way portion (Upstream source component to Downstream target component) is handled via existing Workbench data/transfer connection mechanism.

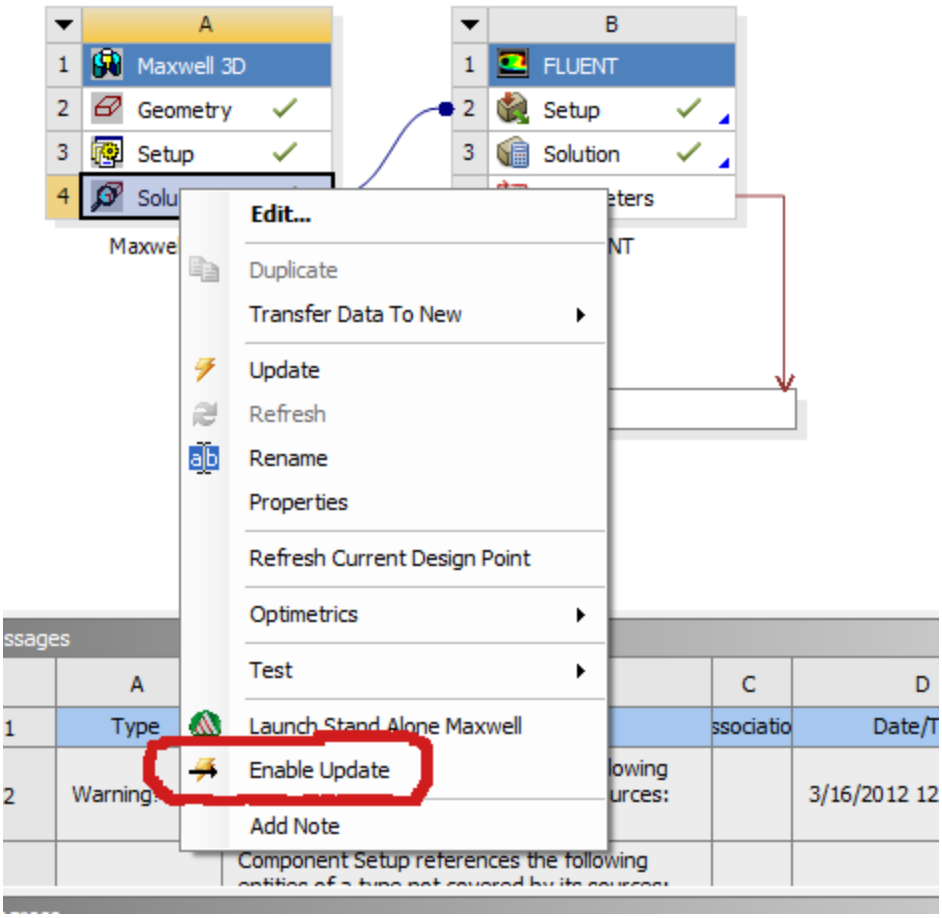


The round trip is handled by a separate protocol, agreed upon by the participating systems, whereby the downstream system exports a set of files to a location specified by the upstream system via its one-way transfer. This exported data is then incorporated by the upstream system in its next update.

To run the next coupled solve iteration, invoke the **Enable Update** operation, as shown below, which updates all systems involved.

These steps (**Enable Update**, **Update Project**) are continued as long as needed.

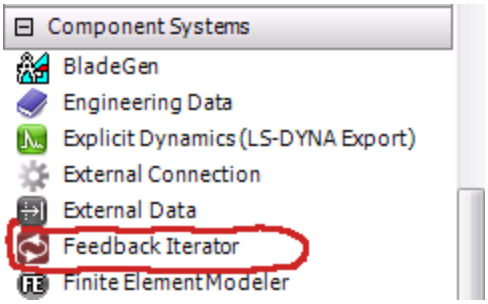
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Feedback Iterator System

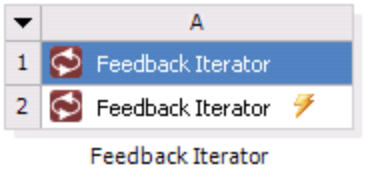
Ansys Electromagnetics Suite provides the Feedback Iterator system for automating the manual steps needed for driving a feedback utilizing system-pair to convergence. In addition to automating feedback incorporation over a user-specified number of iterations, the Feedback Iterator also allows control over the number of iterations (iteration termination criteria), target temperature, and displacement convergence criteria.

The **Feedback Iterator** appears in the Workbench Toolbox user interface under **Component Systems**.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

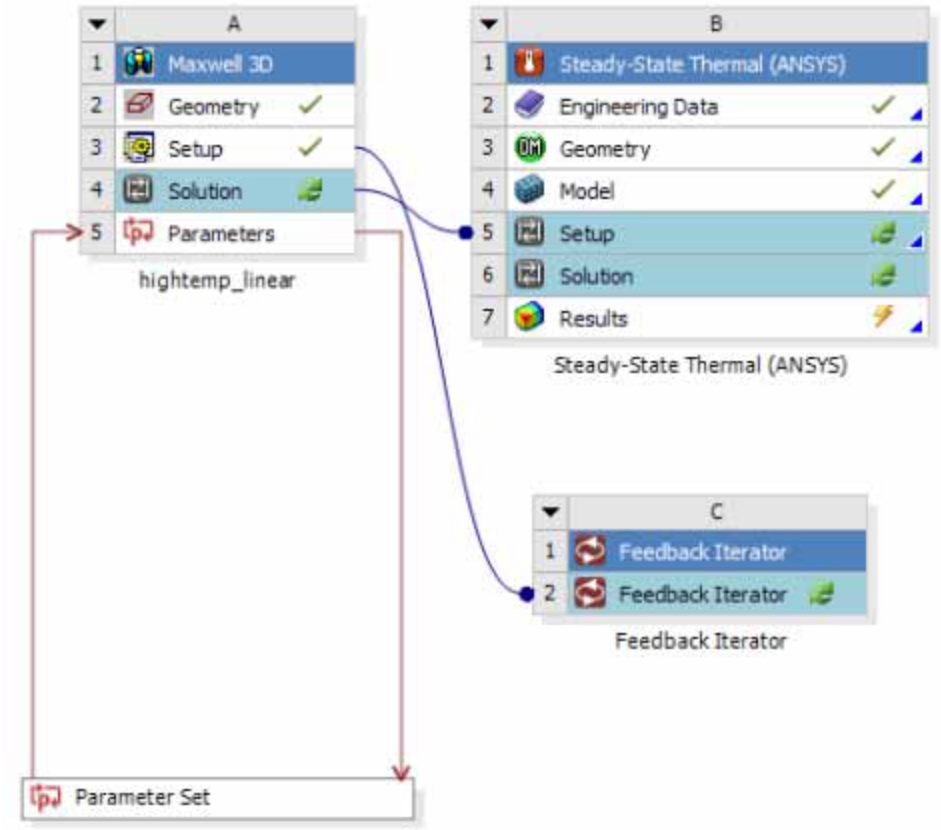
If you drag and drop an instance of the Feedback Iterator into a Workbench schematic, the default system name is Feedback Iterator:



Feedback Iterator in Use

The Feedback Iterator system attaches to the Workbench schematic like any other system.

In the following example, a Maxwell system provides heat loss data to aFluent system. Dropping the Maxwell setup cell onto the Feedback Iterator component results in the links shown:



The Feedback Iterator add-in functions the same way with all systems that participate in the Electromagnetics product two-way loose coupling (Maxwell and HFSS for sources and Fluent, Thermal, and Structural for targets).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Feedback Iterator Component Properties

The Feedback Iterator's primary role is to control iterative solves and its properties target this end.

For Maxwell and HFSS, the Feedback Iterator properties are

- **Iterations Completed** – a read-only property that displays the number of iterations completed.
- **Max Iterations** – sets the maximum number of iterative solve loops to perform before terminating iterations in case the *Target Delta Temperature %* or *Target Delta Displacement %* is not achieved. The default value is 100.
- **Target Delta Temperature %** – specifies the maximum *Target Delta Temperature %* that signifies convergence. The default value is 5%, and the value must be 0.01 or greater.

  > **Note:**
  >
  > • The delta temperature error is calculated in Kelvin.
  >
  > • The Maxwell and HFSS design **Profile** tab displays absolute/relative delta data, while the delta temperate in Workbench means the relative delta (expressed as %).

- **Target Delta Displacement %** – specifies the maximum *Target Delta Displacement %* that signifies convergence. The default value is 5%, and the value must be 0.01 or greater.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	A	B
1	Property	Value
2	⊟ General	
3	Component ID	FeedbackIterator
4	Directory Name	FeedbackIteratorComponent
5	⊞ Notes	
7	⊞ Used Licenses	
9	⊟ Iterations	
10	Iterations Completed	0
11	Max Iterations	100
12	⊞ Callback	
14	⊟ Temperature Convergence	
15	Target Delta Temperature %	5
16	Latest Delta Temperature %	Not Available
17	⊟ Displacement Convergence	
18	Target Delta Displacement %	5
19	Latest Delta Displacement %	Not Available

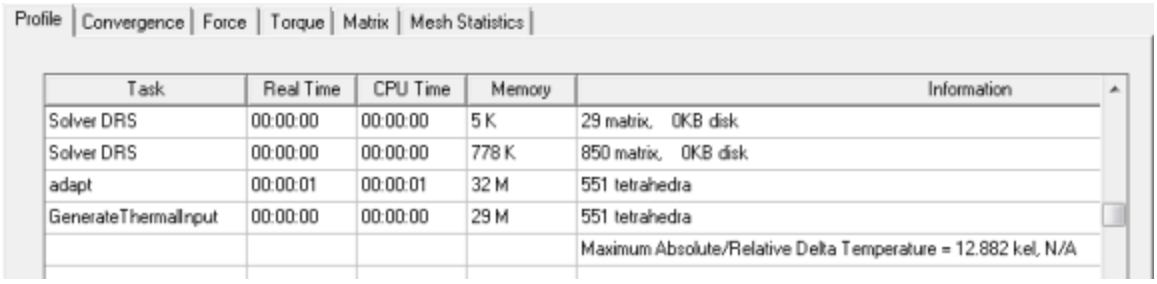
The Feedback Iterator properties also display the most recently achieved *Latest Delta Temperature %* and *Latest Delta Displacement %* values, allowing the user to manually abort when satisfied with the achieved deltas.

> **Note:**
>
> Both the Temperature and Displacement Convergence properties sections are always shown irrespective of which feedback types are enabled for the project.

Running a two-way feedback simulation from the Workbench application while keeping the Maxwell or HFSS **Profile** tab open shows that the 3D solver keeps track of two feedback related quantities:

- Maximum Absolute/Relative Delta Temperature (if temperature feedback is enabled)
- Maximum Absolute/Relative Delta Displacement (if displacement feedback is enabled)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

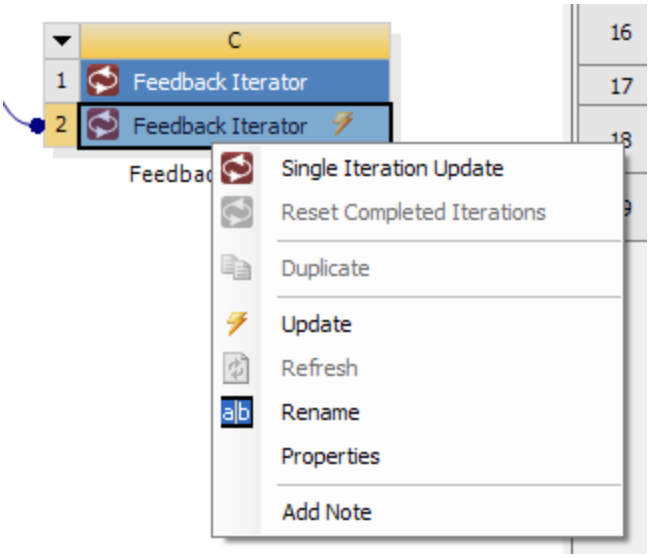


These signify the maximum difference on the solution mesh of the quantity in question from one feedback iteration to the next (thus the value is only available from the second iteration onward). This data forms the basis for the convergence controls in the Feedback Iterator. These controls allow the user to base the termination of feedback iterations on a target max delta T and/or max delta displacement. The target max delta forms the convergence condition.

Feedback Iterator GUI Operations

The **Single Iteration Update** action allows you to run a single iteration worth of updates, as follows:

1. Updates the Electromagnetics product system, which will incorporate any previous feedback from the downstream system.
2. Updates the downstream system.
3. All coupled systems (Electromagnetics product, Electromagnetics product Downstream, and Iterator) are in the UpToDate state.
4. Iterator's “Iterations Completed” property increments by 1.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Right-clicking the Feedback Iterator component and selecting **Update** allows you to run iterations automatically. Iterations continue to run either until convergence criteria are met, or until the Max Iterations value is reached.

> **Note:** After getting temperature feedback from Fluent, Maxwell solving starts from the *last adaptive*, which means there is NO more adaptive pass solving. Consequently, instead of looking for the expression cache value in the adaptive solution, users should look for it in the *last adaptive* solution. For *Last Adaptive*, the expression cache value can be displayed from **Results** > **Create Fields Report** > **Data Table**.

Resetting the Feedback Iterator

Resetting the iterator essentially means starting from iteration-0 conditions, that is:

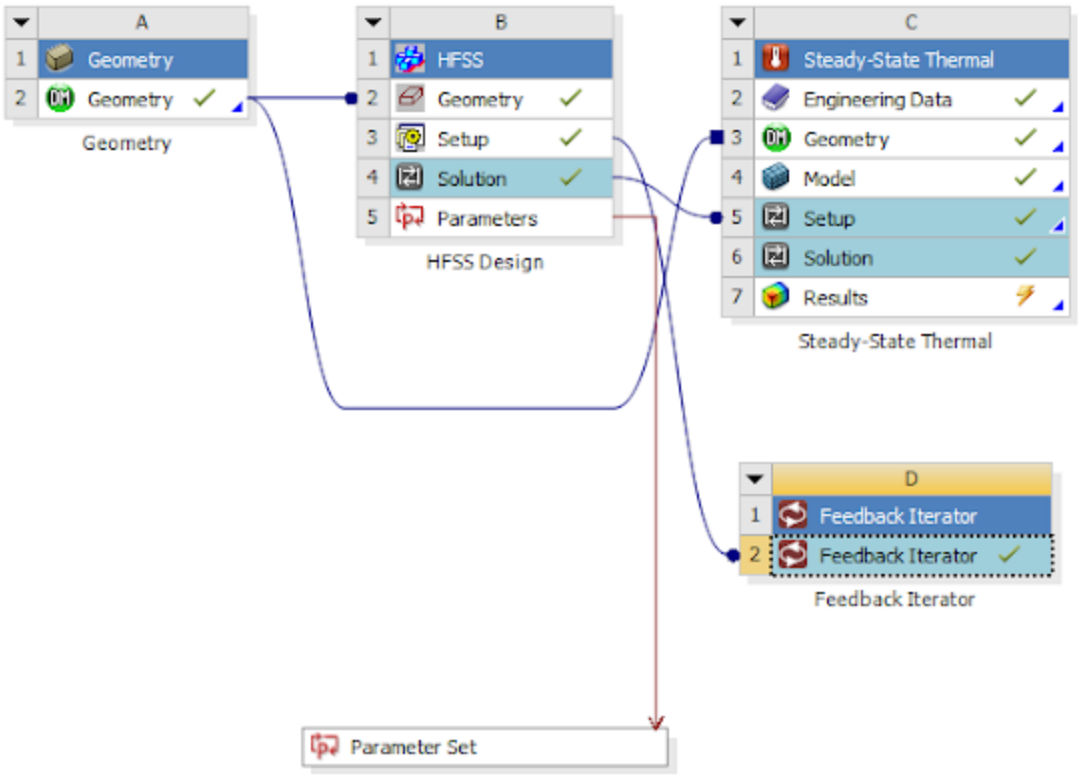
- No Displacement or Temperature feedback from another system is incorporated.
- The first iteration solution will be purely an Ansys EM analysis solution.

Resetting the Feedback Iterator cannot easily be done entirely within the Workbench application. However, the following two processes can be used:

- Resetting via the Ansys EM user interface
- Resetting via a Workbench journal script

The following example uses an HFSS/Thermal project. (This same approach applies to the other Ansys EM design types.)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



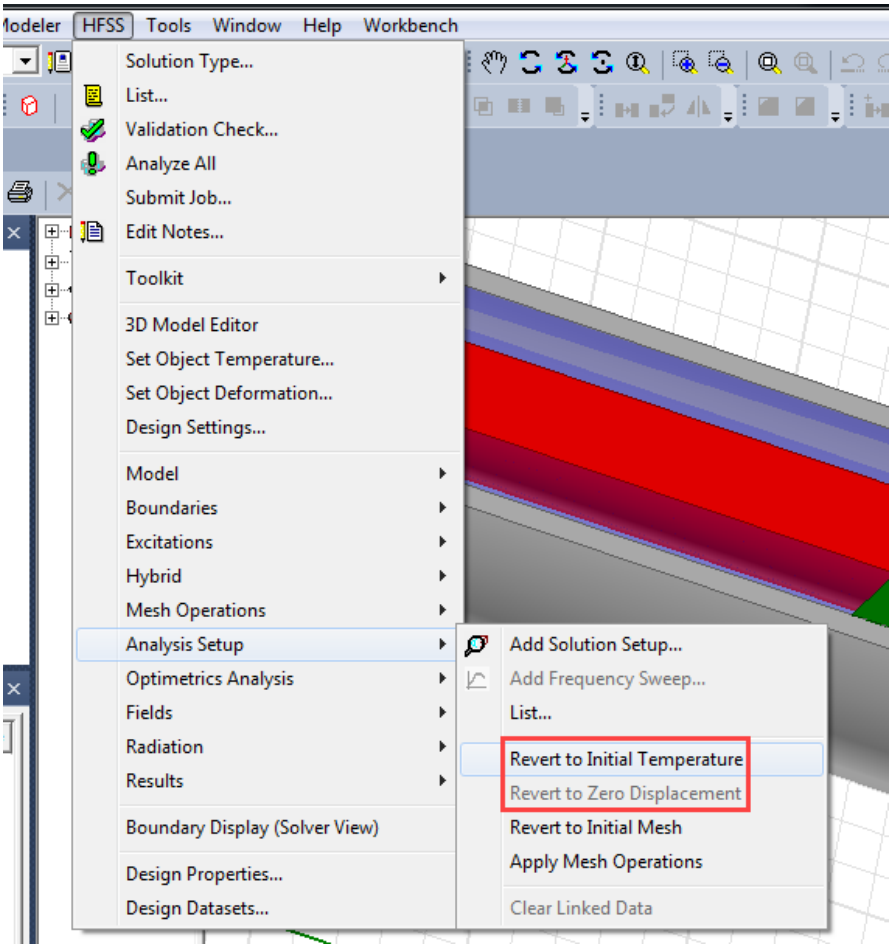
Resetting via the Ansys EM User Interface

Depending on the type(s) of feedback involved, this typically means reverting to initial conditions for one or both of the following settings:

- **Revert to Initial Temperature**
- **Revert to Zero Displacement**

You can find these under the respective design type menu, such as the **HFSS** menu shown below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Resetting via a Workbench Journal Script

> **Note:** The above Ansys EM user interface actions can be recorded into a Workbench journal script and played back for the easiest approach.

You also can build a script from scratch using the following as the template – simply update the system name as appropriate, include the appropriate "Revert..." commands, and either save as a script or enter into the Workbench Command window.

```
oDesign = Ansoft.GetDesign(GetSystem(Name="HFSS"))

oDesign.GetModule("AnalysisSetup").RevertAllToInitialTemperature()

oDesign.GetModule("AnalysisSetup").RevertAllToZeroDisplacement()
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Callback Interface

The callback interface allows you to react to each step of the iteration process as implemented by the Feedback Iterator component. This is useful for implementing special iterations or solving transient simulations with a limited scope.

There are four main API features provided:

- Callback functions, which the Feedback Iterator calls at various points in each iteration
- Utility functions, which the Callback functions can use to extra properties of the containers being processed
- Output functions, which allow script debugging and supply additional messages
- Limited state management, which allows the script to store and retrieve state across callback functions and iterations

Callback and State API

State is managed in a limited fashion. All the API methods take a final dictionary argument. This dictionary is limited to using **string keys** and **number or string values**. Within this limitation, add new keys, read old keys, clear the dictionary, etc., and it will be persisted across functions calls and even across Updates. When the **Iterations Completed** property is **rest**, the dictionary is also cleared out.

It is advisable for the callback script to initialize the dictionary at Iteration 1.

All the API methods use a subset of the following arguments:

`iterationNumber`	An integer representing the current iteration. This always starts from 1.
`ContainerList`	A Python list of containers (DataContainerReference). This is the entire list of the coupled containers managed by the Feedback Iterator. You typically loop through them and using the utility methods listed below, identities them.
`Container`	A single container that is being processed. This is a DataContainerReference.
`State`	A read/write Python dictionary that is used to maintain state across function calls and iterations.

Only return values from `BeforeIterationEx` are processed. Returns from all other functions are discarded.

1. `BeforeIteration(IterationNumber, ContainerList, State)`: This method is called before each iteration. Ideally used to initialize the state dictionary, open editors as

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

required, or initialize setups as needed for each iteration.

2. `BeforeIterationEx(IterationNumber, ContainerList, State)`: Similar to the `BeforeIteration` method except that this allows you to control the number of iterations via the return value.
    - Return: `"more"` to request one more iteration
    - Return `"last"` to indicate that this is the last iteration
    - Any other return (including none) will be treated as a return of `"last"` and terminate iterations.
3. `AfterIteration(IterationNumber, ContainerList, State)`: This method is called after each iteration. This can be used to copy result files over, check results, implement any possible convergence calculations, logging of results, *etc*.
4. `BeforeContainerRefresh(IterationNumber, Container, ContainerList, State)`: Is called before each of the coupled containers is refreshed. The `Container` argument represents the container about to be refreshed.
5. `AfterContainerRefresh(IterationNumber, Container, ContainerList, State)`: Is called after each of the coupled containers is refreshed. The `Container` argument represents the container just refreshed.
6. `BeforeContainerUpdate(IterationNumber, Container, ContainerList, State)`: Is called before each of the coupled containers is Update (after a refresh). The `Container` argument represents the container about to be updated.
7. `AfterContainerUpdate(IterationNumber, Container, ContainerList, State)`: Is called after each of the coupled containers is updated. The `Container` argument represents the container just updated.

If the callback scripts uses other files to send commands to various containers (vb, js, apdl, python, etc.), all of those files are best saved under the user_files directory. This allows you to use the `FBGetUserFilePath(str)` to get the absolute path of the file and allows the files to be packaged with any created archive.

Utility Functions

`FBSystemForContainer` (container)	system	Given a container, returns the system it belongs to
`FBSystemDisplayName` (system)	string	Given a system, returns its display name on the schematic
`FBSystemID`(system)	string	Given a system, returns its ID. This is the same as the `UniqueDIrectory` for the system
`FBGetUserFilesPath` (relativePath)		Given a relative path located under the user_files directory, say "hello.py", this returns the absolute path of the file

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output/Debugging Functions

`FBAddInfoMessage (string)`	Adds an info message to the WB message window
`FBAddWarningMessage (string)`	Adds a warning message to the WB message window
`FBAddErrorMessage (string)`	Adds an error message to the WB message window
`FBMessageBox (string)`	Pops up a dialog box with the supplied string with an **OK** button

Example Scenarios for Feedback Iterator

This section describes several scenarios for using the Feedback Iterator:

Setting up Iteration with Feedback Iterator

Breaking Iteration Control

Starting an Iterative Update

Running a Single Iteration

Interrupting an Iterative Loop

Resuming an Interrupted Iterative Loop

Modifying any of the Systems Involved in Iteration (coupled clients)

Iterating to Convergence

Setting up Iteration with Feedback Iterator

Scenario	Steps to execute	Outcome
Electromagnetics product solution provides solution data to a single downstream setup	• Drop Feedback Iterator system on the setup component of Electromagnetics product OR • Create Feedback iterator system and connect	Electromagnetics product setup component is connected to the Iterator. Electromagnetics product solution and downstream setup and solution components are coupled as clients to the iterator. The iterator component is the coupled master and the rest are coupled clients. Any coupling changes schematic visuals: • Coupled cells including coupling master are colored differently from normal cells

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scenario	Steps to execute	Outcome
	Electromagnetics product setup component to the TwoWay iterator component AND User clicks on the Iterator component and sets the desired Feedback Iterator properties.	• Coupled cell icons change to reflect the icon of the coupling master • Coupled client cells no longer display the **Update** context menu item and cannot be updated via script commands • The cell states of all coupled cells (master and clients) are synchronized to be the most pessimistic state among any of them (e.g., after an update, any modification will set all of them to the "Modified" state).
Electromagnetics product solution provides to multiple downstream setup cells	same as above	All the downstream components (setup and solution in each downstream system) are coupled as clients. The update order in this case depends on any additional data-flow connections between the systems that consume the upstream Electromagnetics product solution data.
Electromagnetics product solution cell does not have any downstream targets	same as above	No changes occur beyond what is expected when you create a new connection (no coupling). Coupling only occurs when Electromagnetics product solution cell has a downstream connection. If a downstream connection is added after the link to the iterator, coupling will be performed as described in the first row.

Breaking Iteration Control

Scenario	Steps to execute	Outcome
Iterator coupled to Electromagnetics product solution cell and downstream setup/solution cells	• Break Electromagnetics product solution cell's provides link OR • Break Electromagnetics product setup link	In addition to what is normally expected from the user actions, all coupled clients are de-coupled (their icons and colors on UI are restored)

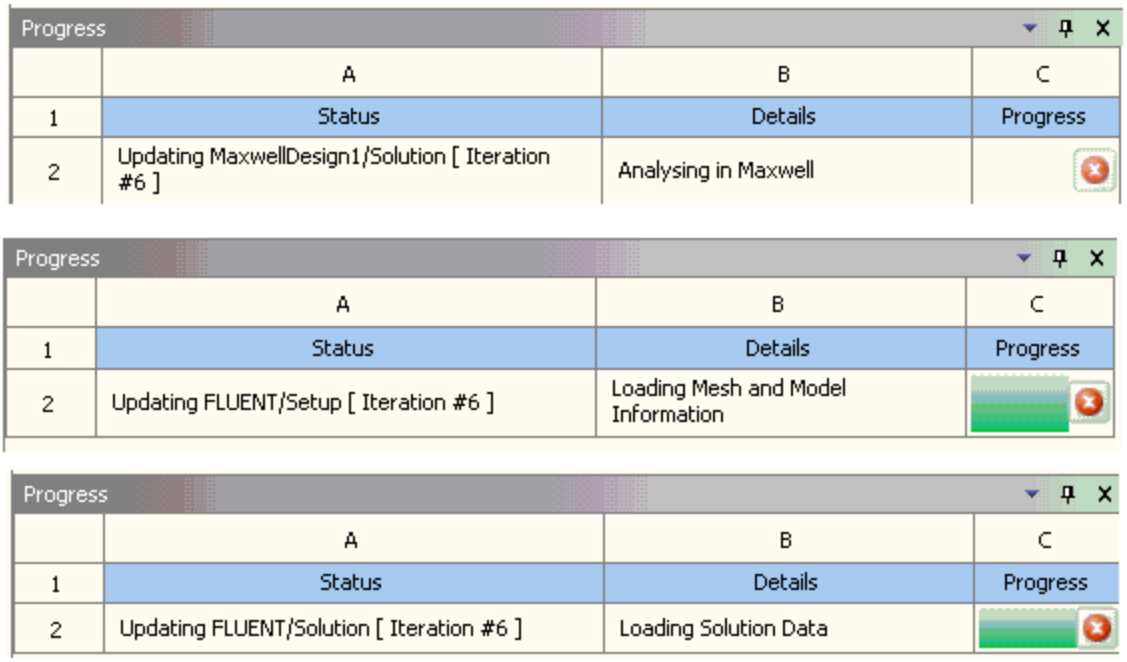
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scenario	Steps to execute	Outcome
	to iterator OR • Delete Electromagnetics product solution consumer system OR • Delete Iterator system	
Electromagnetics product setup cell is connected to iterator but Electromagnetics product solution does not provide anything. No components are coupled to the iterator	• Break Electromagnetics product setup link to iterator • Delete Iterator system	No coupling exists in this scenario so nothing visible changes beyond what is expected from the user actions

Starting an Iterative Update

Scenario	Steps to execute	Outcome
Standard update scenario	• RMB on iterator component and select Update GUI operation OR • Update Project OR • Update all design points OR • DX Update	For each iteration, the Electromagnetics product solution is updated followed by the downstream setup and the downstream solution. Each of the client component updates display their progress monitor and can be aborted (resulting in aborting the current iteration)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Running a Single Iteration

Scenario	Steps to execute	Outcome
NumCompleted < NumIterations	RMB on iterator component and select Single Iteration Update	One iteration is run. "Completed Iterations" property is incremented.
NumCompleted ≥ NumIterations	same as above	same as above

Interrupting an Iterative Loop

Currently, there is no special progress or interruption control. The progress monitor for individual components displays as they are updated and any control they choose to provide (interrupt, abort or both) is available. Choosing to abort any of the client component updates aborts the current iteration, and the Completed Iterations property remains unchanged from the previous iteration.

Resuming an Interrupted Iterative Loop

Scenario	Steps to execute	Outcome
Single step	RMB on iterator component and select Single Iteration Update	One iteration is always run and the Completed iterations property is incremented.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scenario	Steps to execute	Outcome
Run until completion	• RMB on iterator component and select Update OR • Select the Project Update menu option from the toolbar, etc.	If the user specified number of operations are already completed, nothing is done. Otherwise, the remaining iterations are run.

Modifying any of the Systems Involved in Iteration (Coupled Clients)

When any coupled client components are modified (either in Workbench or in an external editor), the coupled clients and master are marked as modified. The Completed Iterations property is set to 0.

Maxwell in a Thermal Deformation Use Case

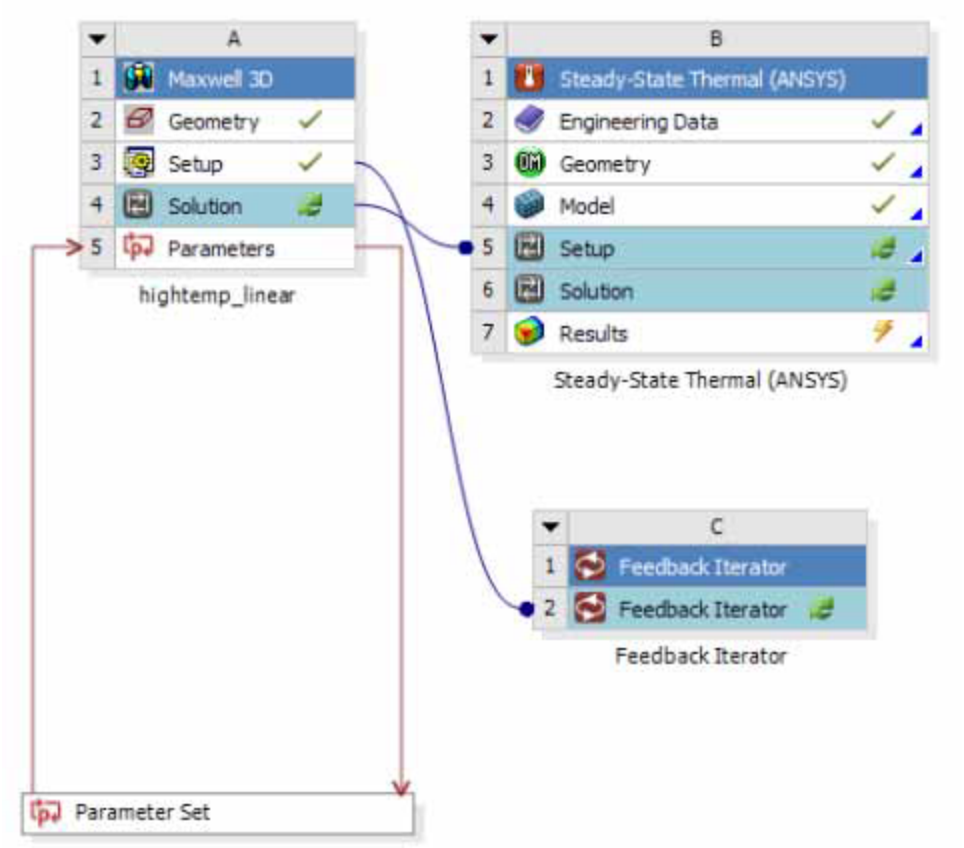
This is a more complex and involved use case. The feedback iterator works identically.

Iterating to Convergence

Scenario	Steps to execute	Outcome
Standard update scenario	• Right-click on the Feedback Iterator component and select Update OR • Update Project OR • Update all design points OR • DX Update	For each iteration, the Electromagnetics product solution is updated, followed by the downstream setup and the downstream solution. Each of the client component updates display their progress monitor and can be aborted (resulting in aborting the current iteration)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

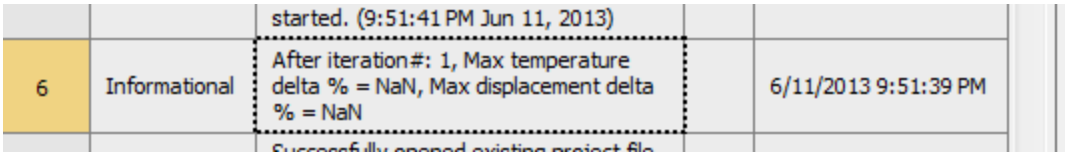
The following temperature convergence example uses Feedback Iterator with a Maxwell design, coupled with a Steady-State Thermal component. Iterating to convergence operates similarly for HFSS projects.



The example uses defaults of 5% for the convergence targets and a value of 100 max iterations. Recall that the max iterations value is set as a safety measure to ensure that iterations do not continue indefinitely if the solution does not converge. As we progress through the iterations, we observe the following:

End of Iteration #1

The first iteration, because it has no previous iteration, cannot return a meaningful delta value.



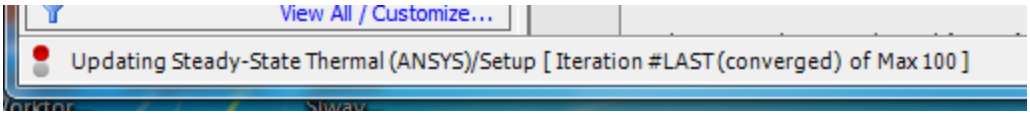
At the end of this iteration, the Latest Delta value is listed as Not Available.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

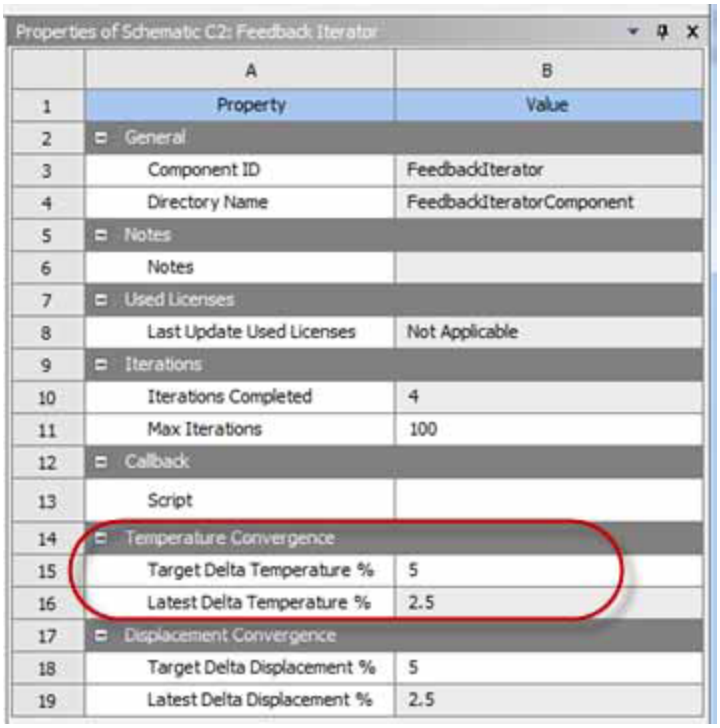
	A	B
1	Property	Value
2	General	
3	Component ID	FeedbackIterator
4	Directory Name	FeedbackIteratorComponent
5	Notes	
6	Notes	
7	Used Licenses	
8	Last Update Used Licenses	
9	Iterations	
10	Iterations Completed	1
11	Max Iterations	100
12	Callback	
13	Script	
14	Temperature Convergence	
15	Target Delta Temperature %	5
16	Latest Delta Temperature %	Not Available
17	Displacement Convergence	
18	Target Delta Displacement %	5
19	Latest Delta Displacement %	Not Available

Convergence Achieved

In this example, convergence has been achieved by the third iteration.

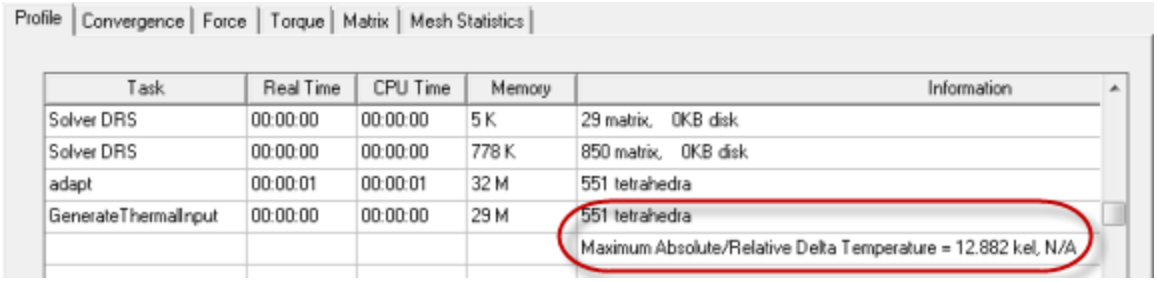


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When convergence criteria are met, the simulation stops. If you attempt to update after convergence has been achieved, the simulation will not be launched since the current solutions satisfy the convergence criteria.

You can monitor the iteration progress by opening the Maxwell editor and selecting **Results** > **Solution data**. Choose the **Profile** tab and keep the window open. As you solve each iteration, you can observe the reported Delta-T (with thermal feedback) and/or delta-displacement (with displacement feedback) and abort the iterations (or stop single iteration updates) when the values reach acceptable convergence levels.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20 - Scripting

Information about creating, editing, and using IronPython scripts is in the product's scripting guide in the help. You can also access PDF versions of the scripting guide from the help system.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

21 - Exporting Equivalent Circuit Data

You can export lumped R, L, C data from a Maxwell solution to Ansys Twin Builder. Importing the new data file to Twin Builder enables you to include wave effects in the circuit simulations.

> **Note:** An equivalent circuit can be exported from a parametric solution or from an imported table.
>
> You can only export an equivalent circuit from a parametric solution when the following two criteria are met:
>
> - The solution type is **Magnetostatic** or **Electrostatic**.
> - A parametric setup exists.

Exporting a Circuit From a Parametric Solution

To create a circuit from parametric solutions, you first need to set up and solve a problem that contains a parametric sweep. Any force, torque, and matrix solutions are automatically available to use in the equivalent circuit. Other solutions can be treated as extra inputs/outputs if they have been added as calculations in the parametric setup — that is, extra inputs/outputs come from the parametric table.

> **Note:** In Maxwell, global variable names begin with the **$** character. However, when you are creating an ECE **.sml** model, this character causes the circuit to fail when importing it to Twin Builder. To resolve this, when the circuit is exported, the **$** character is replaced by the **_** character. For example, the project variable **$MyVar** is converted to **_MyVar**.

To export a circuit from a parametric solution:

1. Click **Maxwell 2D** or **Maxwell 3D** > **Export Equivalent Circuit** > **From Parametric Solutions**.
   The **General** window appears, allowing you to specify basic information about the circuit model.

   > **Note:** This command is enabled only when the following two criteria are met:
   >
   > - The solution type is **Magnetostatic** or **Electrostatic**.
   > - A parametric setup exists.

2. Select one of the following from the **Model Type** drop-down menu depending upon the Design Type (2D or 3D):

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Linear Motion**
- **Rotational Motion**
- **Transformer**
- **Matrix**
- **Lookup Table**

3. Select a **Parametric Setup** from the drop-down menu.
4. Select a **Solution Setup** from the drop-down menu.
5. Select a **Matrix Setup** from the drop-down menu.
6. Select a **Force Setup** or a **Torque Setup** from the drop-down menu, depending on which parameter you have set up for your design.
7. For the force or torque setup, select **X**, **Y**, or **Z** as the **Component**.
8. Select either **Ampere-Turns** or **Amperes** as the **Current Source Variables** option.
9. Click **Next**.

   The **Table** window appears, allowing you to set up the inputs and outputs for the circuit equivalent. Most of the values have been automatically filled in, based on your design. You can keep the defaults or change the selections and values for the various parameters.

10. To apply deep spline interpolation to all inputs in the circuit PWL model, select the **Use Bezier Interpolation** check box.
11. To export the data to a table that you can edit and use to export an equivalent circuit at a later time, do the following:
    a. Click **Export Table**. The **Save As** dialog box appears.
    b. Select a location, and type a name in the **File name** box.
    c. Click **Save**.

    The **Table** window reappears. The file that is created contains header rows with information such as name, i/o, and type, plus all of the actual data. You may export the table to view or use in external programs. You can also modify the exported table file and then re-import it to create an equivalent circuit. On import, the information in the header rows is used as default settings in the circuit export dialog box.

    > **Note:** The **Table** window is the last step for the **Lookup Table** model type. For all other model types, there is one more step: the **Terminals** window.

12. For model types **Linear Motion**, **Rotational Motion**, **Transformer** and **Matrix**, click **Next**.

    The **Terminals** window appears, allowing you to set up terminals, which are nodes with "through" and "across" data. Most of the values have been automatically filled in, based on your design. You can keep the defaults or change the selections and values for the various parameters.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13. Enter a **Scaling Factor** in the text box. The scaling factor is applied to all output quantities and can be used, for example, to scale data from partial models that take advantage of symmetry.
14. Optionally, specify the **Model depth** for 2D XY models for scaling.
15. Optionally, for 2D XY **Rotational Motion** model type, specify the **Skew** angle.

    When a stator core or rotor core is skewed for a 2D Rotational Motion model, the flux linkages and torque of the look-up table will be modified based on the skew angle θ_k .

    For any output of the look-up table, f(θ), where θ is rotor position, the modified output will be

$$f_m(\theta) = \frac{1}{\theta_k} \int_{\theta-\theta_k/2}^{\theta+\theta_k/2} f(\theta)\delta\theta$$

16. Specify the **Terminals**:

    To specify a coil terminal:

    a. Select a **Source** for the **Flux** or **Charge**, depending on whether your design is mechanical or electrical.
    b. Enter a **Resistance**.
    c. Enter the number of **Turns** in the coil winding.
    d. Enter the number of **Branches**.

    To specify a mechanical terminal:

    a. Select the **Force** (or **Torque)** variable from the drop-down menu.
    b. Select the **Position** variable from the drop-down menu.
    c. Optionally, for Rotational Motion model type, select **Use rotational velocity**.
17. Click **Finish** to export the equivalent circuit.

Types of Equivalent Circuit Models

Currently, you can generate the following model types:

- **Linear Motion** - Models inductance (or capacitance) and force as functions of current (or voltage) and position. The parametric solution should cover the range of motion and conductor current (or voltage).
- **Rotational Motion** - Models using rotational rotor position sweep.
- **Transformer** - For one coil only, models nonlinear mutual (magnetizing) inductance with a linear leakage inductance branch. Therefore, this model is only valid for 1-phase transformers. The parametric solution should cover the range of current for one coil only in an open-circuit test. Nonlinear mutual inductances for 3-phase (or more) transformers are better solved using the transient solver.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Matrix** - Models an inductance or capacitance matrix for one or more conductors, as a function of current or voltage. The parametric solution should cover the range of each conductor's current or voltage.
- **Lookup Table** - This option creates a lookup table model in Simplorer format.

Exporting a Circuit From an Imported Table

To create a circuit from an imported table, you need to have a table file containing tabular data. The first row in the file must contain the column names. The table export feature writes other useful information into subsequent rows, but the import succeeds whether or not this information is present. The rows of tabular data come next.

> **Note:** In Maxwell, global variable names begin with the **$** character. However, when you are creating an ECE **.sml** model, this character causes the circuit to fail when importing it to Twin Builder. To resolve this, when the circuit is exported, the **$** character is replaced by the **_** character. For example, the project variable **$MyVar** is converted to **_MyVar**.

To export a circuit from an imported table:

1. Click **Maxwell 3D or Maxwell 2D** > **Export Equivalent Circuit** > **From Imported Table**.

   The **Select File** dialog box appears.

2. Find and select the file containing the table you want to use.
3. Click **Open**.

   The **General** window appears, allowing you to specify basic information about the circuit model.

4. Select one of the following from the **Model Type** drop-down menu:
   - **Linear Motion**
   - **Rotational Motion**
   - **Matrix**
   - **Lookup Table**
5. Select either **Ampere-Turns** or **Amperes** as the **Current Source Variables** option.
6. Click **Next**.

   The **Table** window appears, allowing you to set up the inputs and outputs for the circuit equivalent. Most of the values have been automatically filled in, based on your design. You can keep the defaults or change the selections and values for the various parameters.

7. To apply deep spline interpolation to all inputs in the circuit PWL model, select the **Use Bezier Interpolation** check box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. To export the data to a table that you can edit and use to export an equivalent circuit at a later time, do the following:
    a. Click **Export Table**. The **Save As** dialog box appears.
    b. Select a location, and type a name in the **File name** box.
    c. Click **Save**.

    The **Table** window reappears. The file that is created contains header rows with information such as name, i/o, and type, plus all of the actual data. You may export the table to view or use in external programs. You can also modify the exported table file and then re-import it to create an equivalent circuit. On import, the information in the header rows is used as default settings in the circuit export dialog box.

    > **Note:** The **Table** window is the last step for the **Lookup Table** model type. For all other model types, there is one more step: the **Terminals** window.

9. For model types **Linear Motion**, **Rotational Motion**, and **Matrix**, click **Next**.

    The **Terminals** window appears, allowing you to set up terminals, which are nodes with "through" and "across" data. Most of the values have been automatically filled in, based on your design. You can keep the defaults or change the selections and values for the various parameters.

10. Enter a **Scaling Factor** in the text box. The scaling factor is applied to all output quantities and can be used, for example, to scale data from partial models that take advantage of symmetry.
11. Optionally, specify the **Model depth** for 2D XY models for scaling.
12. Optionally for 2D XY **Rotational Motion** model type, specify the **Skew** angle.

    When a stator core or rotor core is skewed for a 2D Rotational Motion model, the flux linkages and torque of the look-up table will be modified based on the skew angle θ_k.

    For any output of the look-up table, f(θ), where θ is rotor position, the modified output will be

$$f_m(\theta) = \frac{1}{\theta_k} \int_{\theta-\theta_k/2}^{\theta+\theta_k/2} f(\theta)\delta\theta$$

13. Specify the **Terminals**:
    - To specify a coil terminal:
        a. Select a **Source** for the **Flux** or **Charge**, depending on whether your design is mechanical or electrical.
        b. Enter a **Resistance**.
        c. Enter the number of **Turns** in the coil winding.
        d. Enter the number of **Branches**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To specify a mechanical terminal:
  a. Select the **Force** (or **Torque)** variable from the drop-down menu.
  b. Select the **Position** variable from the drop-down menu.
  c. Optionally for Rotational Motion model type, select **Use rotational velocity**.

14. Click **Finish** to export the equivalent circuit.

Setting Up Current Variables

For problems with current, the **Current Variables Represent** setting allows you to specify one of the following two options:

- **Ampere-Turns**
- **Amperes**

This setting provides flexibility in setting up current sources with different numbers of turns and branches.

In Maxwell, current source values are specified in Ampere-turns, meaning that if a given source is driven with 2 Amps through 100 turns, then the source value must be set to 200. Inductance is calculated per-turn; therefore, to calculate EMF, the flux must be scaled by the number of turns to give the total flux.

You can create two kinds of circuit models:

- A model format where the data table is based on current in Amp-turns and flux per turn. This model internally converts currents and fluxes using the specified turn ratio. This model format was the only one available in versions of Maxwell before Maxwell 11.
- A model format where the data table is based on current in Amps and total flux. This model does *no* internal scaling of values, since the data is already in a form the outside circuit expects. This model format is available in Maxwell 11 and later versions.

For either model type, you can view the inductance setup to obtain information about the specified groups, turns, and branches.

Current Variables Represent Ampere-Turns

How to use this format:

1. Apply variables directly to sources. *Source1* value = *Current1*.
2. Sweep variable values in Ampere-turns. *Current1* sweeps from 100 to 200.
3. In an inductance setup, specify the number of turns consistent with the source setup. Make sure all sources in a group have the same number of turns.

When to use this format:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If all sources in a group have the same number of turns.
- If you want to be able to modify the number of turns in the exported circuit.
- If you are more comfortable thinking of variables in Ampere-turns.

When using this format, the grouping is taken from the inductance setup. The number of turns and branches per group from the inductance setup are used as a default but can be modified either before exporting the circuit or in the exported circuit.

Current Variables Represent Amperes

How to use this format:

1. Define sources as the current variable times turns. *Source1* value = *100*Current1*.
2. Sweep variable values in Amperes. *Current1* sweeps from 1 to 2.
3. In an inductance setup, specify the number of turns consistent with the source setup.

When to use this format:

- In cases where one current is applied through sources with different numbers of turns.
- If you do *NOT* need to modify the number of turns in the exported circuit.
- If you are more comfortable thinking of variables in Amperes.

When using this format, the groups, turns, and branches are taken from the inductance setup and are used as is. You will *not* be able to modify them during circuit export or in the exported circuit.

Setting Inputs and Outputs in the Table Window

The table includes rows for all of the following that are appropriate to the model type:

- Swept variables from the parametric setup.
- Components of flux or charge from the selected matrix.
- Force or torque.
- Calculations from the parametric setup.

Where possible, the columns are set with reasonable defaults based on the information from the design. In many cases, you should not have to change anything.

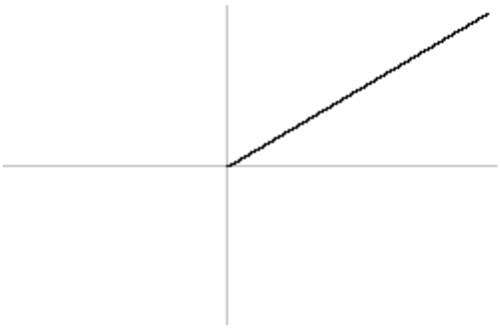
Additional information about the columns:

- Under **I/O**, the choices are **Input**, **Output**, or **Unused**. **Unused** means that the quantity will not be included in the circuit.
- Under **Type**, the choices are **Position**, **Rotation**, **Current**, **Voltage**, **Flux**, **Charge**, **Force**, **Torque**, and **Other**. These choices affect the terminal setup in the next window. **Other** is typically used for extra inputs/outputs.

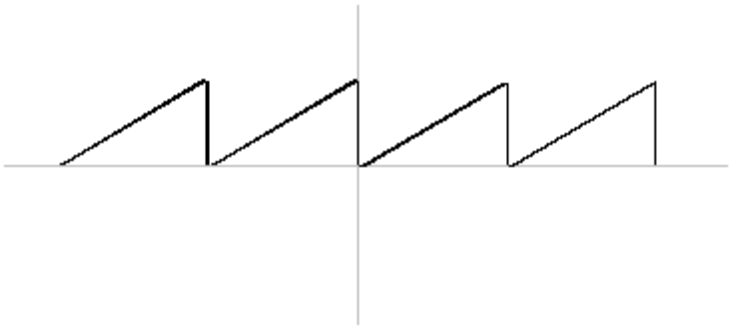
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Under **Extrapolate**, the choices are all standard extrapolations plus **None**. For inputs, the **Extrapolate** column is automatically set to **None** and is disabled.

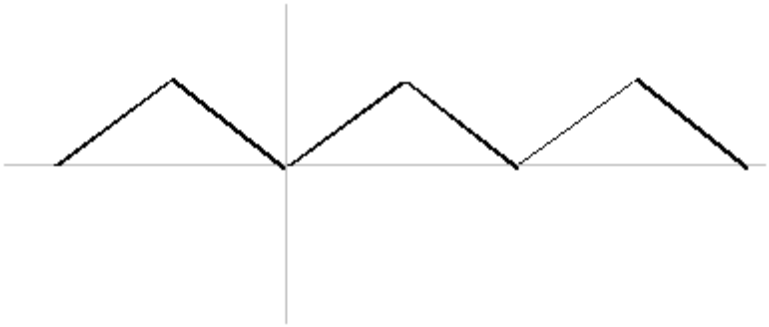
Linear This option takes the last two points of the data and generates a straight line extending beyond the range of the parametric sweep.



Periodic This option repeats the data outside the range of the parametric sweep.

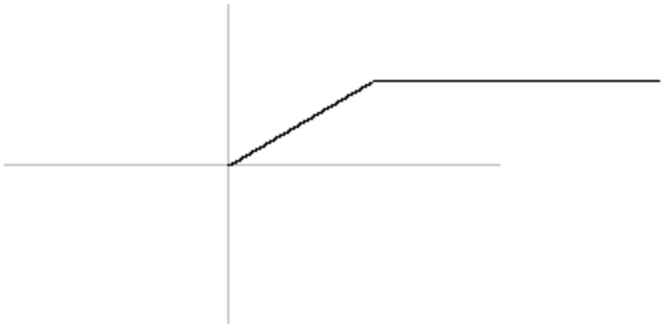


HalfPeriod This option mirrors then repeats the waveform outside the range of parameter sweep.

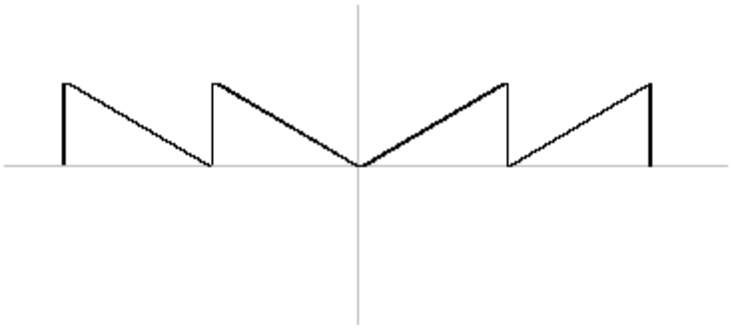


Constant This option extrapolates a constant value from the last point in the interpolation.

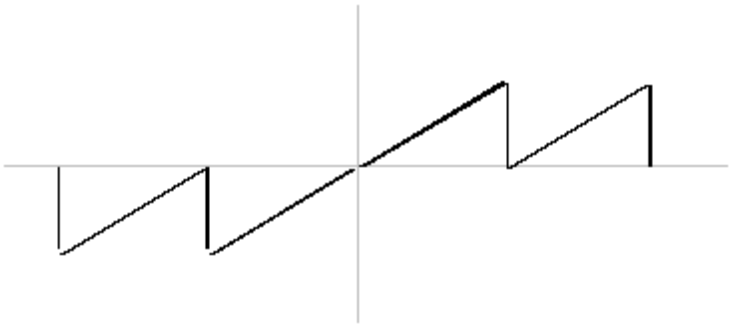
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Even This option repeats the wave outside the range of parametric sweep.



Odd This option repeats a reflection of the waveform outside the range of the parametric sweep.



Setting Coil and Mechanical Terminals

There are currently two types of terminals:

- Coil Terminals
- Mechanical Terminals

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For models with motion, the mechanical terminal setup determines which force (or torque) causes the motion and which position (or rotation) is affected.

When setting up coil terminals, flux and current are used for magnetic models, and charge and voltage are used for electric models.

In the coil terminal setup, there is one row for each flux (or charge) group. In the **Current** (or **Voltage**) column, each cell is a selection box where you can choose the current (or voltage) that applies to that flux (or charge) group. Usually the software can choose the correct current by default, based on the source setup in the design. The **Resistance** column is set to **0** by default; you can enter any value here. The **Turns** and **Branches** columns only appear for magnetic models where current variables represent Ampere-turns. These columns are initialized with the turns and branches specified in the inductance matrix setup.

In the **Current** column, in addition to each current, there is another choice, **<Dependent>**. This feature allows you to solve some problems using fewer parametric rows. This is explained using the following example:

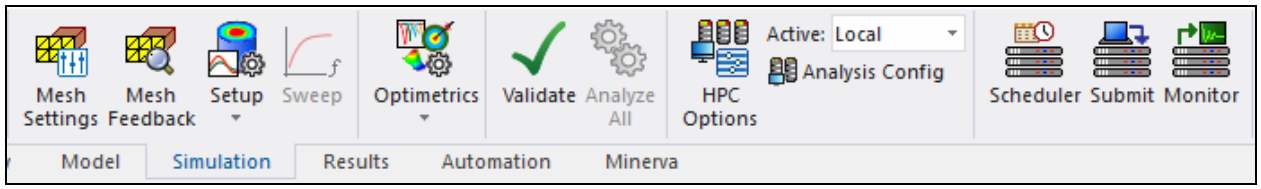
> Suppose you are working on a three-phase machine. You create three sources: CurrentA, CurrentB, and CurrentC. In Maxwell, you can create only variables iA and iB and then set the value of CurrentC to -(iA + iB). The parametric table will have ten values each for iA and iB, so it will only have 100 rows. When exporting the circuit, you would set the current for Flux [CurrentA] to iA, Flux[CurrentB] to iB, and Flux[CurrentC] to <Dependent>. In the circuit model, the PWL table will contain two input currents (iA and iB), but it will contain all three fluxes. So it will look up all three flux values based on only the two current values. As long as you always connect a current of -(iA + iB) to CurrentC, this model will be valid.

Note: A dependent source can have a different number of turns from the sources that it depends on (in the example above, CurrentA, CurrentB, and CurrentC can all have different numbers of turns). But in this case, you would have set up the current variables as Amperes instead of Ampere-turns — otherwise there would be no way to obtain the correct scaling of currents iA and iB for both their own current sources and for CurrentC. Therefore, this is another case where specifying current variables in Amperes (rather than in Ampere-turns) is useful.

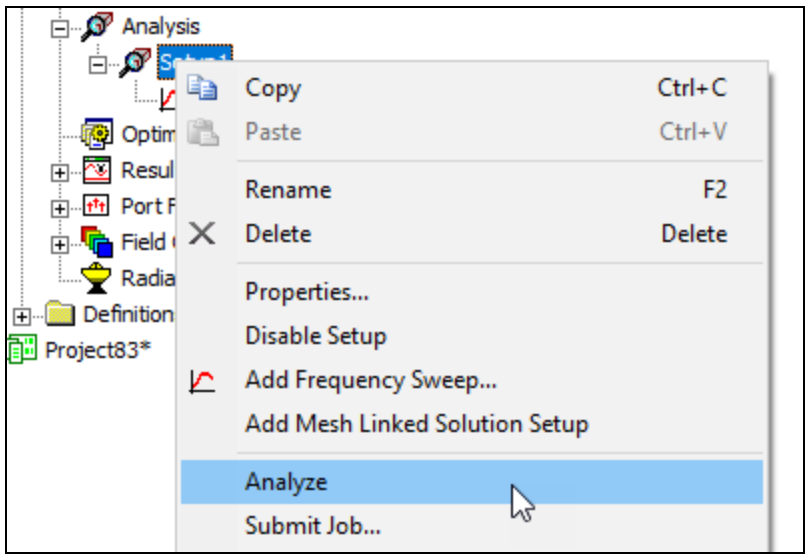
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22 - Running Simulations

After you specify how Ansys Electronics Desktop is to compute the solution, you need to begin the solution process. You can access the **Analyze** commands for a specific setup by right-clicking on a solution or sweep in the **Project** tree and selecting from the shortcut menu, you by selecting the **Analyze All** from the solver menu, or the **Analyze All** icon on the **Simulation** tab of the ribbon.

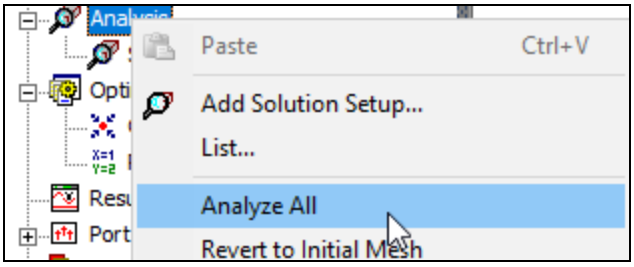


In general, the **Analyze** command on the shortcut menu applies to the selected setup and associated sweeps, if any, or to a selected sweep. To use this command, right-click on a setup or sweep in the Project Manager and click the command on the context menu.



The **Analyze All** command applies to allenabled setups, , and sweeps at or below the level invoked in the Project Manager. To use this command, either click **Maxwell**> **Analyze All** or right-click the **Analysis** icon in the *Project Manager* and select **Analyze All**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



What do you want to do?

- Local Analysis
- Solve a single setup with or without sweeps (when applicable)
- Solve a specific sweep
- Enable a queue so that multiple simulations can run sequentially as resources become available.
- Run more than one simulation, whether multiple setups, or multiple sweeps under a single setup, or setups with dependencies.
- Enable or Disable one or more solution setups or sweeps.
- Submit Job to RSM or HPC Scheduler
- Monitor queued simulations
- Monitor the solution process
- Change a solution priority for system resources
- Abort an analysis
- Re-solve after modifying a design
- Re-solve after Ansys Workbench Feedback

Local Analysis

Ansoft RSM Service does not need to be installed or running for local analysis and should not be running if the user only will be running local analysis. If running a distributed analysis, consult Distributed Analysis.

> **Note:**
>
> MPI distribution may be affected by starting or stopping a VPN, even if the MPI distribution is limited to the local host.

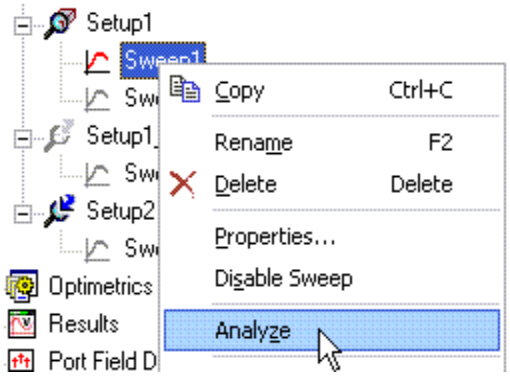
To run more than one analysis at a time, follow the same procedure while a simulation is running. If you have enabled queuing, the next solution setup will be solved when the previous solution is complete.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solving a Single Setup or Sweep

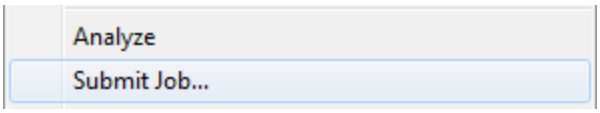
To run a single setup or sweep:

1. Select a solution setup or sweep in the Project Manager.
2. Right-click and select **Analyze** from the shortcut menu. The graphic shows a single sweep selected for analysis.



The 3D field solution is computed inside the structure for a solution. For a select sweep, it is computed for the sweep variables.

When you right-click a Setup, rather than a Sweep, the shortcut menu also includes the **Submit Job...** command.



For more information on the **Submit Job...** command see, Distributed Analysis and High Performance Computing (HPC) Integration.

To run more than one analysis at a time, follow the same procedure while a simulation is running. If you have enabled queuing, the next solution setup will be solved when the previous solution is complete.

> **Note:**
>
> If a linked dependency in the setup is already simulating (for example, due to setup links to the same external source for a near or far field wave, or a magnetic bias), Ansys Electronics Desktop will not allow another dependent simulation to start until the first use of the source has completed.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

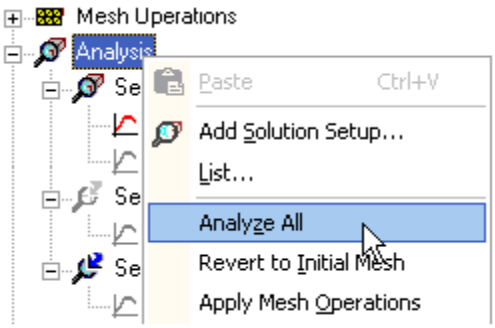
Running More than One Simulation

To solve every enabled solution setup in a design:

1. In the project tree, under the design you want to solve, select **Analysis**.
2. Click **Maxwell**> **Analyze All**.

Each enabled solution setup is solved in the order it appears in the Project Manager.

The example here shows an analysis invoked from the Project Manager shortcut menu with three setups: one disabled, two enabled. The first setup has one sweep enabled, and one disabled (shaded icon). The second setup is disabled, and the third is enabled, with a disabled sweep.



Note:

The **General** tab for the Setup includes an **Enabled** check box. By default, this is checked. Clearing the **Enabled** check box excludes a setup from running

Note:

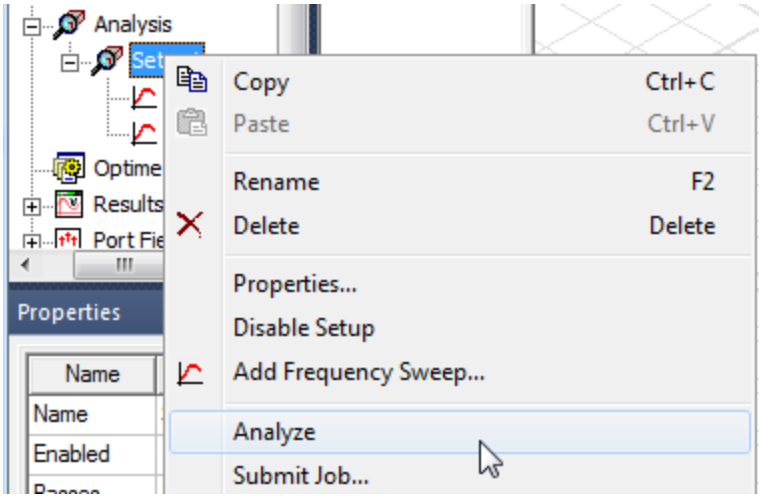
You can simulate all designs in a project by clicking **Project** > **Analyze All**.

To solve *two or more sweeps or two or more parametric analyses under a setup*:

1. In the project tree, under the design you want to solve, right-click **the setup** icon that includes the sweeps of interest.
2. Click **Analyze** on the shortcut menu.

   Each solution sweep under that setup is solved in the order it appears in the Project Manager, using the available machines. The following example shows a setup with two enabled sweeps.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Monitoring Queued Simulations

If you have multiple setups for a design and have selected **Analyze All**, the simulations can be queued until there is a machine available. You enable queuing in the HPC and Analysis Options: Options tab. If queuing is enabled, and you run multiple setups, they are solved in the order that they appear in the Project Manager. You can prioritize setups by changing their order in the queue.

1. To view the solution queue, click **Tools** > **Show Queued Simulations**.

   This displays a dialog box showing all the simulations and their current status. You can select and remove any simulation from the queue.

   You can also select any setup and use the **Move up** and **Move down** buttons to prioritize them.

2. To remove a simulation from the queue, select the simulation and click **Remove from Queue**.

   This removes the selected simulation from the queue.

Monitoring the Solution Process

While a simulation is running, you can monitor the solution's progress in the **Progress** window. The progress bar shows the relative progress of the simulation.

You can also view the following solution data at any time during or after the solution:

- The convergence data
- The matrices computed for the S-parameters, impedances, and propagation constants

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- A profile showing the status of the analysis process, including the progress and number of valid passes completed for adaptive solutions, elapsed time, memory usage, and more
- Mesh statistics (numbers of elements, lengths, volumes, and volume standard deviation; all on a per-object basis).
- A profile showing the status of the analysis process, elapsed time, memory usage, and more

To view the *Solutions* window:

1. Right-click the solution **Setup** in the Project Manager.

   A shortcut menu appears.

2. Select **Convergence**, **Solutions**, **MatrixData**, **Mesh Statistics**, or **Profile** from the shortcut menu.

   The *Solutions* window appears with the corresponding tab selected and the current data displayed.

For "out of core" problems, quite different amounts of memory may be used for factorization and for solution. So if the amount for factorization is displayed under the progress bar and the amount used is calculated for the profile at the end of the solution, they may be quite different numbers.

To view the solution status:

- Click **Maxwell** > **Results** > **Browse Solutions**.

  The *Solutions* dialog box appears with the **Browse** tab selected. It displays data about the number of valid passes completed (when adaptive solutions are applicable). It contains a tree structure showing the solutions listed according to Setup, Solution, and Variation. A table lists the setup, the solution, the sweep variable (when applicable), and the state of the solution.

- You can use the **Properties** button to display a dialog box that lets you change the way the Setup, Solution, and Variation are listed in the tree structure of the *Solutions* dialog box.
- You can delete one or more solutions by selecting from the table and clicking **Delete**. Click on a solution to select it, and use Ctrl+click to select multiple solutions, or Shift+click to select a range of solutions. You can also select all solutions using the **Select All** button.

  > **Note:**
  >
  > If a license is lost, the software waits for the license to be regained, checking every 2 minutes or until you abort.

- The **Statistics** tab of the *Solutions* dialog box displays path information as well as format, number of files, and size.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Changing a Solution Priority for System Resources

You can modify the priority of Ansys Electronics Desktop simulations so that system resources are allocated to other computer processes before the solver. If you reduce the priority of Ansys Electronics Desktop simulations, your other software tools will respond as they normally would, but Ansys Electronics Desktop simulations may take longer.

> **Note:**
>
> The Windows Task Manager does not indicate a reduced priority for the Ansys Electronics Desktop solvers. It only lists the priority of the engine manager, which appears normal, not the actual engine. The actual engine is in a separate thread, whose priority is not visible in the Windows Task Manager.

To change the priority of simulations for the system's resources:

1. While a solution is running, right-click the **Progress** window, and click **Change Priority** on the shortcut menu.
   - To affect priority for future simulation runs, click **Tools** > **Options** > **HPC and Analysis** and click the **Options** tab of the resulting dialog box.
2. From the **Change Priority** menu (or the **Default Process Priority** drop-down menu), select one of the following priorities:

Highest	
Above Normal	
Normal	Default
Below Normal	
Lowest Priority	

3. Click **OK**.

Related Topics

Monitoring Queued Simulations

Aborting an Analysis

To end the solution before the currently running process is complete:

- Right-click in the **Progress** window and click **Abort**.

  The solver ends the analysis immediately. The data for the currently running adaptive pass, frequency point, time step, or variation is lost. Previously completed solutions may

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

or may not be retained (depending on what process was interrupted by the Abort command). If you want to ensure that the results of all completed solutions are saved, use *Clean Stop* instead of *Abort*.

To abort the solution after the currently running process is complete:

- Right-click the **Progress** window and click **Clean Stop** on the shortcut menu.

  The analysis ends after the currently running adaptive pass, frequency point, time step, or variation has been solved. Solutions completed before the stop request are retained.

If you request a clean stop during the third adaptive pass, the solution for the third pass will be available once the third pass has finished solving, but the fourth pass will not run.

Ansys EM Application as an LSF Job

If you have an Ansys EM application running as an LSF job, you can use the command "bkill -s SIGTERM *jobid*" to terminate that application. Here *jobid* is the LSF job id. The response will be "Job <jobid> is being signaled." The response is the same whether the job is actually being signaled or not.

In cases where the SIGTERM parameter is ignored, the command kills the LSF job, but does not clean the lock files, and other files may not be in a consistent state. See http://www.vital-it.ch/support/LSF/programmer/advanced.html for a detailed description under *Signal Handling in Windows*.

Linux

For Linux, you can use TERM commands. Sigterm handling is done in Desktop library. You can abort a running batchsolve by sending a TERM signal to hfss.exe.

Re-solving after Modifying a Design

In some cases, if you modify a design after generating a solution, the solution in memory will no longer match the design. In such cases you receive a warning message that "Solutions have been invalidated. Undo to recover."

To generate a new solution after modifying a design, follow the procedure for running a simulation.

Also see Re-Solving with Ansys Workbench Thermal Feedback.

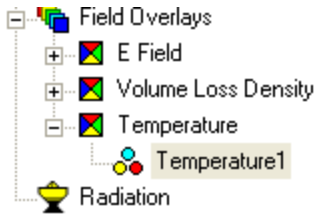
Re-solving after Ansys Workbench Thermal Feedback

With the Enable Feedback box in Setting the Temperature of Objects dialog box is checked, you can manage analysis with feedback in Ansys Workbench. After solving an HFSS or Q3D Extractor or Maxwell design, after performing the corresponding linked thermal analysis in Ansys

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Workbench, you can receive a temperature distribution back from the thermal solution. Ansys Workbench will write the feedback files directly to the Project Solution directory.

After an analysis that includes thermal feedback from Ansys Workbench, you can see temperature changes expressed in Temperature field overlays (both visually in the overlay and in the color key) as well as in the Solution data.



In the **Solution** data Profile tab you will see a new entry for Maximum Delta T, for the change in temperature from the previous simulation. The solver calculates delta in the first iteration by comparing the temperature distribution output from thermal with the initial temperature setting in HFSS/Maxwell/Q3D. Subsequent simulation iterations provide a number for the temperature delta.

Solver MCS4	00:00:00	00:00:00	31.4 M	Disk = 0 KBytes, matrix size 3468 , matrix bandwidth
Field Recovery	00:00:00	00:00:00	31.4 M	Disk = 310 KBytes, 2 excitations
				Maximum Delta T = 7.0685

This simulation feedback loop from Ansys Electronics Desktop to Ansys Workbench and back can continue until you decide that Temperature delta reported in the Solution Report low and stable for the designs.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

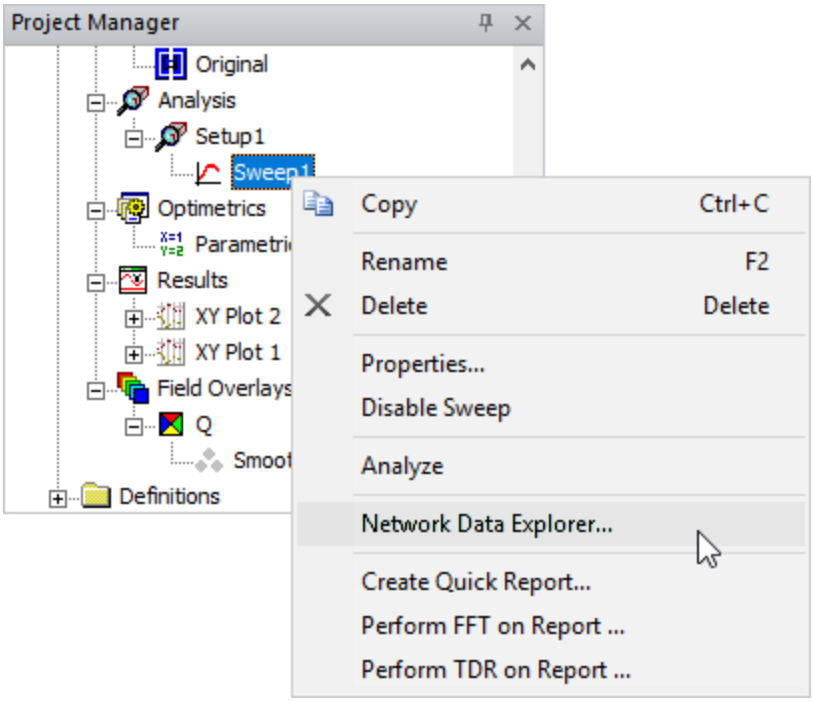
23 - Network Data Explorer

The Network Data Explorer provides visualization, analysis, and manipulation tools for network data.

To access Network Data Explorer, click **Tools** > **Network Data Explorer**.

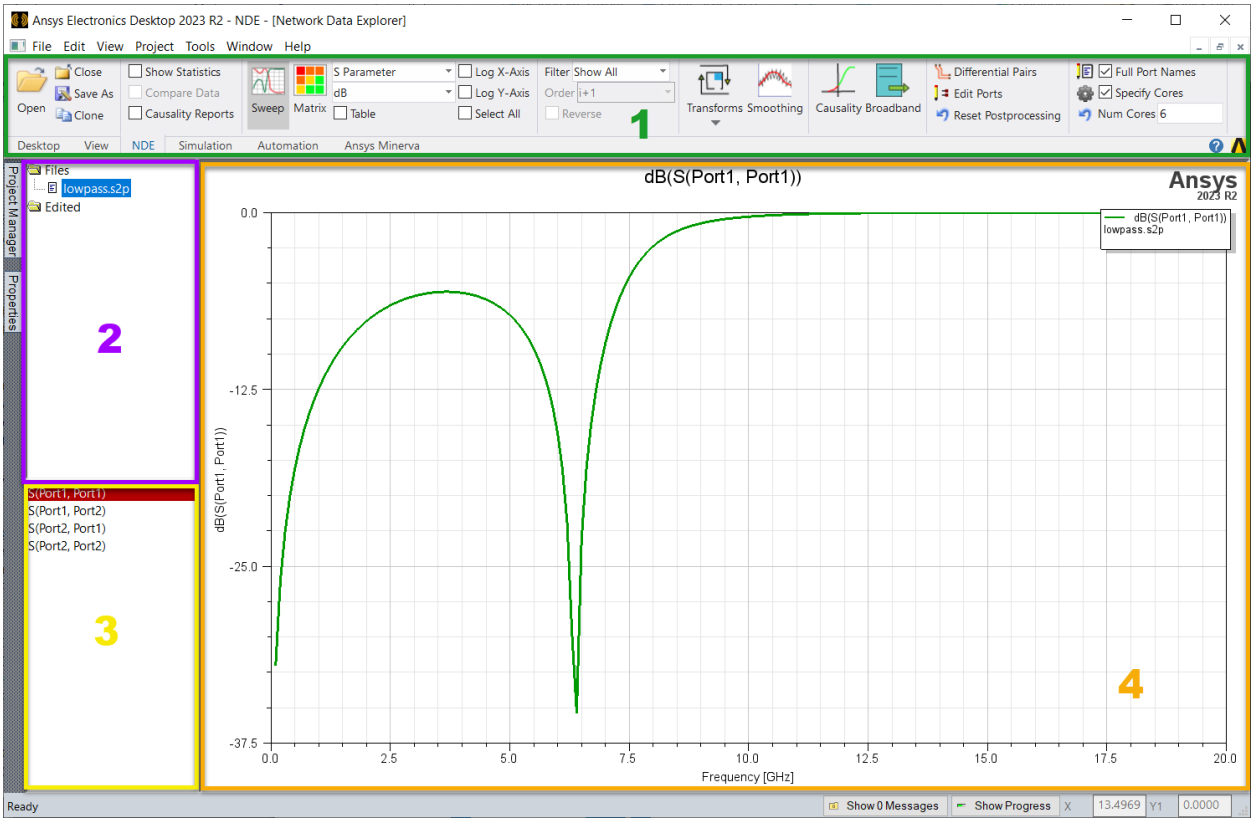
If you have run a simulation with data that can be converted into network data, you can also access Network Data Explorer through the shortcut menu in the Project Manager.

> **Note:**
>
> Accessing Network Data Explorer from this menu automatically loads sweep data.



The **Network Data Explorer** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



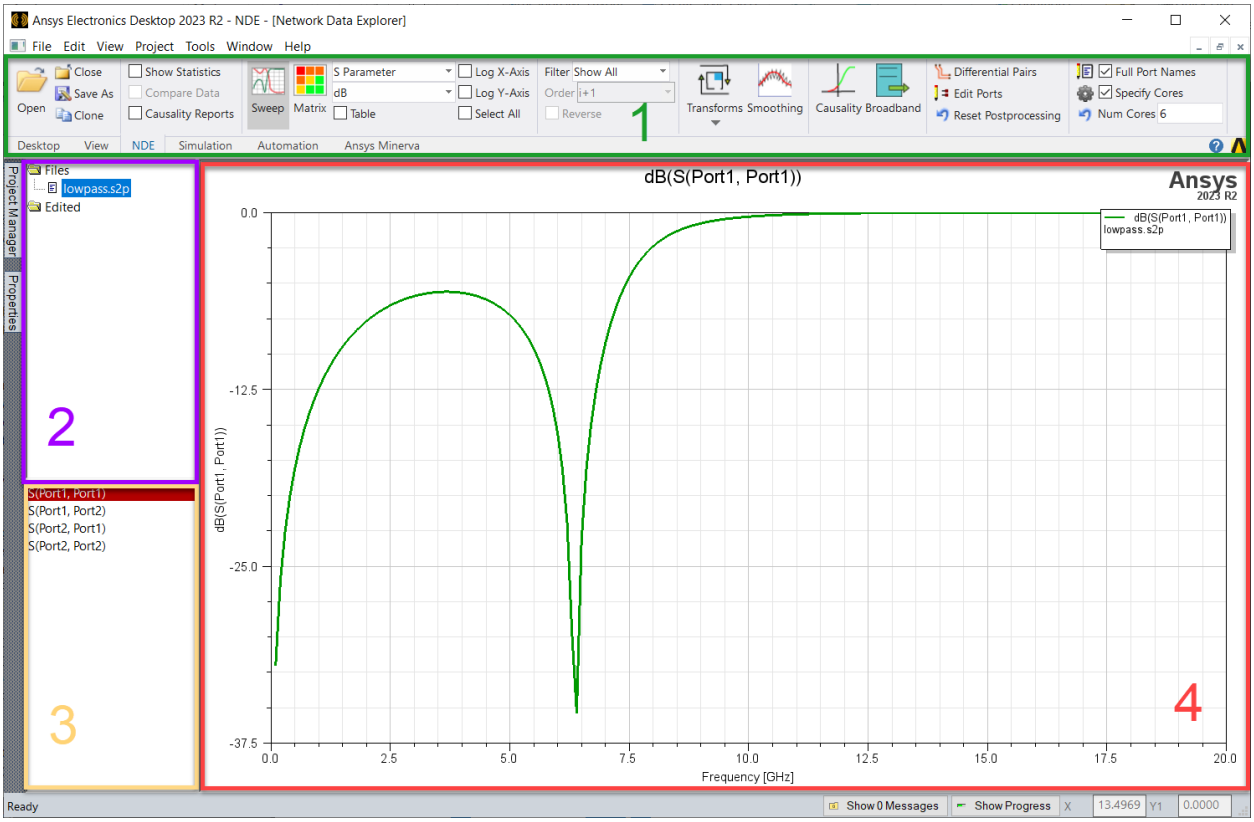
Network Data Explorer Overview

The Network Data Explorer window is divided into the following panes:

1. **NDE ribbon** – allows you to perform many functions of the Network Data Explorer.
2. **Network Data Selection pane** – allows you to select a network data file.
3. **Cell and Frequency Selection pane** – allows you to narrow your selection.
4. **Data View pane** – displays data in table or plot format.

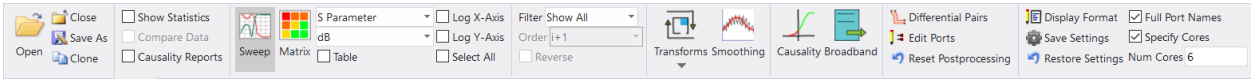
The panes are shown in the following figure. Additional information about each pane follows.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



NDE Ribbon

The ribbon provides access to many of the Network Data Explorer's functions and display options.



On this ribbon, you can control:

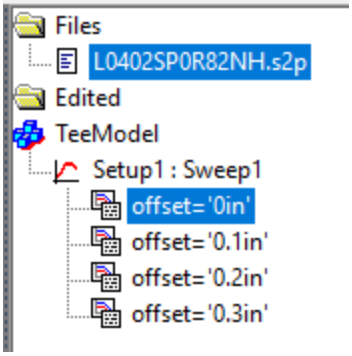
- Plotting - allows you to determine how the data is displayed.
- Quantity – allows you to select the type of quantity to display (parameter values, matrix statistics, or causality plots).
- Parameter Type – allows you to choose the parameter for display (S, Y, or Z parameters, Port Impedance, or Gamma).
- Format – allows you to decide the display function to apply to the data (e.g., magnitude, phase, dB, real, imaginary).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Export – allows you to export either SYZ data (*.s1p, *.ts, *.nmf, *.tab, *.m, *.cit) or Broadband data (*.sp).
- Check – allows you to check causality.
- Cores – allows you to enable or disable multithreading.
- Post-Process Selection – allows you to choose between Terminal Data and Differential Pairs, if your design includes Differential Pairs

Network Data Selection Pane

This pane allows you to view and compare various data sets. Original data sets appear under **Files**. Click one of these to see the data set as it was when it was opened. Altered data sets are listed under **Edited**. These data sets appear here when they have been smoothed, transformed, or changed in some way.



This pane also lists available variations for AEDT design solution data. Variations are listed under a setup name icon (for example “Setup1 : Sweep1”) that is listed under a design icon (“TeeModel”). Variations can be selected and displayed just like other data sets.

Use the **Shift** and **Ctrl** keys to select and display data sets from multiple files simultaneously. Multiple data sets can be selected to display multiple traces in sweep plots, but not in tables. Click the **Files** icon to select all data sets under **Files**. Click the **Edited** icon or a design/setup icon to do the same.

Cell and Frequency Selection Pane

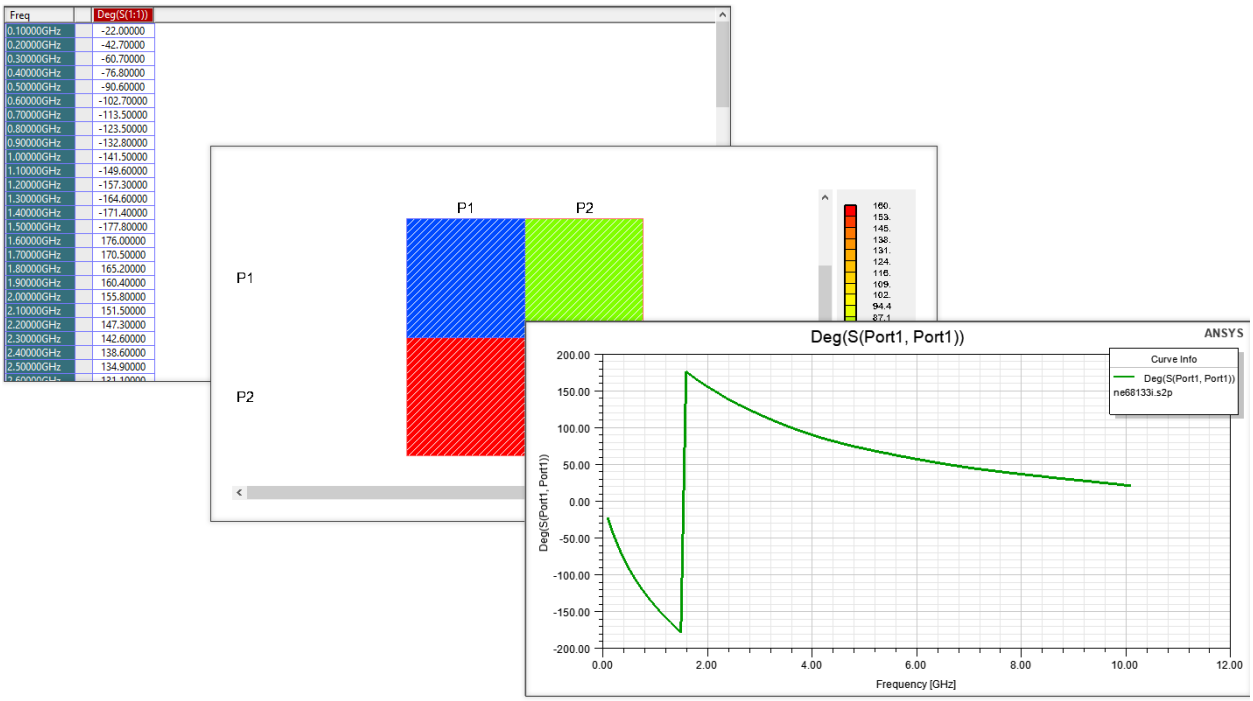
Select the **Sweep** button on the NDE ribbon to display cell entries, for example S(1,1). Select the **Matrix** button to display frequencies. When displayed by frequency, the entire matrix is presented in the Data View pane for each selected frequency. When displayed by matrix cell, the data for the individually chosen cells is shown across all frequencies. Use the **Select All** check box to select all frequencies or cells.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** For Maxwell 2D solutions when the geometry is in XY mode, a Model Depth selection is available. The default Matrix selection is the alphabetically first defined matrix parameter. The default Model Scale setting is 1. The default Model Depth setting is 1 meter.

Data View Pane

The Data View Pane displays data for **Sweep** or **Matrix** in either plots or a table, depending on your selection.



Loading Data into Network Data Explorer

You can launch **Network Data Explorer** from within several Ansys products. In the Project Manager, open <project> > **Analysis** > **Setup***n* > **Sweep***n*. Right click on the sweep and click **Network Data Explorer** in the shortcut menu. The current solution data is automatically loaded and ready for viewing. Otherwise, you must load a data file into Network Data Explorer.

> **Note:** When solution data that is loaded into NDE is modified and resimulated in another Ansys product, the NDE data automatically updates.

You can import the following file types:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Touchstone Format (*.s*p)
- Touchstone 2 Format (*.ts, *.sp)
- Citifile (*.cit)
- Neutral Format (*.nmf)
- State Space File (*.sss)

> **Note:**
>
> When this type of file is loaded, Network Data Explorer regenerates s-parameter data based on the file.
>
> You can compare the regenerated s-parameters to the original data.

To import a file into Network Data Explorer, either drag and drop an analysis from the Project Manager into Network Data Explorer or

1. On the **NDE** ribbon, click **Open**. An **Open** window opens.
2. Navigate to and select a file.
3. Click **Open**. The file appears in the **Files** tree.

The file browser allows you to open multiple files at a time. However, the displayed data always corresponds to the data set indicated in the Network Data Selection pane. Click the file you want in that pane to switch between data sets.

Exporting Data from Network Data Explorer

Network Data Explorer allows you to export data to a variety of different file formats.

In this section, you will learn about:

Exporting SYZ Data

Exporting Macro Model

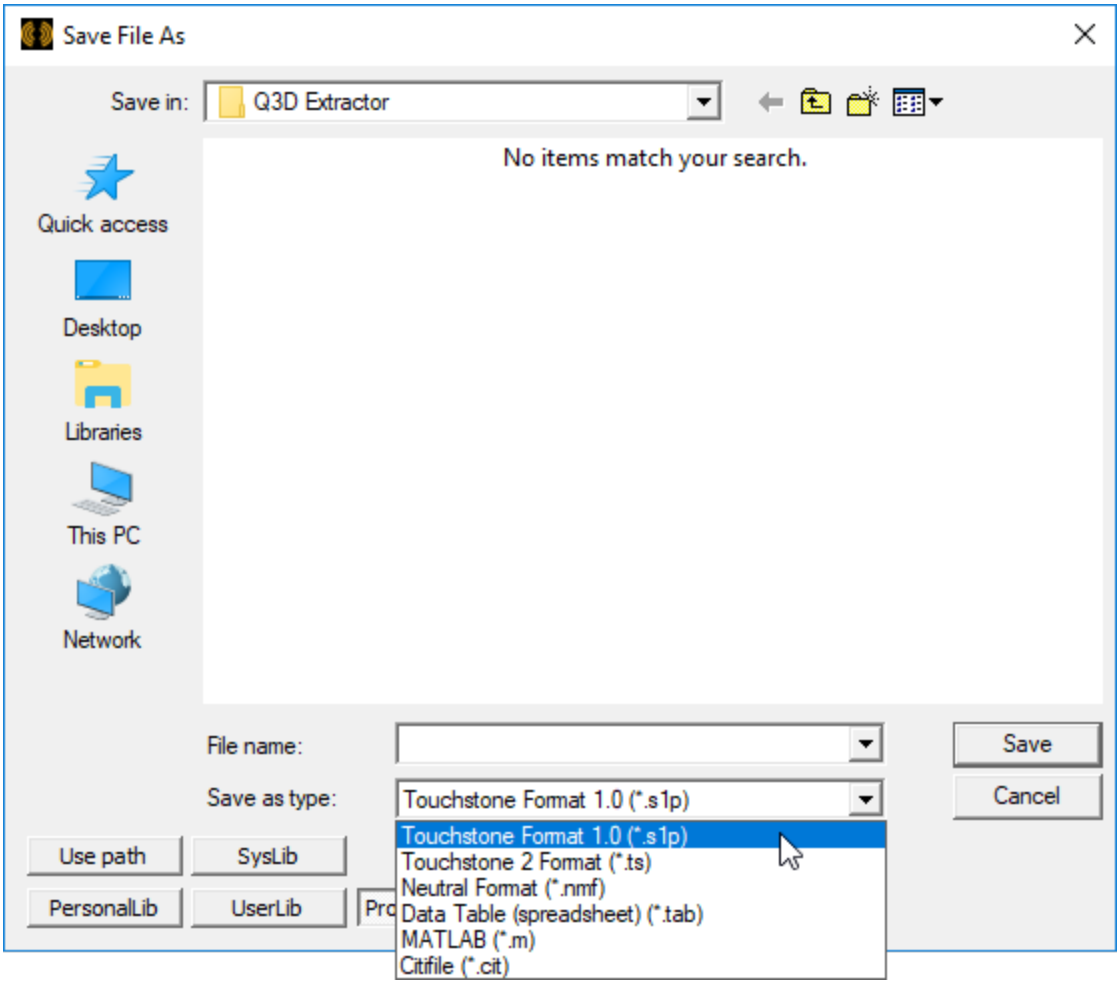
Creating an NPort Model

Scripting for Network Data Explorer

Exporting SYZ Data

To export SYZ data from within Network Data Explorer, click the **Save As** icon on the **NDE** ribbon. The **Save File As** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

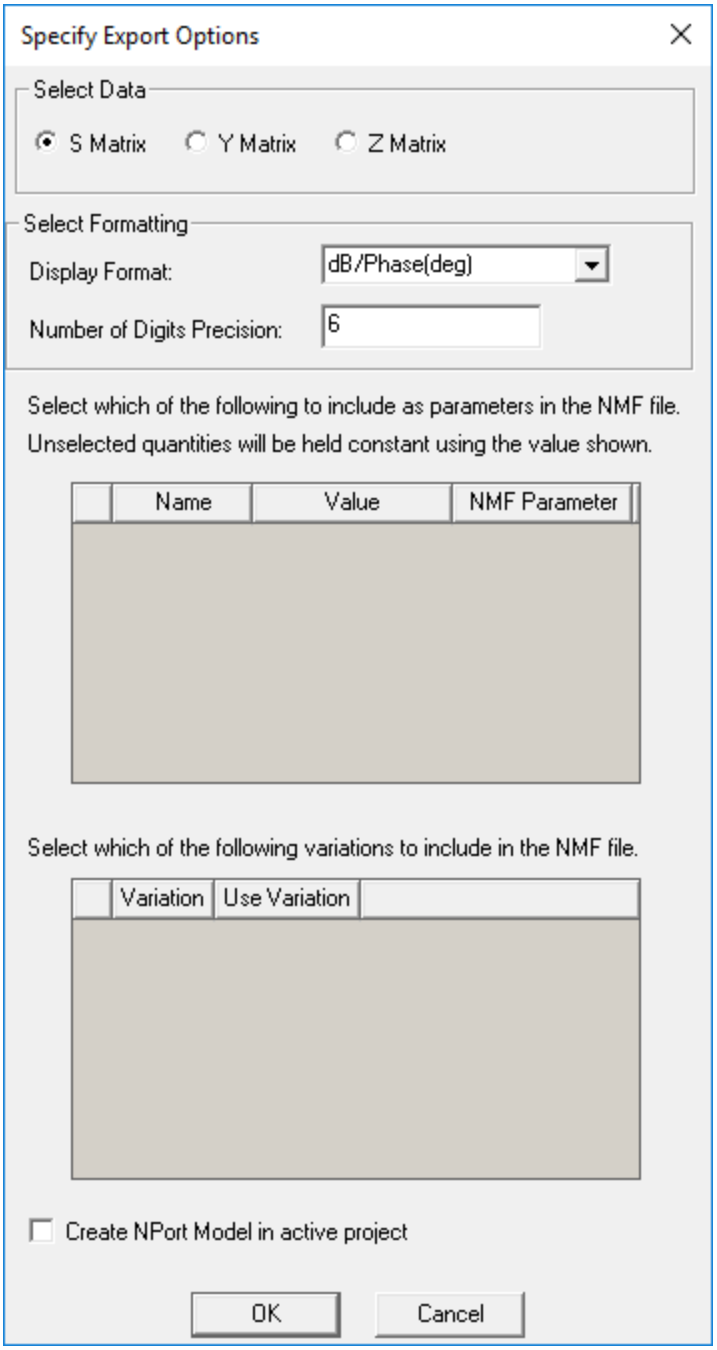


You can export data in any of six file types:

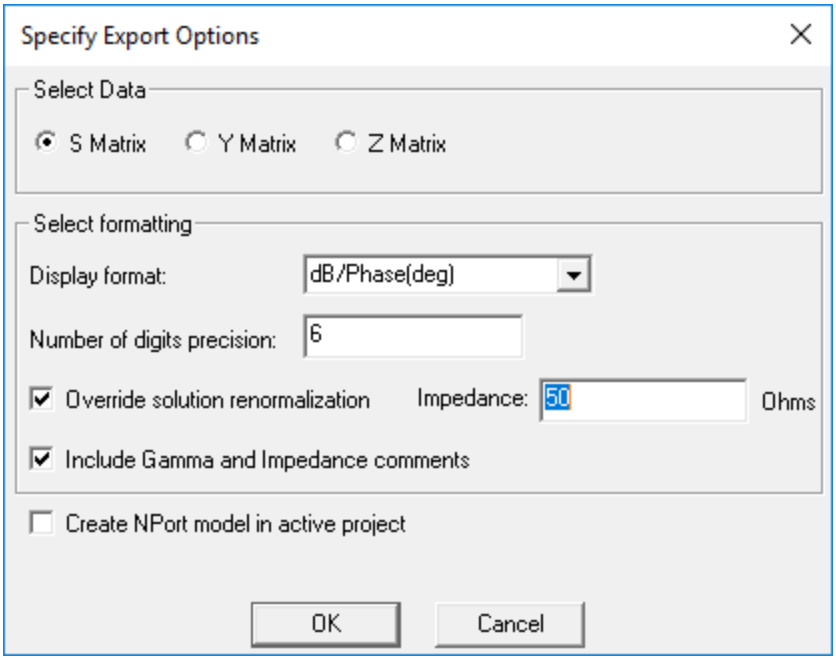
- Touchstone Format 1.0 (*.s*p)
- Touchstone 2 Format (*.ts)
- Neutral Format (*.nmf)
- Data Table Spreadsheet (*.tab)
- MATLAB (*.m)
- Citifile (*.cit)

Select a file type and name for export. A **Specify Export Options** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Depending on the type of export file, different options appear. However, all file types allow you to:

- Select from **S Matrix**, **Y Matrix**, and **Z Matrix** data.
- Select the **Display Format**.

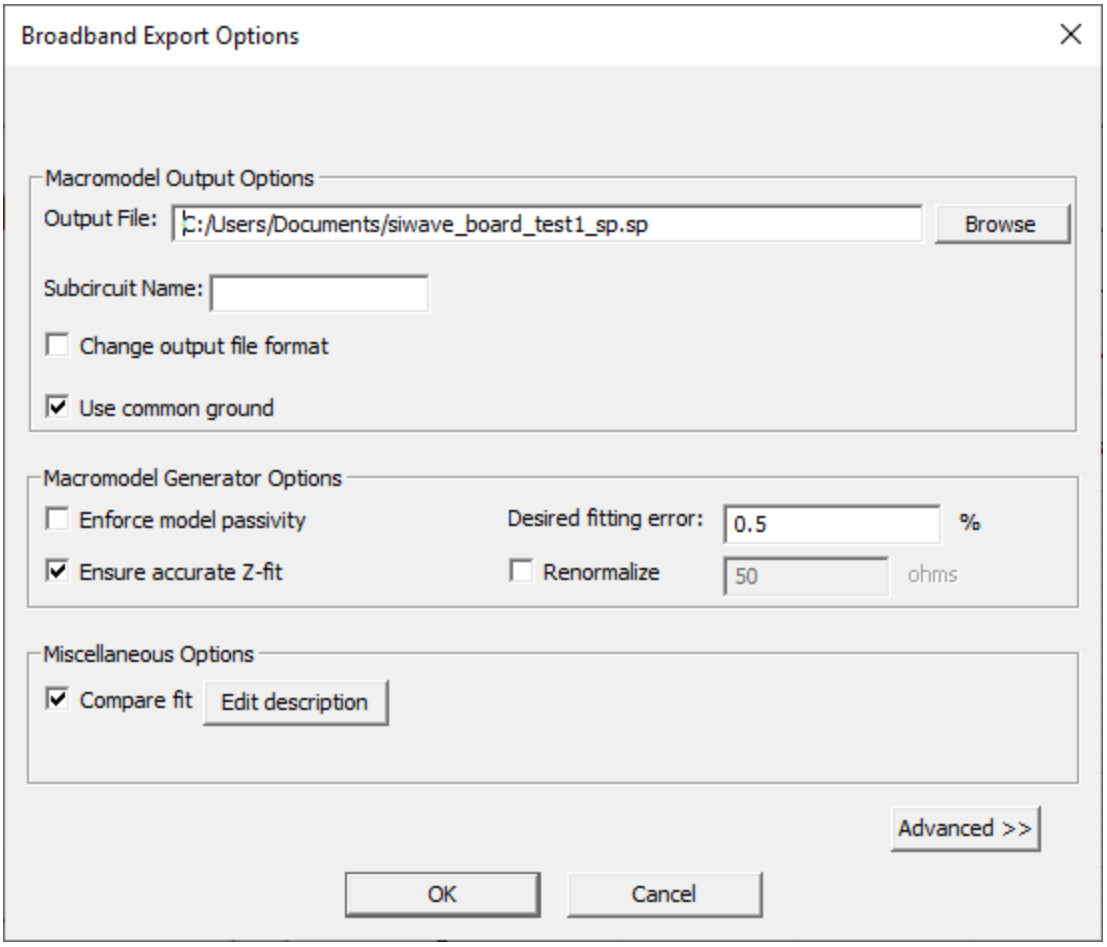
Some types allow you to create an NPort Model in the active project.

Exporting Macro Model

Network Data Explorer lets you export macro model data. To export data, click the **Broadband** icon on the **NDE** ribbon.

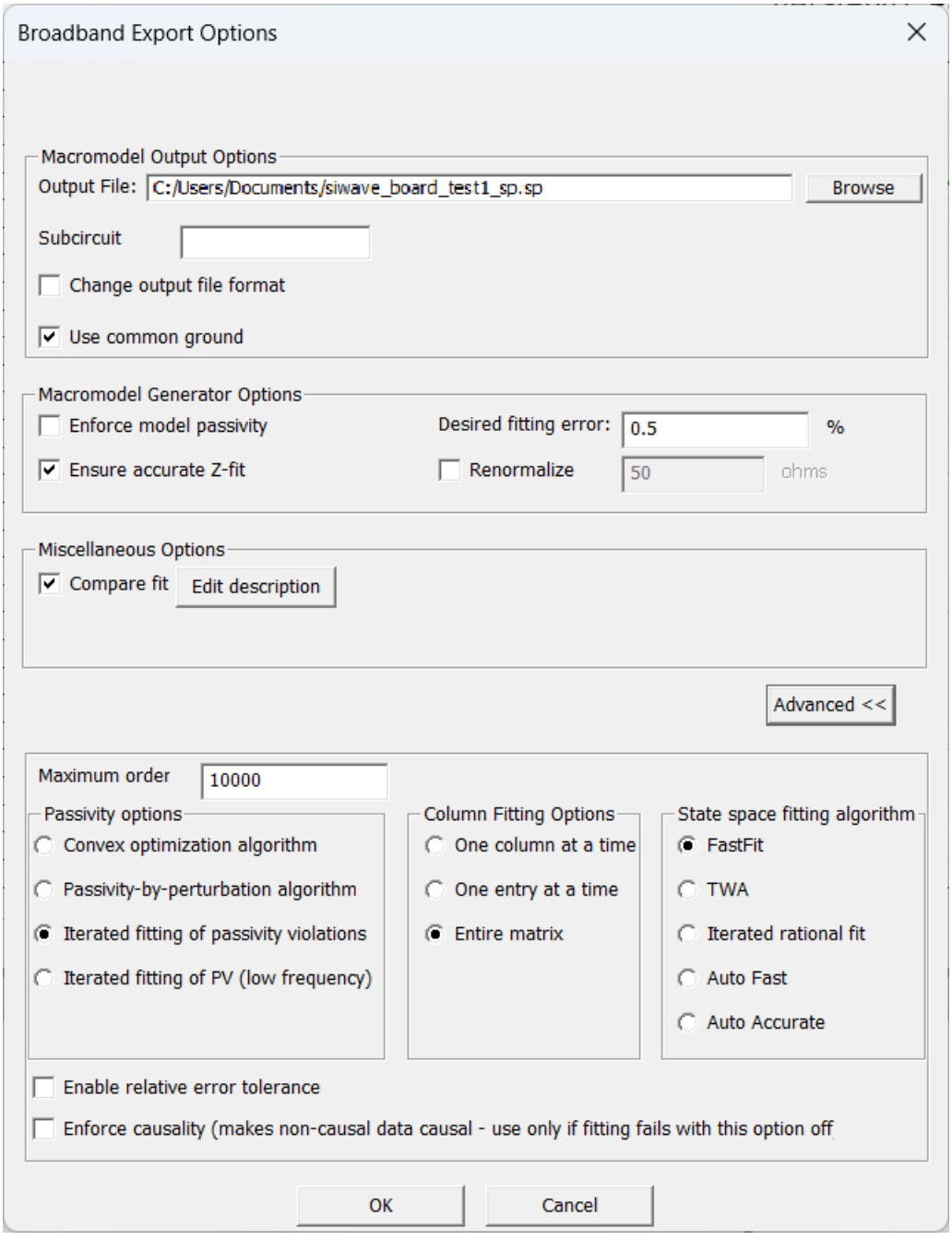
The **Broadband Export Options** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Click **Advanced >>** to view all options.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Macromodel Output Options include:

- **Output File** – Allows you to choose the name and location of the file.
- **Subcircuit Name** – Use this field to name the subcircuit.
- **Change Output File Format** – Check this box to open a submenu allowing you to select a new output format.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Use Common Ground** – Check this box to use common ground. When this option is on, ports are referenced to ideal ground (node 0). When this option is off, extra ports are generated to provide the reference levels. Common grounding is best when the pins are physically near to each other and ideal ground is suitable. For distant connections and circuits with non-ideal reference levels such as differential pairs, common grounding is not used.

  Note:

  - R and L values may be quite sensitive to the values of the S-parameters. This is an issue if the actual impedance value is much greater than or much less than the reference impedance of the S-parameters.
  - Since resistances of power cables is typically in the milliohms range at DC, using a reference impedance of 50 ohms is 5000 times higher. This causes any fitting errors in the state space model to get multiplied by 5000 times when the R and L values are computed.
  - As a general rule, for high power applications a reference impedance of 1 ohm is probably a better choice than 50 ohms.

Macromodel Generator Options include:

- **Enforce Model Passivity** – Check this box to enforce passivity.
- **Ensure Accurate Z-fit** – Check this box when state-space fitting of Y-parameters or Z-parameters does not produce an accurate fit.
- **Desired Fitting Error** – Allows you to select the value at which rational fitting fails (if the fitting error exceeds this value).
- **Renormalize** – Check this box to renormalize using the specified impedance item. 50 ohms is the default setting, but you can type a different value.

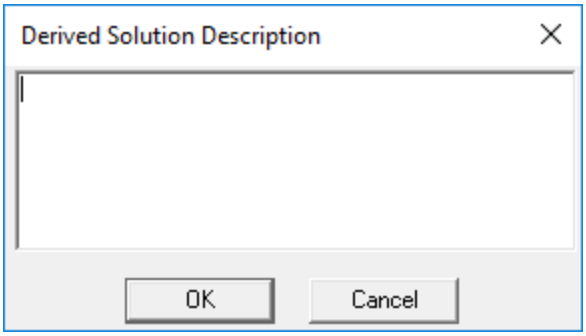
  Note:

  - R and L values may be sensitive to S-parameter values. This presents an issue if the actual impedance value is much greater than or much less than the reference impedance of the S-parameters.
  - Since resistances of power cables are typically in the milliohms range at DC, using a reference impedance of 50 ohms is 5000 times higher. This causes any fitting errors in the state space model to be multiplied by 5000 when the R and L values are computed.
  - For high-power applications, a reference impedance of 1 ohm is generally a better choice than 50 ohms.

Miscellaneous Options include:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Compare Fit** – When this box is checked, the original and derived solution will be available for comparison. You can click **Edit Description** to open the **Derived Solution Description** window and add a text description to better identify the export.



Advanced Options include:

- **Maximum Order** – Allows you to specify the number of poles. See Note below.
- **Passivity Options** – If you enabled **Enforce Model Passivity**, this area allows you to select the passivity enforcement method.
  - **Convex optimization algorithm** – guarantees a passive state-space realization, but is very slow and memory-intensive. Not practical for numbers of ports beyond ten.
  - **Passivity-by-perturbation algorithm** – designed for systems with a large number of ports. Less accurate than the Convex optimization method.
  - **Iterated fitting of passivity violations (IFPV)** (default) – less accurate than other algorithms but more suitable for larger numbers of ports.
  - **Iterated fitting of PV (low frequency)** – similar to IFPV while improving the fit to "Z" at DC and low frequencies. A better choice of passivity enforcement when the fit to corresponding "Z-data" is important such as power delivery, EMI/EMC applications.

    > **Note:**
    >
    > For a more detailed explanation on any of the passivity options, see the technical note section of the Circuit help.

- **Column Fitting Options** – This area allows you to choose how poles are matched to columns:
  - **One Column at a Time** – The set of poles will be shared across all entries of a single column.
  - **One Entry at a Time** – Each entry will be fitted using a separate set of poles.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Entire Matrix** – The set of poles will be shared across all entries of the matrix being fitted.

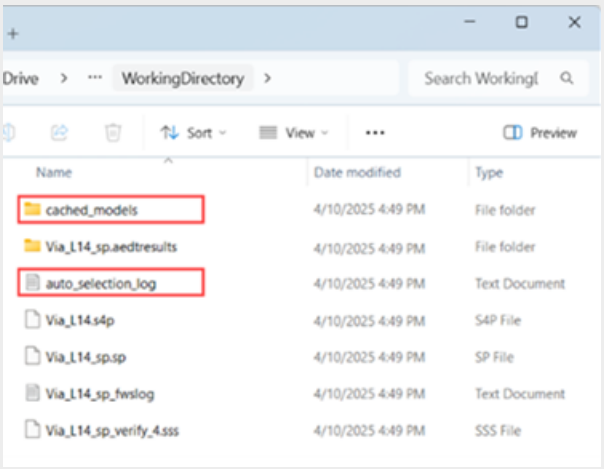
> **Note:**
>
> - Typically, using the same set for all entries is adequate, and yields the most compact models. However, if all the entries of the matrix have completely unrelated transfer functions, it may be better to fit them using separate pole sets.
> - The options **One column at a time** and **One entry at a time** do not work when either **Ensure accurate Z-fit** or FastFit is used.

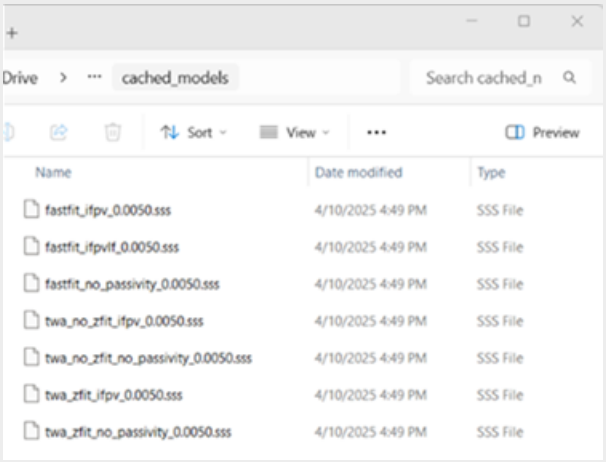
- **State Space Fitting Algorithm** – Allows you to select FastFit, TWA, Iterated rational fitting, Auto Fast, or Auto Accurate.
  - **FastFit** (default) – FastFit is the Ansys-proprietary method for state-space fitting. Network Data Explorer uses FastFit for calculating the state-space matrices from the network data. The FastFit algorithm for state-space fitting is an alternative to the Tsuk-White algorithm (TWA) and Iterated Rational Fitting (IRF) methods. FastFit is generally as accurate as TWA, but is significantly faster than both TWA and IRF. It also aims to fit the lower frequencies with higher fidelity.
  - **TWA** – The Tsuk-White Algorithm is an Ansys-proprietary method for fitting a state space model to extracted s-parameter data. It uses techniques based on Singular Value Decomposition (SVD) to quickly determine required number of poles for fitting a model.
  - **Iterated Rational Function** – The IRF fitting approach takes a matrix of S-parameter data and, for each matrix entry, tries a succession of different pole-zero approximations (increasing the number of poles used at each iteration) until it can find an acceptable fit to the data. For broad frequency sweeps and large numbers of excitations, this process can be time consuming because of all the iterations and is not guaranteed to produce a good fit to the data. It is retained as a fallback if the TWA algorithm fails.
  - **Auto Fast**- (Beta) To use the Auto Fast option, first enable **State Space Fitting Auto Selection** in the Beta Options dialog. The Auto Fast option runs all the fitting options in parallel and returns the first model that meets the user's specifications. It takes less time, however, the model may not be the one with the least fitting error.
  - **Auto Accurate** - (Beta) To use the Auto Accurate option, first enable **State Space Fitting Auto Selection** in the Beta Options dialog. The Auto Accurate option runs all the fitting options to completion, then returns the model with the least fitting error. This option may take longer but it is guaranteed to return the best fit model.

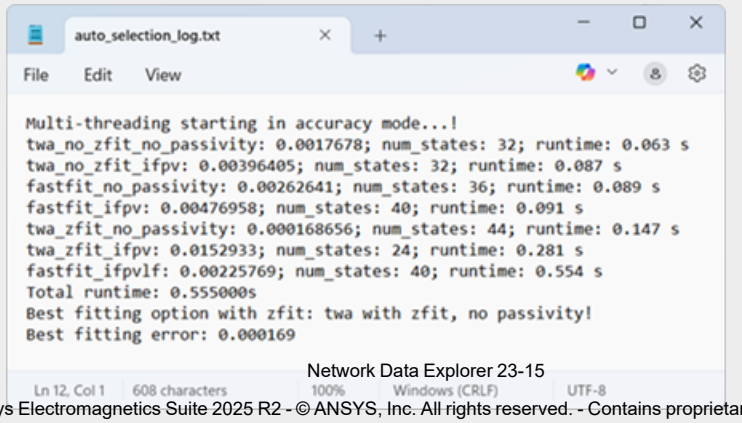
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

After a successful completion of Auto Fast or Auto Accurate, a new folder and a log file (.txt) is created in the working directory. The folder, named cached_models, contains the .sss files from the parallel runs of the different state-space fitting options. The log file, named auto_selection_log.txt, contains the status of each fitting thread.







Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

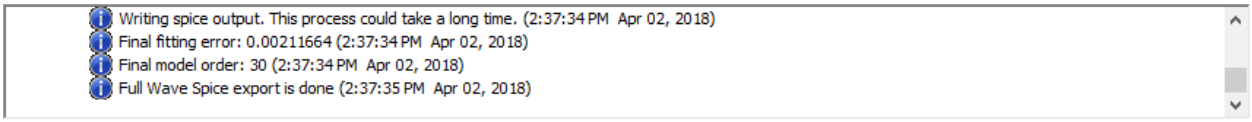
- **Enable Relative Error Tolerance** – Allows you to enable relative error tolerance, which works best with TWA fitting.

  > **Note:**
  >
  > The **Enable Relative Error Tolerance** option works best with the TWA fitting algorithm, is not recommended for use with iterated rational fitting, and is disabled when either FastFit or **Ensure accurate Z-fit** is used.

- **Enforce Causality** – Allows you to make non-causal data causal. Use this option only if fitting fails without it.

  > **Note:**
  >
  > Broadband models are built from a rational-function approximation of the data. The fidelity of this approximation can be controlled by setting the Maximum order (number of poles).

Click **OK** to begin the export. The **Messages** pane details the export process.



Comparing Original S-Parameters with Exported S-Parameters

If **Compare Fit** was checked during export, the **Data Selection** pane updates to list both the original and exported solution *and* the **Compare** checkbox is checked.

Creating an NPort Model

Touchstone, Neutral Format, and Citifile exports allow you to select an option to **Create NPort Model in active project**.

Creating an NPort Model exports the active data set back to the design, either as a static n-port model or as a parametric n-port model. If there are multiple variations, a parametric n-port model is automatically created containing all variations. When only a single variation exists, the user can create a static n-port model that either links to a file (the data is exported first) or stores the data itself. Linking to a file reduces the size of the ADSN file. The newly created model can be placed as a component in a circuit. This is particularly useful after reducing the number of ports via termination.

The import of the exported solution is done by reading the exported Touchstone file.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scripting for Network Data Explorer

Scripting is available for each Network Data Explorer export method, which means a script can be recorded to duplicate the export process.

Network Data Explorer can be invoked in the following contexts, and scripting is available from the Project Context and the Design Instance Context (simulation setup):

- **Project Context** – In the context of a project, you can open a touchstone file and then export.
- **Design Instance Context** – In the context of a solution (RCM from the simulation setup in a design), you can export the corresponding network data solution.

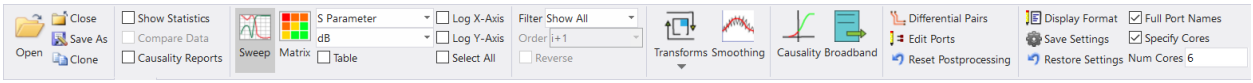
When there is no design available, export functionality and scripting are not available.

When there is no design available, export functionality and scripting are not available.

For more information, see Network Data Explorer Script Commands in the *Ansys Electronics Desktop Scripting Guide*.

Network Data Explorer Ribbon

The **NDE** ribbon provides access to many of the Network Data Explorer's functions.



In this section, you will learn about:

Data Sources

Setting Display Format

Displaying Full Port Names

Saving or Resetting Default Settings

Smoothing All Frequencies

Cell Filtering

Editing Port Properties

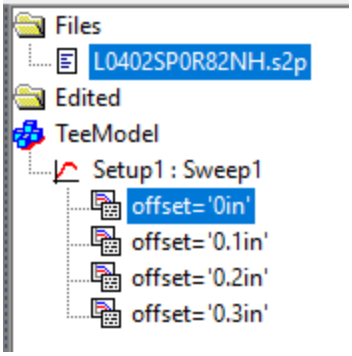
Defining Differential Pairs and Displaying Mixed Mode Parameters

Resetting All Port Properties

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Network Data Explorer Data Sources

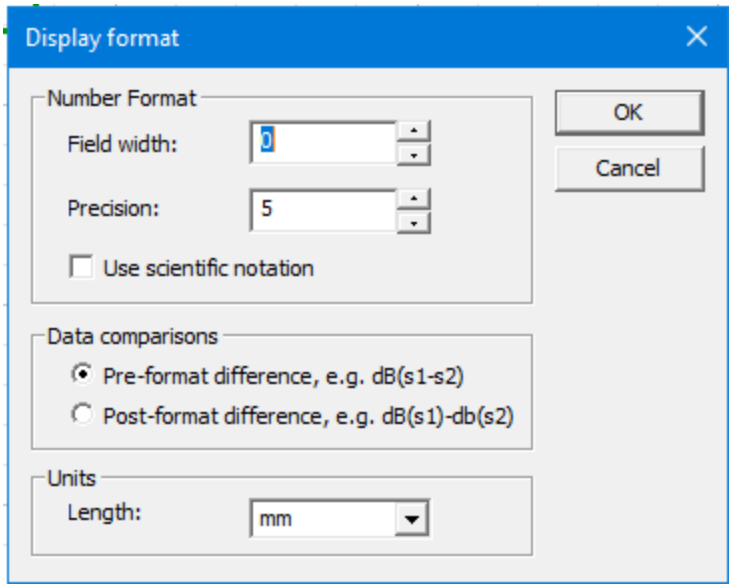
Network Data Explorer allows you to easily view all data sources in the **Network Data Selection** pane.



Network Data Explorer Display Format

The **Display Format** window affords additional control over the display of values in Network Data Explorer.On the **NDE** ribbon, click the **Display Format** icon.

The **Display Format** window appears.



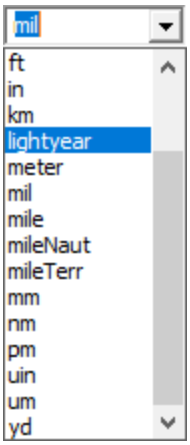
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Number Format** options allow you to specify **Field Width** (the minimum number of characters used to display a number) and **Precision** (the number of decimals to display). You can also check the **Use scientific notation** check box, if desired.

The **Data comparisons** options allow you to choose Post-format difference or Pre-format difference.

- **Pre-format difference** – when comparing data sets, subtract values before applying the formatting function (e.g. dB, magnitude); the values displayed will be the magnitude, dB, etc., of the complex difference.
- **Post-format difference** – when comparing data sets, subtract values after applying the formatting function (e.g. dB, magnitude); the values displayed will be the difference between the magnitude, dB, and so on.

The **Units** option allows you to specify the **Length** unit (the unit used to display and interpret length values). The default is mm.

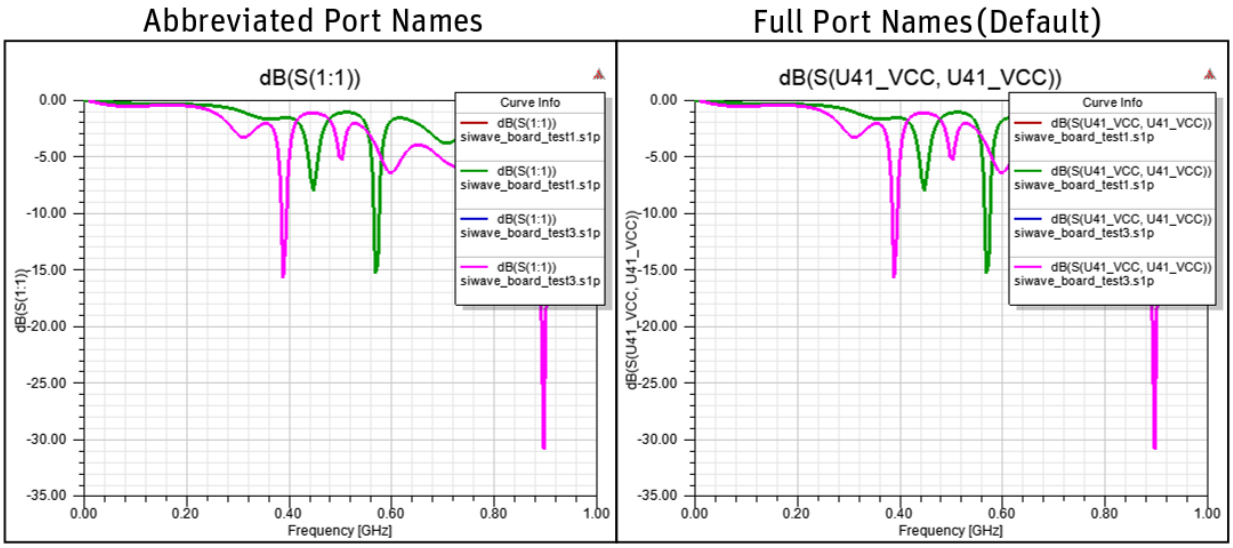


Network Data Explorer Display Full Port Names

By default, full port names are displayed. This applies to both the Data Selection pane and the Data View pane. To change this so that port names in Network Data Explorer are displayed in an abbreviated form (P1, P2, etc.), click the **Full Port Names** check box on the **NDE** ribbon.

The following figure shows the difference in display for a plot.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

Tool-tips always display the full port name.

Network Data Explorer Save or Reset Default Settings

To save field settings as the default, click the **Save Settings** icon on the **NDE** ribbon. The next time Network Data Explorer is opened, the chosen settings will be selected by default.

To restore the default settings (i.e. the settings previously saved using the **Save Settings** button), click the **Restore Settings** icon on the **NDE** ribbon. The next time Network Data Explorer is opened, the original settings will be selected by default.

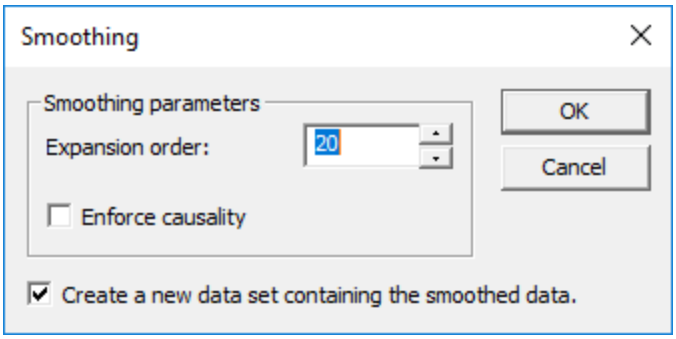
Smoothing

To access data smoothing options:

1. Click the **Smoothing** icon on the **NDE** ribbon.

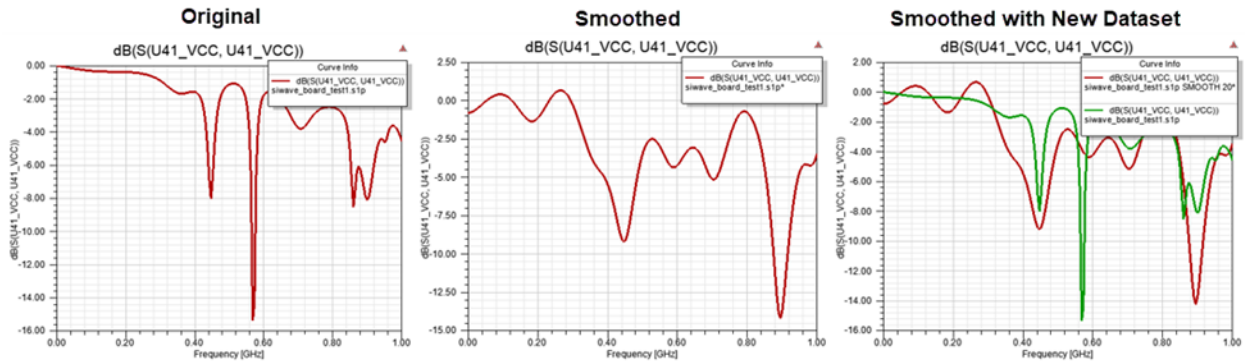
   The **Smoothing** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. The **Smoothing Parameters** area allows you to choose the **Expansion Order**. This can be any discrete value between 1 and 150.
3. If desired, check the **Enforce Causality** check box.
4. If desired, check the **Create a new data set containing the smoothed data** check box. If selected, the smoothed data appears alongside the original data.
5. Click **OK**.

The Data View Pane updates. The following image shows a plotted result with **Create a new data set containing the smoothed data** unchecked and checked.

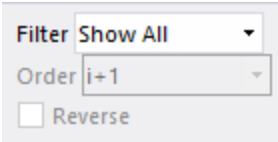


A least-squares polynomial fit of the specified order is used to interpolate new data points for the magnitude and phase components of the S-parameters.

Cell Filtering

The cells available in the data selection pane may be restricted using cell filtering. The **Cell Filtering** control are located on the **NDE** ribbon.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Cell filtering is modeless, and filters are immediately applied to the cell list. Filtering remains in effect when the window has been closed.

For an n-port model with a total of 2n pins in the standard arrangement, the choices are:

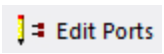
- **Show all** – display all available cells. There are n-squared choices.
- **Return loss** – show S(i, i). There are n choices.
- **Insertion loss** – show S(i, i+1). There are n choices.
- **Lower triangle** – show S(i, j) for all j < i. There are n(n-1)/2 choices.

Three pin arrangements are recognized:

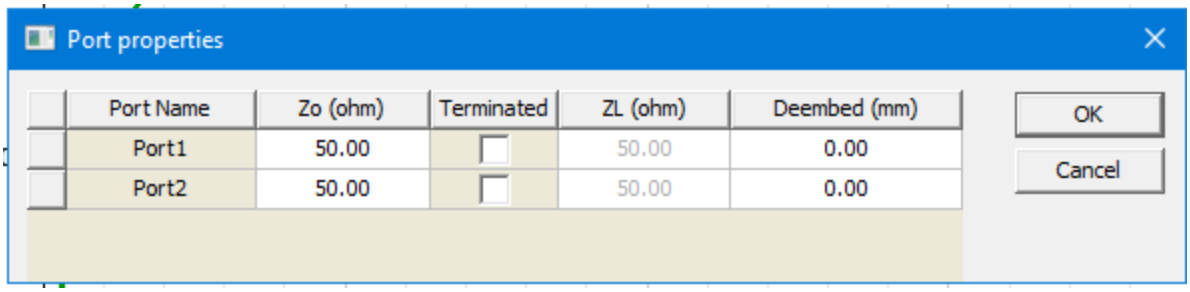
- S(i, **i+1**) and its **Reverse order**, S(i, i-1)
- S(i, **i+n**) and its **Reverse order**, S(i, i-n)
- S(i, **2*n-i+1**) and its **Reverse order**, S(i, 2*n-i-1)

Changing Port Properties and Reducing Matrix Size

The normalization impedance, termination, port order, gamma values, and de-embedding distance may all be edited from the **Port Properties** window.

To access these options, click **Edit Ports** 

on the **NDE** tab. The **Ports Properties** window opens.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** For HFSS Driven Terminal designs, selecting the solution data will disable this ribbon button. The user must make port/differential pair changes directly in the HFSS Design and re-import the solution.

Ports appear in a table. Click a column heading to sort by that column. Click within a cell to edit the port property:

- **Zo (ohm)** and **ZL (ohm)** – specify Impedance values. Accepted syntaxes are:
  - real (e.g., 50)
  - real + imag i (e.g., 50+5i)
  - imag i (e.g., 5i).
- **Terminated** – use the check box to terminate a port. Terminated ports are eliminated from the matrix, reducing the matrix size. Existing data sets with mismatching port numbers will no longer be available for data comparisons.
- **De-Embedding** – this column appears only if gamma values are available. Default units can be changed from the Set Display Format window.

To reorder ports, click and drag a row to a new location.

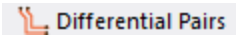
To save changes, click **OK**.

Displaying Mixed-Mode Parameters using Differential Pairs

Network Data Explorer displays mixed-mode parameters when differential pairs are both defined and activated.

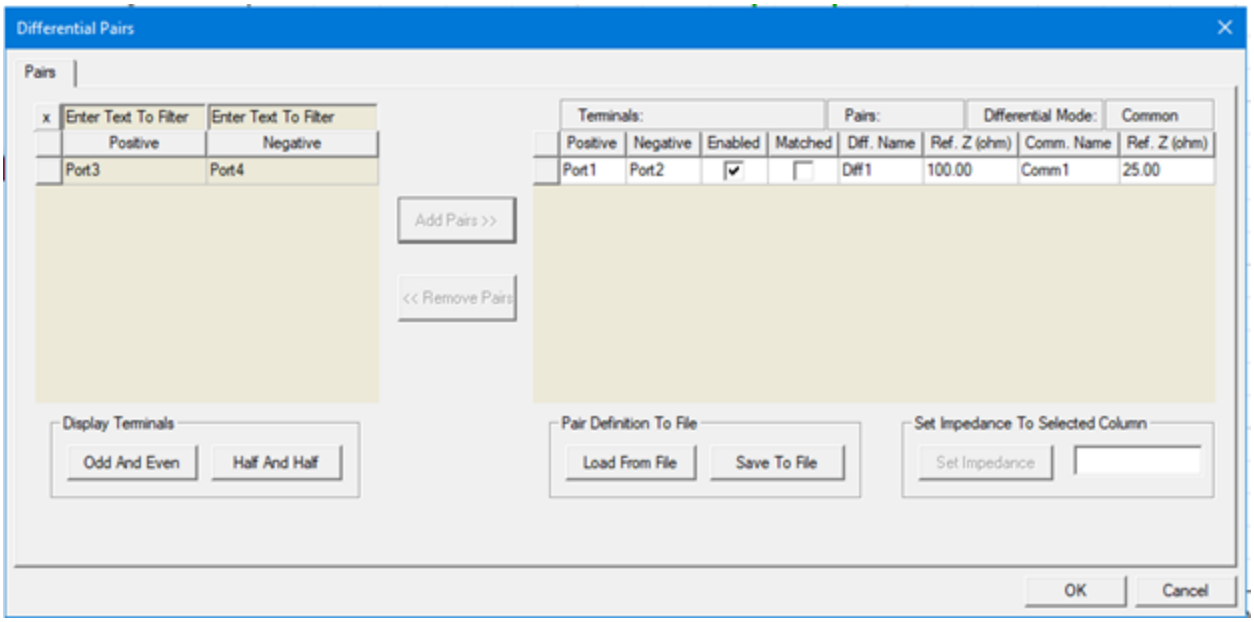
To define differential pairs:

1. Select existing ports.
2. Open Network Data Explorer (**Tools**>**Network Data Explorer**).
3. On the **NDE** ribbon, select **Differential Pairs**.



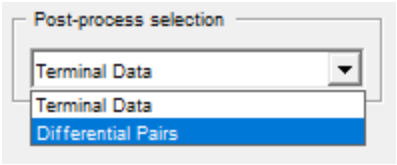
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Differential Pairs** window appears.



4. Select a pair from the list on the left and click **Add Pairs**.
5. Click **OK**.
6. To disable all differential pairs, click the **Enabled** column header in the **Differential Pairs** dialog to deselect all pairs.

For HFSS Driven Terminal designs, selecting the solution data will disable this ribbon button. The user must make port/differential pair changes directly in the HFSS Design and re-import the solution. When the HFSS Driven Terminal Design has differential pairs defined, NDE shows this drop down menu and you can change between showing reports for Terminals or Differential Pairs.Select **Differential Pairs** from the **Post-process selection** field to view mixed-mode parameters.



Note:

The Network Data Explorer **Edit** menu option **Reset All Port Properties** deactivates all pairs, but it does not clear the differential pair settings. And since **Reset All Port Properties** also clears reference impedances and terminations, it should not be used when the user simply wishes to disable all differential pairs.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reset All Port Properties

Reset All Ports resets all changes in the **Edit Ports** window. It also deactivates all differential pairs defined in the **Differential Pairs** window but does not remove the definitions.

> **Note:** For HFSS Driven Terminal designs, selecting the solution data will disable this ribbon button. The user must make port/differential pair changes directly in the HFSS Design and re-import the solution.

Data View Pane Context Menus

The Data View pane presents different right-click menu options, depending on the context. Some commands are the same as those on the NDE ribbon. Others appear only in the context menus.

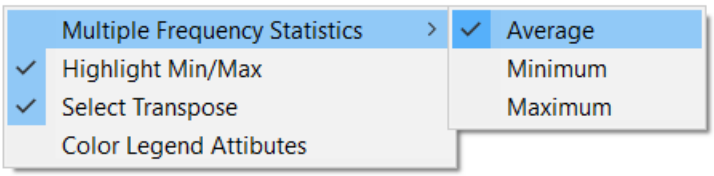
Network Data Explorer commands unique to the context menus are:

- Multiple Frequency Statistics
- Highlight Min/Max
- Select Transpose
- Color Legend Attributes
- Matrix Entries Plot Menu

Multiple Frequency Statistics

The **Multiple Frequency Statistics** menu option determines the statistical composite to display when multiple frequencies have been selected for the matrix display. The statistical data is always the first matrix displayed, followed by matrices for each individual frequency. The **Multiple Frequency Statistics** option also indicates the data used in the colored matrix plot when multiple frequencies have been selected.

This right click menu option appears in the **Matrix** plot, regardless of whether you are in Table or Plot view.



The menu options are:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

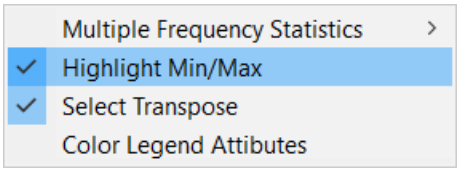
- **Average** – display the average of the matrix values across selected frequencies.
- **Minimum** – display the minimum matrix values across selected frequencies.
- **Maximum** – display the maximum matrix values across selected frequencies.

The selected information also appears in a tool-tip when you hover the cursor over a cell.

Highlight Min/Max

The **Highlight Min/Max** menu option determines whether the minimum and maximum matrix entries should be highlighted in the matrix table and color plot view.

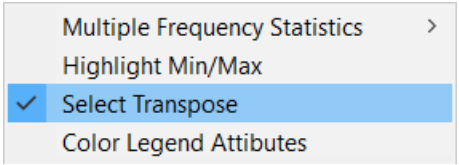
This right click menu option appears in the **Matrix** plot, regardless of whether you are in Table or Plot view.



Select Transpose

The **Select Transpose** menu option determines whether transpose cells are highlighted along with selected cells.

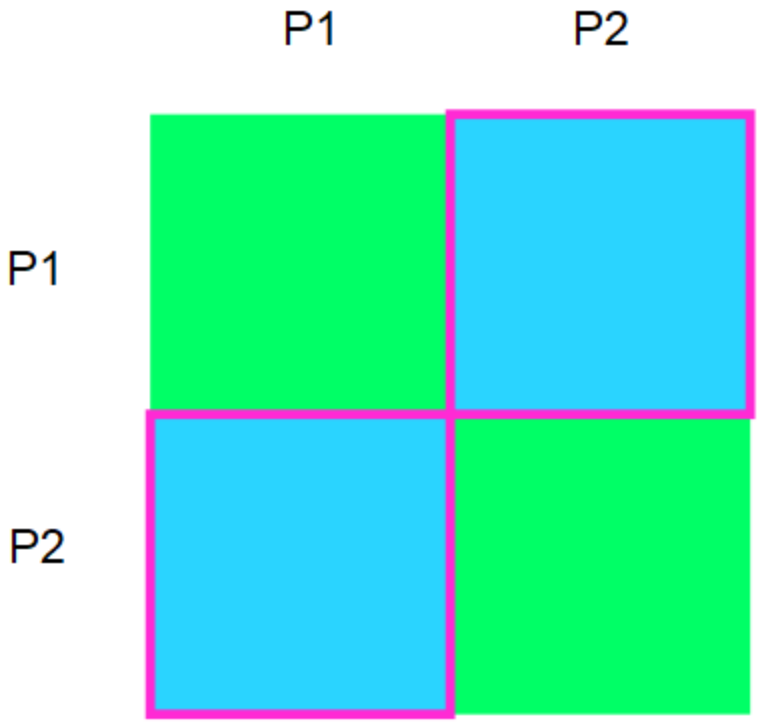
This right click menu option appears in the **Matrix** plot, regardless of whether you are in Table or Plot view.



Transpose cells are highlighted in Table or Plot view, as shown below.

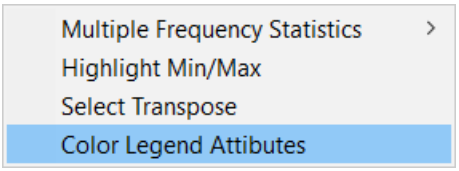
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0.01000GHz	P1	-57.10340, -81.97841	-57.72581, 85.70836	-0.00213, -0.18916
	P2	-57.72582, 85.70825	-57.09556, -81.98749	-66.27113, 99.16916
	P3	-0.00213, -0.18916	-66.27110, 99.16941	-57.10246, -81.95892
	P4	-66.27112, 99.16942	-0.00213, -0.18929	-57.72556, 85.71465



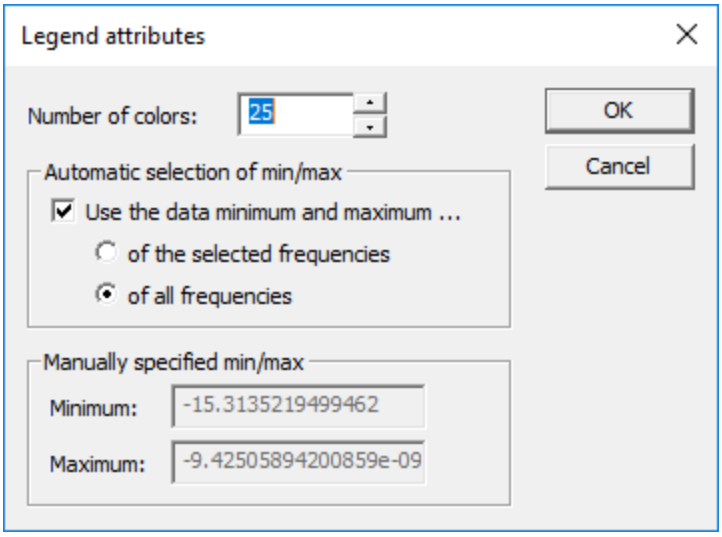
Color Legend Attributes

The **Color Legend Attributes** menu option allows you to change the granularity of the color scheme and the value range for plots. This right click menu option appears in the **Matrix** plot.



Alternatively, double-click the matrix plot's legend to open the **Legend Attributes** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



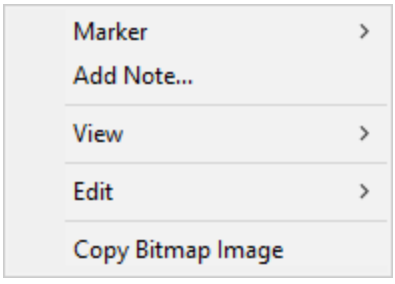
Options include:

- **Number of colors** – allows you to select number of color entries in the legend (the number of divisions between the start/end of the data range). This can be set to any discrete number between 1 and 50.
- **Automatic selection of min/max** – check the **Use the data minimum and maximum** check box to automatically select the data range using the minimum and maximum values from either selected frequencies or all frequencies in the data set.
- **Manually specified min/max** – when the when the range is not automatically determined, these fields permit the user to manually enter hard values. For example, for S parameter data magnitude data, you could enter a minimum of 0 and a maximum of 1.

Using a standard range across all frequencies permits you to quantitatively compare plots, and ndExplorer remembers legend settings for each data-type and display-format pair.

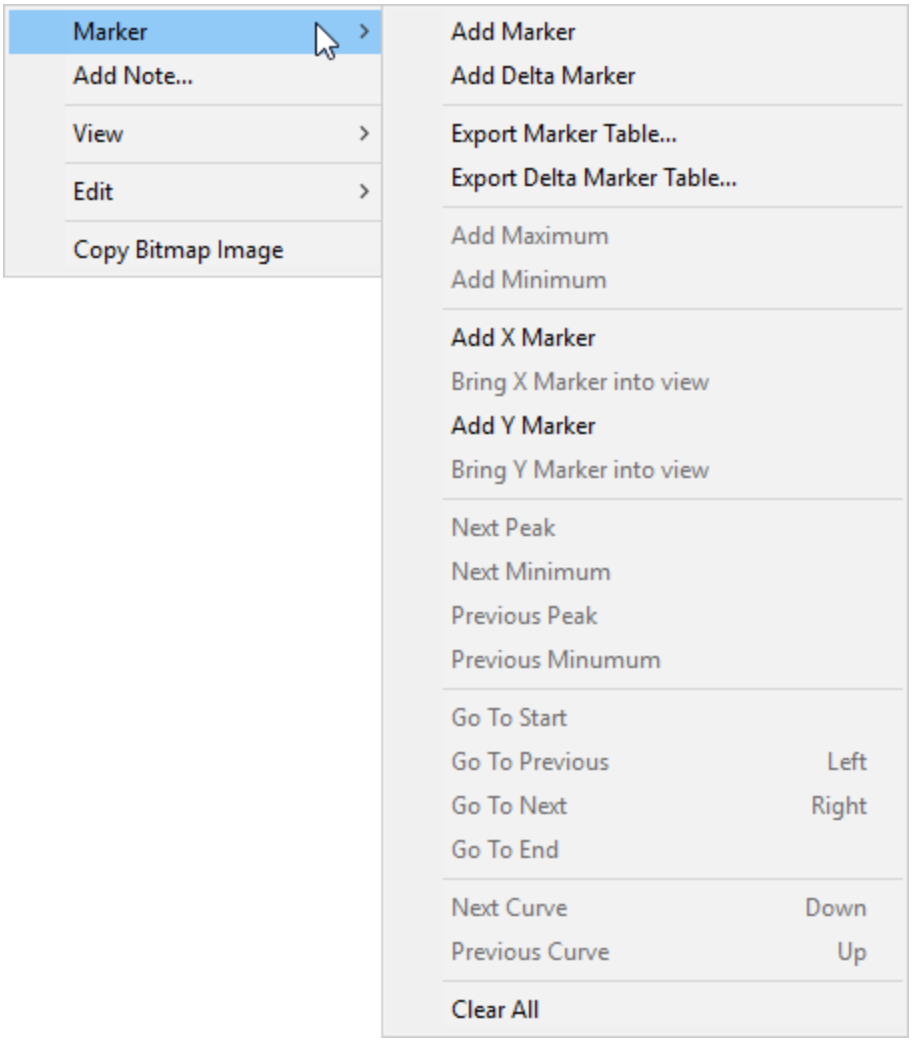
Matrix Entries Plot Menu

On the **Matrix** plot, several new right-click menu options appear.

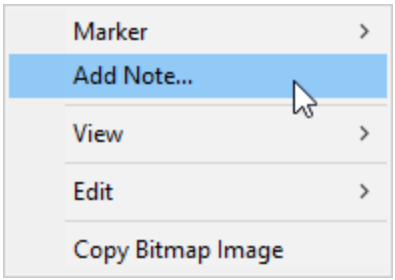


The **Marker** sub-menu provides commands for adding markers to plots.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

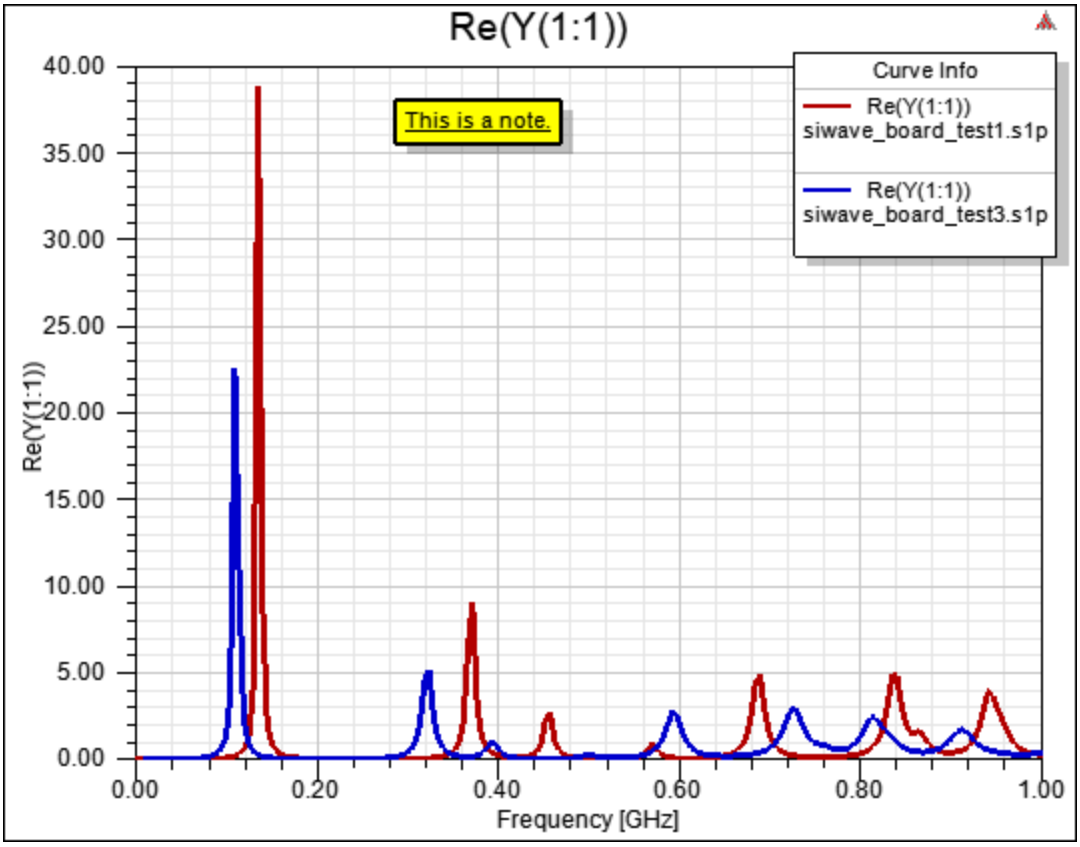


The **Add Note** menu option allows you to add a note to the plot.



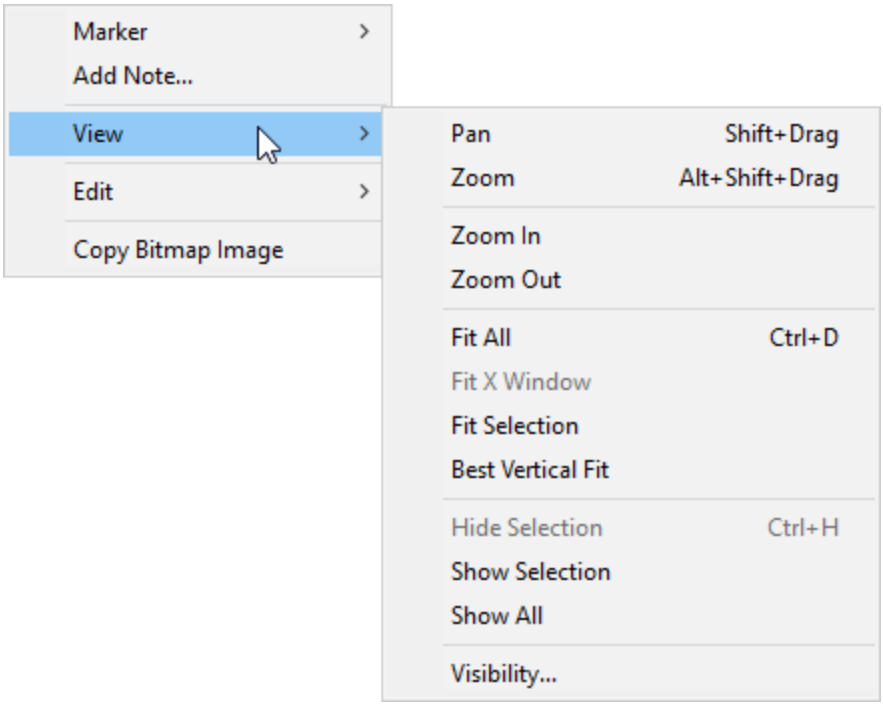
The note appears at the location you right-clicked. You can click and drag the note to a new location, or double-click the note to change its color and font.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



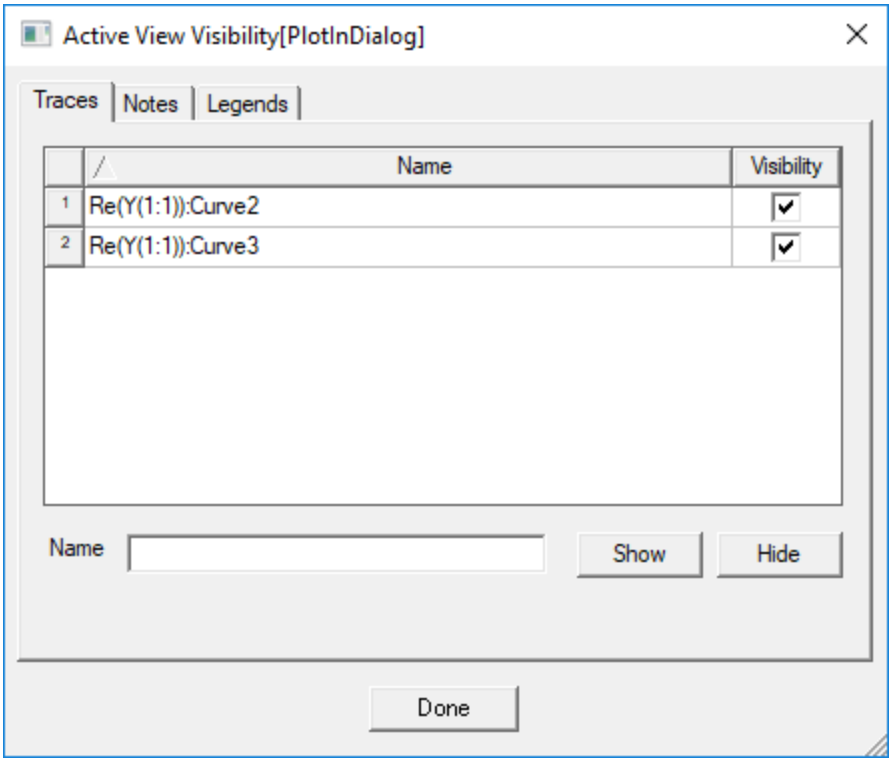
The **View** sub-menu provides commands for viewing, panning, zooming, and fitting elements to your plot.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

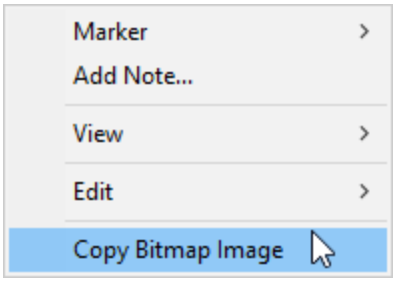


Click **Visibility** to open the **Active View Visibility** window, where you can select the visibility of traces, notes and legends.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Copy Bitmap Image** menu option copies the plot to your clipboard. You can then paste it into a graphics editor.



Exploring Network Data and Modifying the Display

Network Data Explorer allows you to view data and modify various aspects of the display, including color plots, color coding, viewing across frequencies, and displaying individual statistics. This section provides examples of Network Data Explorer capabilities:

- Viewing the S, Y, or Z Matrix for a Frequency
- Viewing a Color-coded Matrix Plot
- Displaying a Cell Graph Across All Frequencies

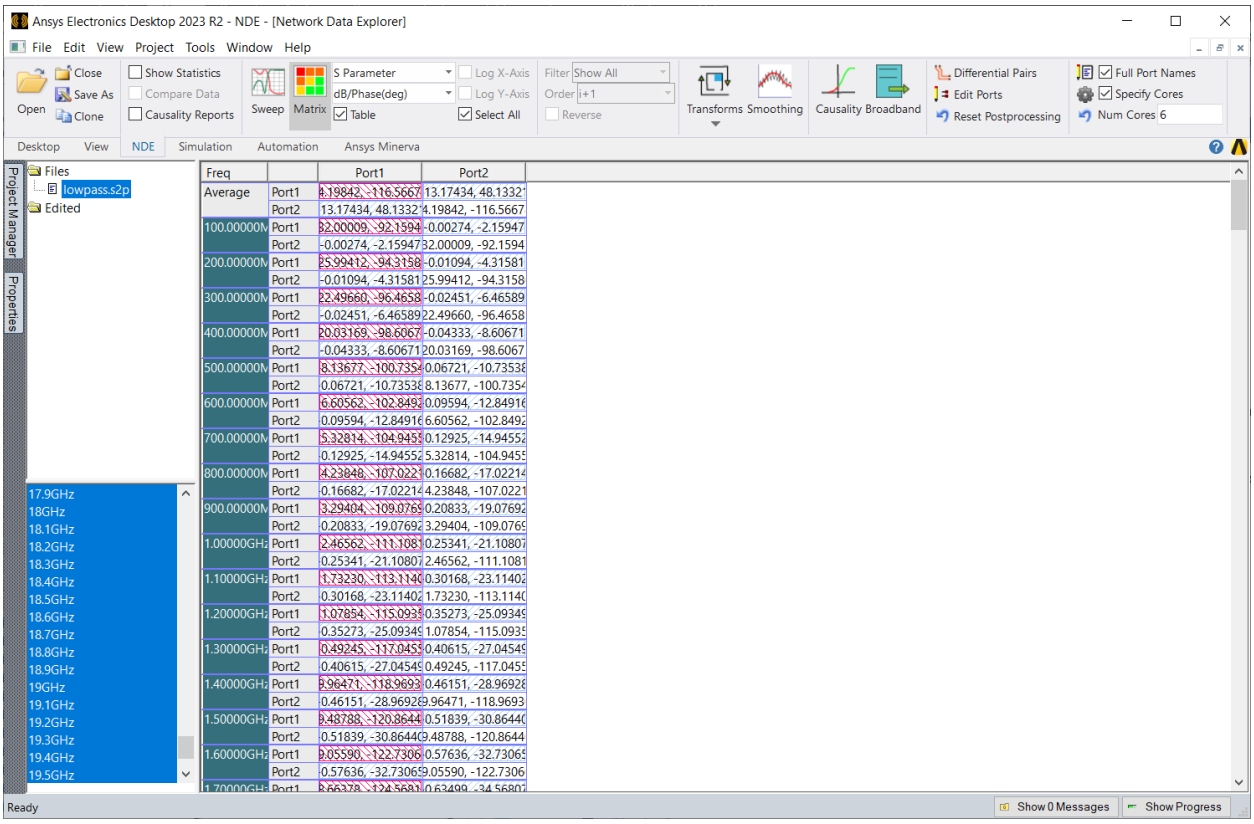
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Displaying Matrix Statistics by Frequency
- Displaying Individual Statistics for All Frequencies
- Creating a Statistics Plot
- Comparing Variations

Viewing a Matrix Table

To view the S, Y or Z matrix:

1. On the **NDE** ribbon, click **Matrix**.
2. Click the **Table** check box.
3. Use the **Parameter** type drop-down menu to select **S parameter**, **Y parameter**, **Z parameter**, or another choice.
4. Choose a format from the drop-down menu, for example dB or Mag.
5. In the Cell and Frequency Selection pane, select frequencies to display or click the **Select All** check box on the **NDE** ribbon to select all frequencies. The S, Y, or Z matrix displays.



6. Hover over a cell to see more information in a tool tip.
7. Right click to see a right click menu with other commands.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Maximum values are highlighted in red stripes. If Select Transpose is enabled, transposes are highlighted in red stripes as well.

Minimum values are highlighted in blue stripes.

Selected cells appear in solid blue. If Select Transpose is enabled, the transpose is highlighted in blue as well.

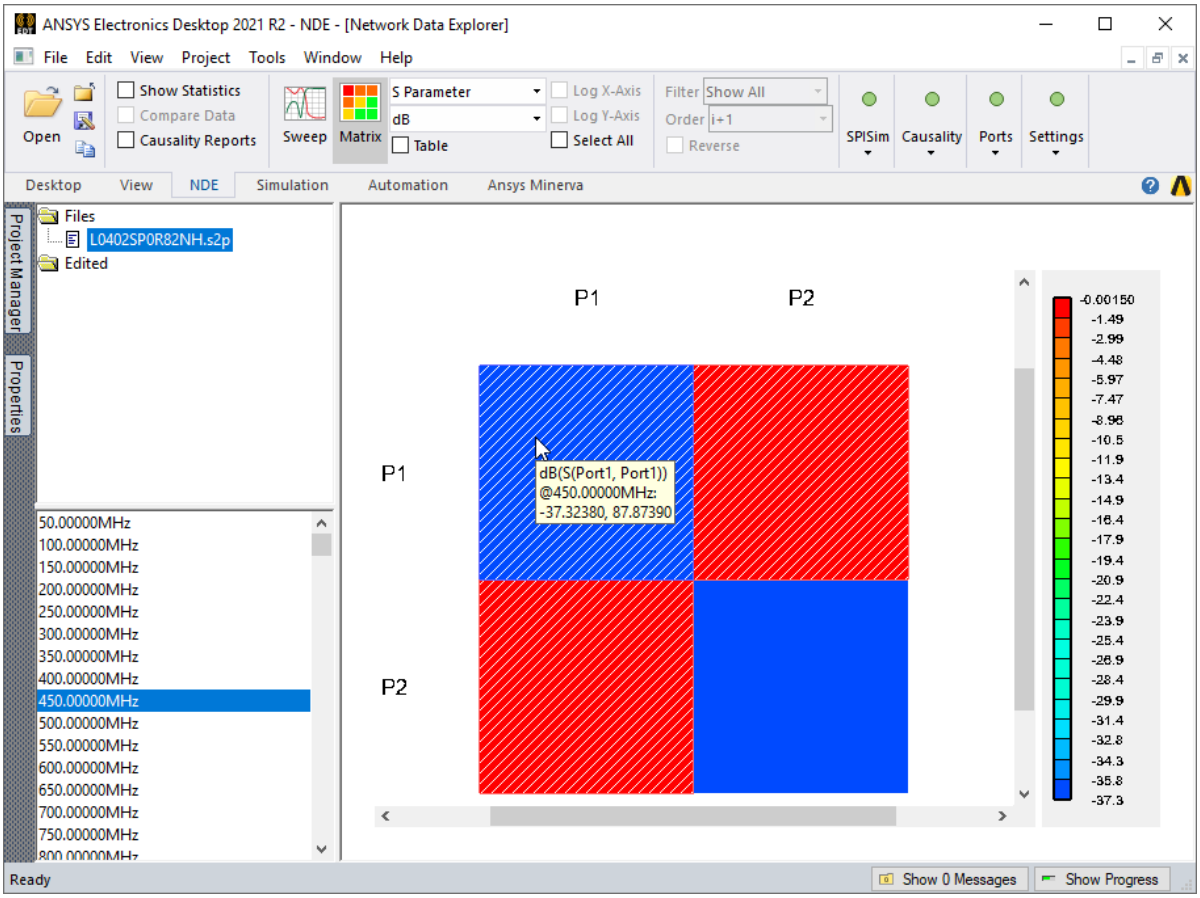
Double-clicking a cell switches to a matrix cell view, in which values for all frequencies for that cell are displayed. The double-clicked frequency is highlighted with solid red shading

Complex values are compared using their modulus. When multiple frequencies are selected, the data display depends on the Multiple Frequency Statistics setting.

Viewing a Color-Coded Matrix Plot

1. On the **NDE** ribbon, click **Matrix**.
2. Use the **Parameter** type drop-down menu to select **S parameter**, **Y parameter**, **Z parameter**, or another choice.
3. In the Cell and Frequency Selection pane, select the frequencies you want to plot or click the **Select All** check box on the **NDE** ribbon to select all frequencies.
4. Hover over a cell to see more information in a tool tip.
5. Right click to see a right click menu with other commands.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Matrix values display in a color-coded grid. If the selected **Format** is a complex value, only the real component is used to determine the display color. When multiple frequencies or variations are selected, the data display depends on the Multiple Frequency Statistics setting. Maximum values are highlighted in red. Minimum values are highlighted in dark blue.

Hover the cursor over any cell to view information about it.

Click any cell to select it. Selected cells appear with a pink outline. If Select Transpose is enabled, the transpose is selected in pink as well.

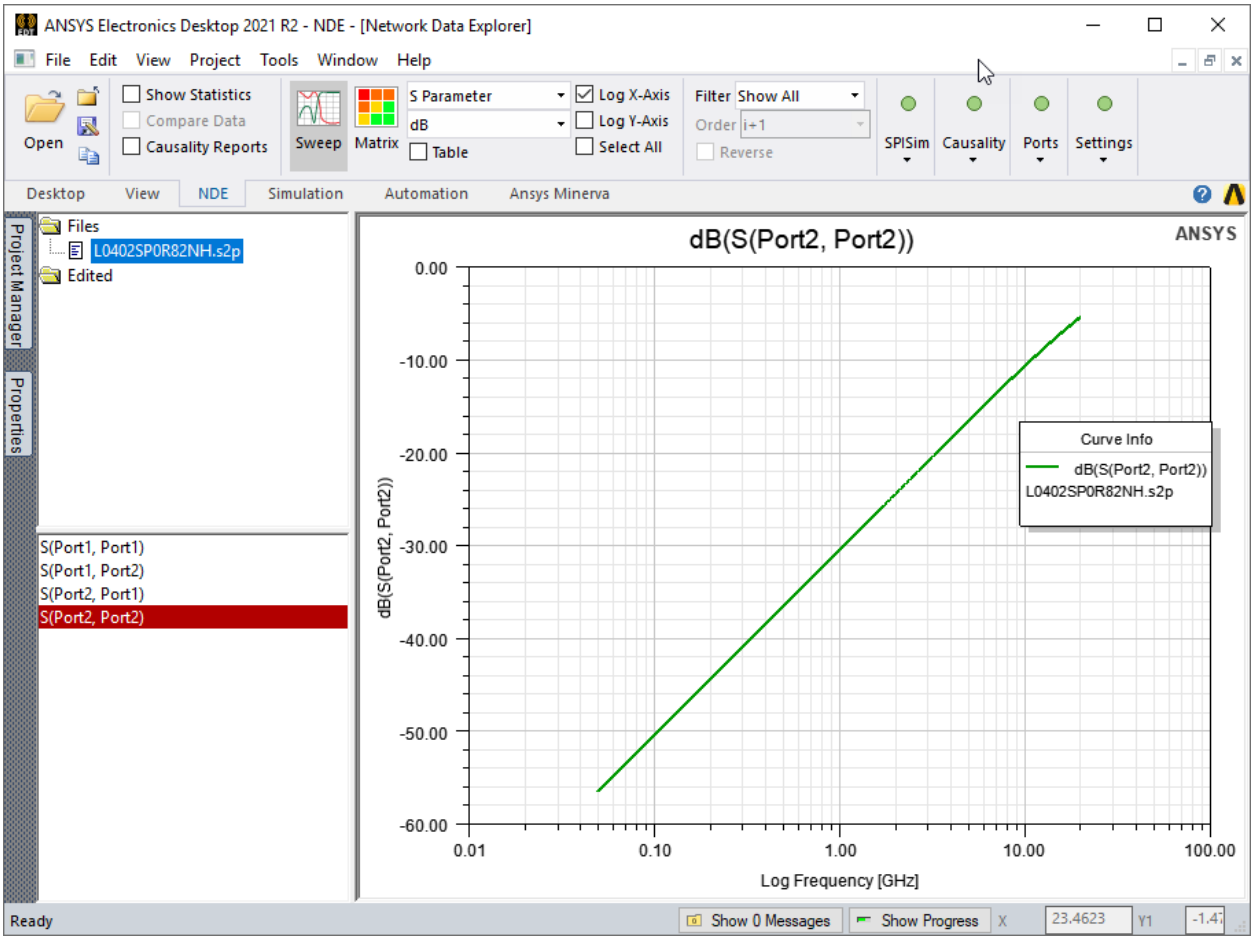
Double-click any cell to view a matrix cell plot in which all frequency values for that matrix cell are displayed as a graph.

Displaying a Cell Graph Across All Frequencies

Network Data Explorer can plot a cell across all frequencies.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. On the **NDE** ribbon, click **Sweep**. The Data View Pane updates if necessary.
2. Use the **Parameter** type drop-down menu to select **S parameter**, **Y parameter**, or **Z parameter**.
3. In the Cell and Frequency Selection pane, select cells to display or click the **Select All** check box on the **NDE** ribbon to select all cells.
4. To add a log scale to the X-axis, click **Log X-Axis** on the **NDE** ribbon.



5. To add a log scale to the Y-axis, click **Log Y-Axis** on the **NDE** ribbon.
6. Right click to see a right click menu with other commands.

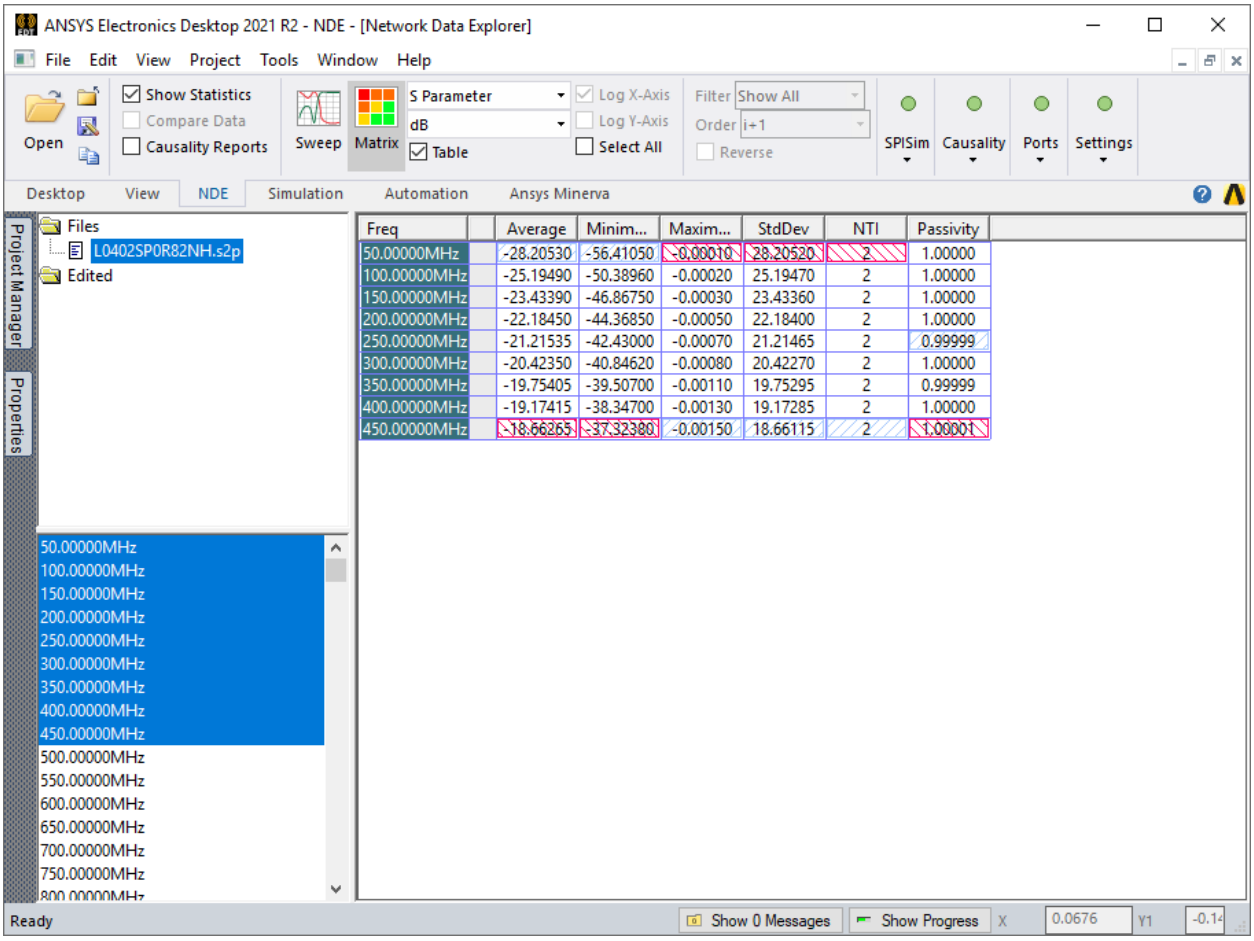
Displaying Matrix Statistics by Frequency

Network Data Explorer can display various statistical measurements.

1. On the **NDE** ribbon:
   a. Click **Matrix**.
   b. Click **Table**.
   c. Click **Show Statistics**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the Cell and Frequency Selection pane, select frequencies to display.



3. Click a column header to sort data by that column.
4. Hover the cursor over a cell to view information about it.

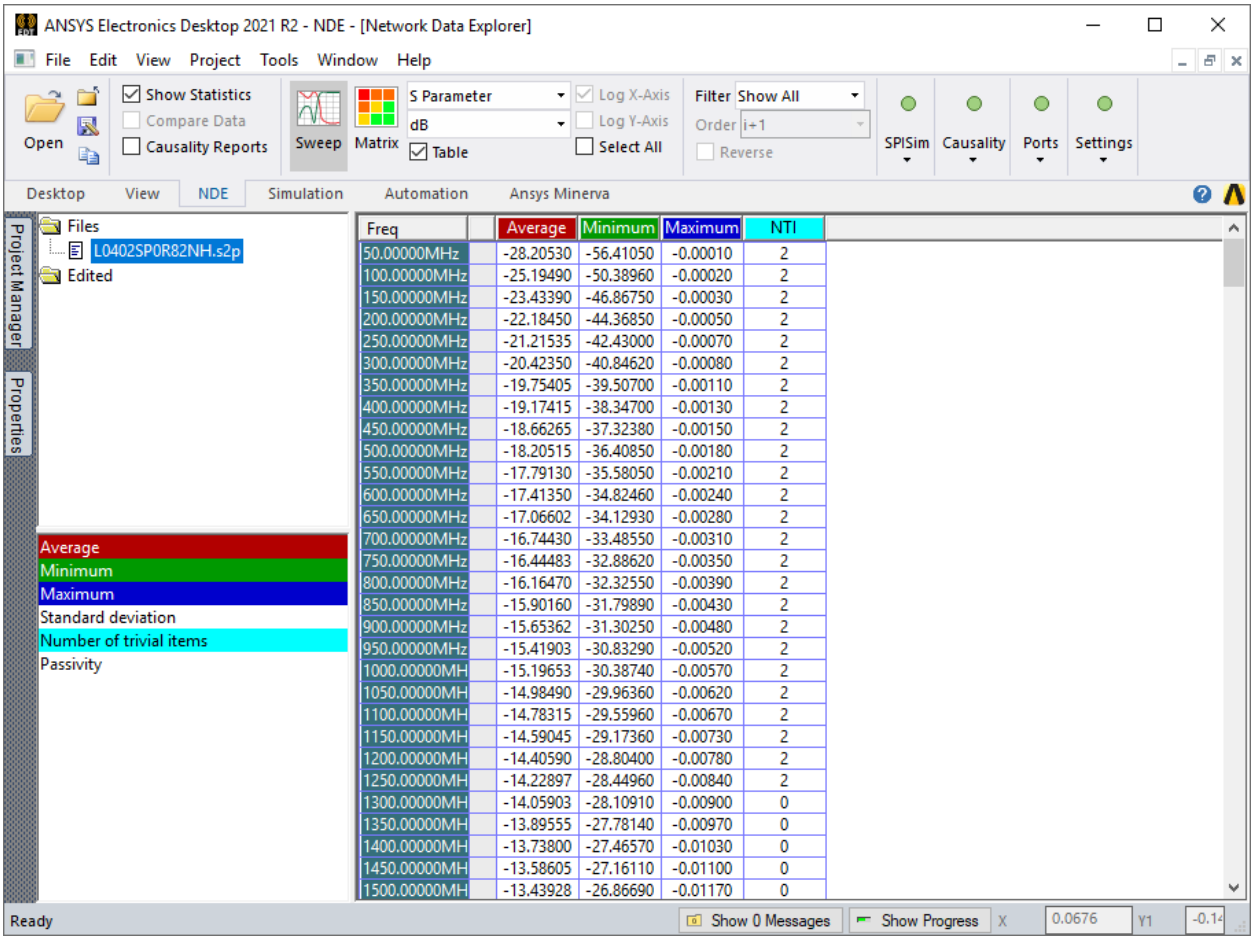
Only real (not complex) data formats are offered for statistical analysis. **Passivity** is only available for S-parameter data (comparisons inactive). **NTI** refers to the number of trivial items; for S-parameters, this includes all zeros and ones; for all other data (and data comparisons), only zeros are counted as trivial. The minimum value for each column is highlighted in blue; the maximum is highlighted in red.

Displaying Individual Statistics for All Frequencies

1. On the **NDE** ribbon:
    a. Click **Sweep**.
    b. Click **Table**.
    c. Click **Show Statistics**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the Cell and Frequency Selection pane, select statistics to display. The information displays in a table in the Data View pane.



Selected statistics are displayed for all frequencies.

Passivity is only available for S-parameters (comparisons inactive).

For S-parameters, the **Number of Trivial Items (NTI)** includes all values of 0 and 1. For other data and data comparisons, only values of 0 are counted as trivial.

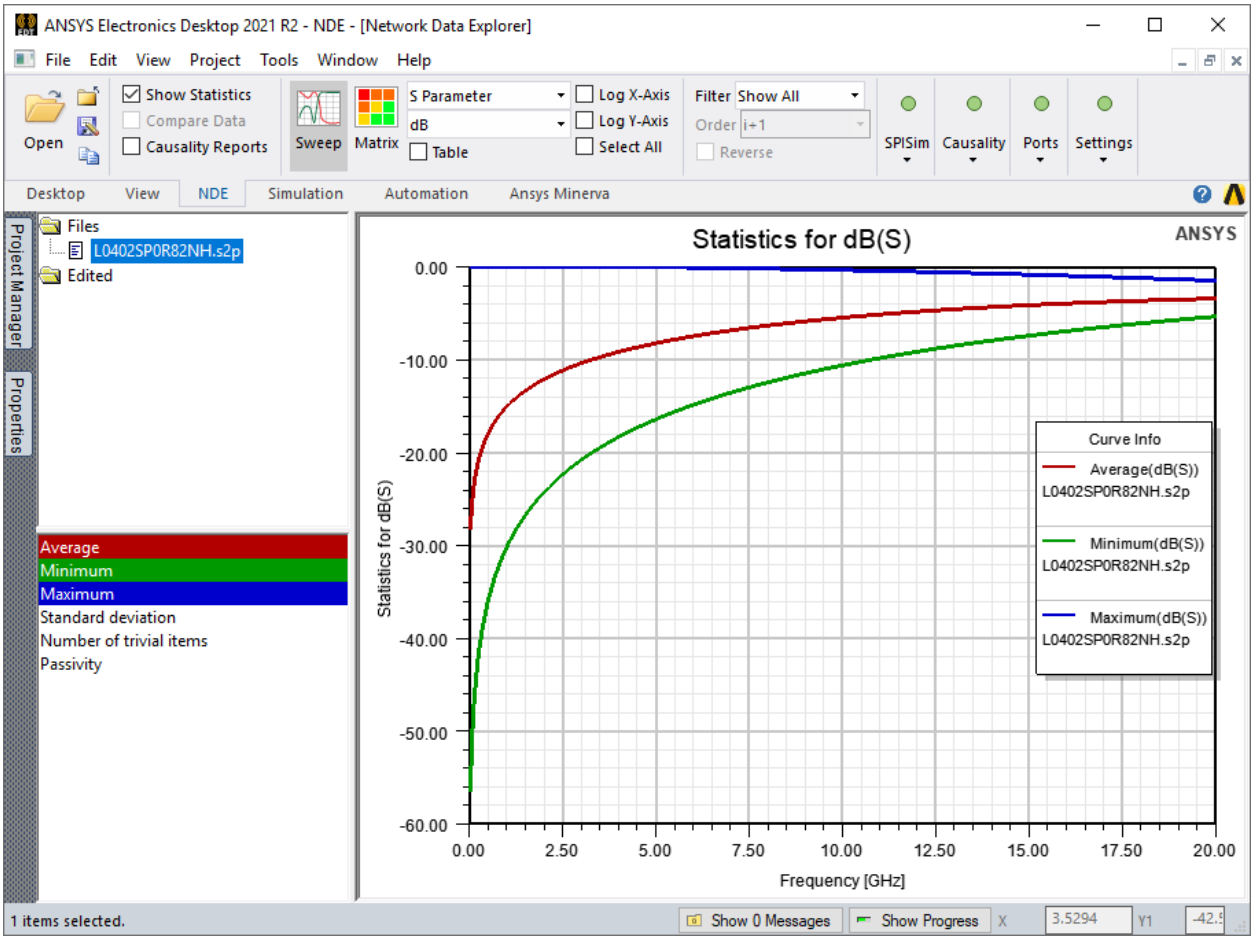
Creating a Statistics Plot

Network Data Explorer can display a graph of selected statistical measures across all frequencies.

1. On the **NDE** ribbon:
    a. Click **Sweep**.
    b. Click **Show Statistics**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the Cell and Frequency Selection pane, select statistics to display. The selected statistics are plotted.



3. Hover the cursor over a statistic to view more information about it.

Passivity is only available for S-parameters (comparisons inactive).

For S-parameters, the **Number of Trivial Items (NTI)** includes all values of 0 and 1. For other data and data comparisons, only values of 0 are counted as trivial.

Comparing Network Data

Network Data Explorer can compare variations for two network data sets that are the same size.

1. On the **NDE** ribbon, click **Sweep**.
2. In the Network Data Selection pane, select exactly two data sets.
3. On the **NDE** ribbon, click **Compare Data**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. In the Cell and Frequency Selection pane, select which cells you want to compare . For each value along the X-axis, the Y-axis values are subtracted, one from the other, to create the comparison plot. The second selected data is subtracted from the first selected data.
5. Optionally, check **Show Statistics** to show values applied to all cells and all frequencies

It is not possible to compare a data set against itself *unless the data set has been cloned.* Then, you can compare the original data set to the clone.

Traces for a given cell or statistical measure are displayed for all data sets; you can use tool tips to distinguish between them.

If a single cell or statistical measure is displayed, different colors are used for each data trace. If multiple cells or statistical measures are selected, a single color is used for all data traces for each cell or statistical measure.

In a data comparison, traces are shown for all selected data sets. This is true for both cell and statistical traces.

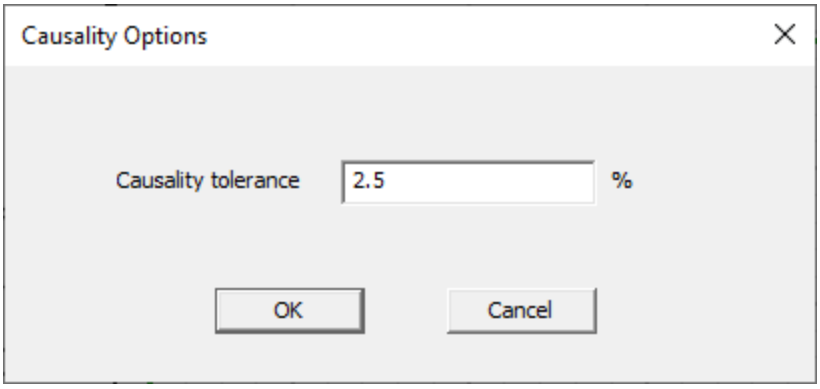
If **dB** format is shown, you can either subtract values before or after applying the dB function. See Display Format to make this choice.

Causality Checking and Plots

Network Data Explorer can perform a causality check on S-parameter data from any source (solution or file), and provide plots of the results in various formats.

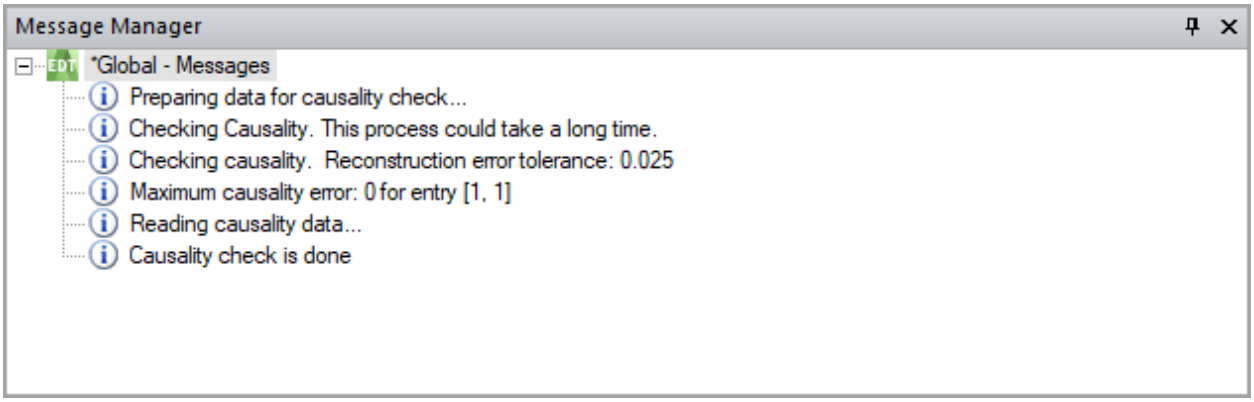
When S-parameter data is loaded into Network Data Explorer, the **Causality** button is enabled.

1. On the **NDE** ribbon, click **Causality**. The **Causality Options** dialog opens.



2. Enter a **Causality tolerance** and click **OK** to start the causality check. Depending on the size of the S-parameter data, the causality check may take several minutes to complete. The check's status displays in the Message Manager.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

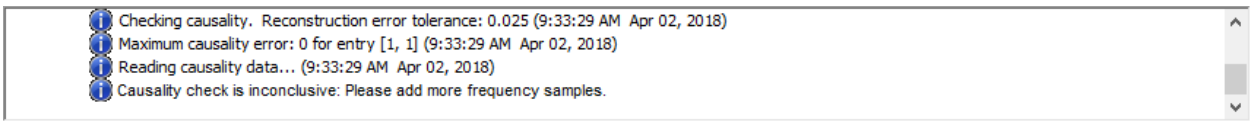


When the check completes, the Message Manager updates to display a summary of results.

Reconstruction Error Tolerance – Causality of a frequency response is determined by calculating the generalized Hilbert transform of the data at all frequencies. A causal frequency response is equal to its generalized Hilbert transform. The reconstruction error is the difference between the tabulated data and its transform at a given frequency. The message shows the maximum reconstruction error tolerance for a causal frequency response. The default tolerance of 0.01 is equal to the state-space fitting tolerance.

Maximum Causality Error – The maximum causality error for all port pairs and all frequencies, along with the matrix indices (port numbers) where the maximum noncausality occurs. A noncausal response is one where all matrix entries can be conclusively analyzed, and at least one entry exceeds the causality tolerance. The maximum reconstruction appears first, followed by port numbers in brackets (e.g., [port number, port number]). When all results are conclusive but no matrix reconstruction error exceeds the tolerance, the maximum causality error is reported as zero, and no matrix entry is listed.

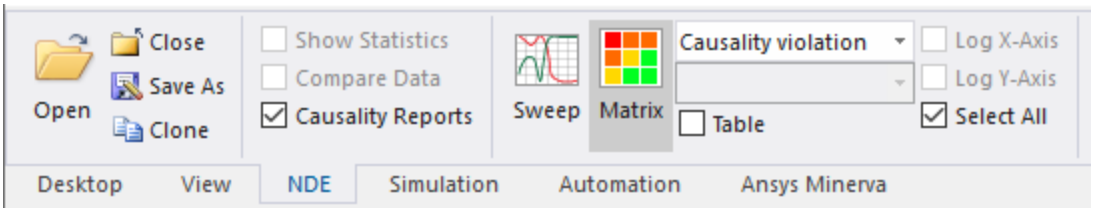
If the data does not contain enough frequency points to determine whether the data is or is not causal, the Messages Pane will note an inconclusive result. Network Data Explorer will also report the data set as inconclusive if any cells are inconclusive, even if other entries exhibit causality violations.



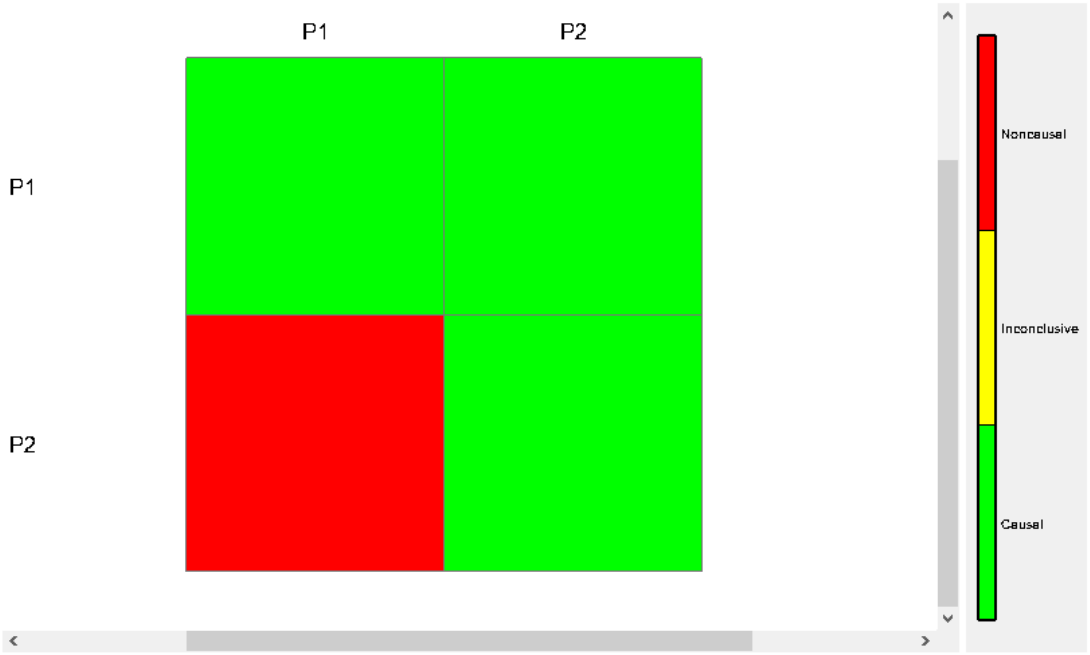
To see the plotted results of the causality check:

1. On the **NDE** ribbon, click **Causality Reports**.
2. Either click the **Matrix** icon or select **Causality violation** in the **Parameter** type drop-down menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A rectangular plot displays, with dimension N x N and color-coding to indicate the causality status of each port pairing.



In this plot, the cells go from (Port 1, Port 1) at the upper-left area to (Port N, Port N) at the lower-right corner. The result shows the causality over all frequencies in the data. In this example, the matrix is asymmetric, so that S12 is noncausal, while S11, S21, and S22 are causal.

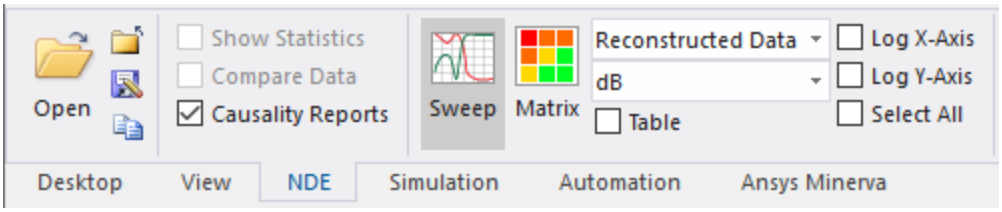
To see the details for each frequency, click the **Table** check button on the **NDE** ribbon.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Freq		Port1	Port2	Port1	Port2
0.10000GHz	Port1	0.01715, 0.00000	0.00050, 0.00000	0.01493, 0.00000	0.00098, 0.00000
	Port2	0.53111, 0.00000	0.00579, 0.00000	0.11471, 0.00000	0.00832, 0.00000
0.20000GHz	Port1	0.01367, 0.00000	0.00059, 0.00000	0.00736, 0.00000	0.00052, 0.00000
	Port2	0.27237, 0.00000	0.00394, 0.00000	0.05933, 0.00000	0.00423, 0.00000
0.30000GHz	Port1	0.01362, 0.00000	0.00105, 0.00000	0.00344, 0.00000	0.00028, 0.00000
	Port2	0.24509, 0.00000	0.00239, 0.00000	0.01921, 0.00000	0.00195, 0.00000
0.40000GHz	Port1	0.00907, 0.00000	0.00103, 0.00000	0.00171, 0.00000	0.00006, 0.00000
	Port2	0.16665, 0.00000	0.00163, 0.00000	0.01530, 0.00000	0.00084, 0.00000
0.50000GHz	Port1	0.00693, 0.00000	0.00062, 0.00000	0.00075, 0.00000	0.00006, 0.00000
	Port2	0.11877, 0.00000	0.00106, 0.00000	0.00542, 0.00000	0.00040, 0.00000
0.60000GHz	Port1	0.00295, 0.00000	0.00044, 0.00000	0.00047, 0.00000	0.00005, 0.00000
	Port2	0.05535, 0.00000	0.00042, 0.00000	0.00462, 0.00000	0.00015, 0.00000
0.70000GHz	Port1	0.00015, 0.00000	0.00009, 0.00000	0.00029, 0.00000	0.00003, 0.00000
	Port2	0.00510, 0.00000	0.00012, 0.00000	0.00114, 0.00000	0.00008, 0.00000
0.80000GHz	Port1	0.00058, 0.00000	0.00068, 0.00000	0.00047, 0.00000	0.00004, 0.00000
	Port2	0.02772, 0.00000	0.00088, 0.00000	0.00115, 0.00000	0.00010, 0.00000
0.90000GHz	Port1	0.00210, 0.00000	0.00081, 0.00000	0.00040, 0.00000	0.00013, 0.00000
	Port2	0.05492, 0.00000	0.00133, 0.00000	0.00173, 0.00000	0.00010, 0.00000
1.00000GHz	Port1	0.00221, 0.00000	0.00064, 0.00000	0.00033, 0.00000	0.00010, 0.00000
	Port2	0.07703, 0.00000	0.00142, 0.00000	0.00111, 0.00000	0.00009, 0.00000
1.10000GHz	Port1	0.00410, 0.00000	0.00045, 0.00000	0.00037, 0.00000	0.00008, 0.00000
	Port2	0.08851, 0.00000	0.00151, 0.00000	0.00137, 0.00000	0.00011, 0.00000
1.20000GHz	Port1	0.00433, 0.00000	0.00078, 0.00000	0.00017, 0.00000	0.00013, 0.00000
	Port2	0.09013, 0.00000	0.00100, 0.00000	0.00168, 0.00000	0.00017, 0.00000
1.30000GHz	Port1	0.00377, 0.00000	0.00072, 0.00000	0.00020, 0.00000	0.00018, 0.00000
	Port2	0.09487, 0.00000	0.00151, 0.00000	0.00081, 0.00000	0.00012, 0.00000
1.40000GHz	Port1	0.00380, 0.00000	0.00035, 0.00000	0.00020, 0.00000	0.00014, 0.00000
	Port2	0.08918, 0.00000	0.00136, 0.00000	0.00139, 0.00000	0.00009, 0.00000
1.50000GHz	Port1	0.00417, 0.00000	0.00005, 0.00000	0.00012, 0.00000	0.00011, 0.00000
	Port2	0.08091, 0.00000	0.00174, 0.00000	0.00094, 0.00000	0.00021, 0.00000
1.60000GHz	Port1	0.00315, 0.00000	0.00038, 0.00000	0.00006, 0.00000	0.00015, 0.00000
	Port2	0.07512, 0.00000	0.00208, 0.00000	0.00065, 0.00000	0.00011, 0.00000
1.70000GHz	Port1	0.00275, 0.00000	0.00034, 0.00000	0.00006, 0.00000	0.00021, 0.00000
	Port2	0.06069, 0.00000	0.00152, 0.00000	0.00101, 0.00000	0.00021, 0.00000

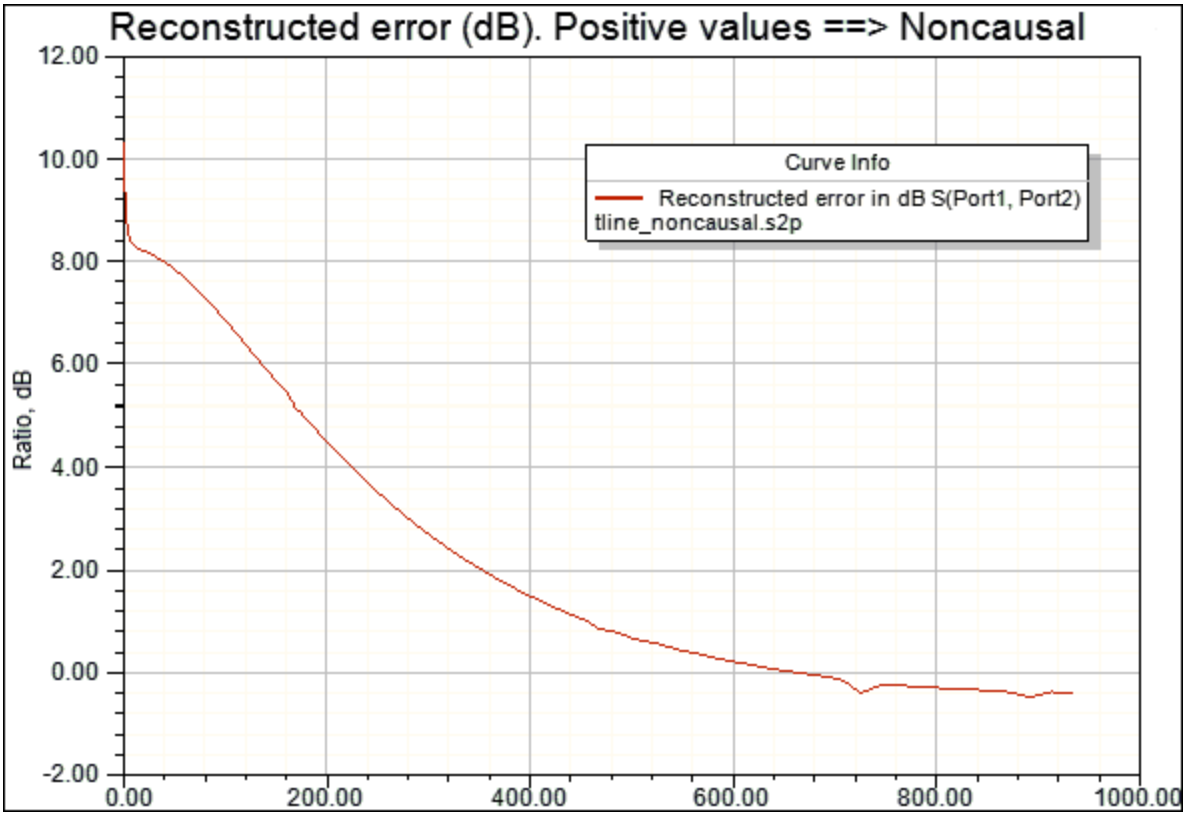
To plot the reconstructed frequency response generated by the causality checker:

1. On the **NDE** ribbon, click **Causality Reports**.
2. Either click the **Sweep** icon or select **Reconstructed Data** in the **Parameter** type drop-down menu.
3. In the **Format** drop-down menu, select the desired format. **dB** is selected by default.



The plot appears, showing the reconstruction error at each frequency divided by the tolerance.

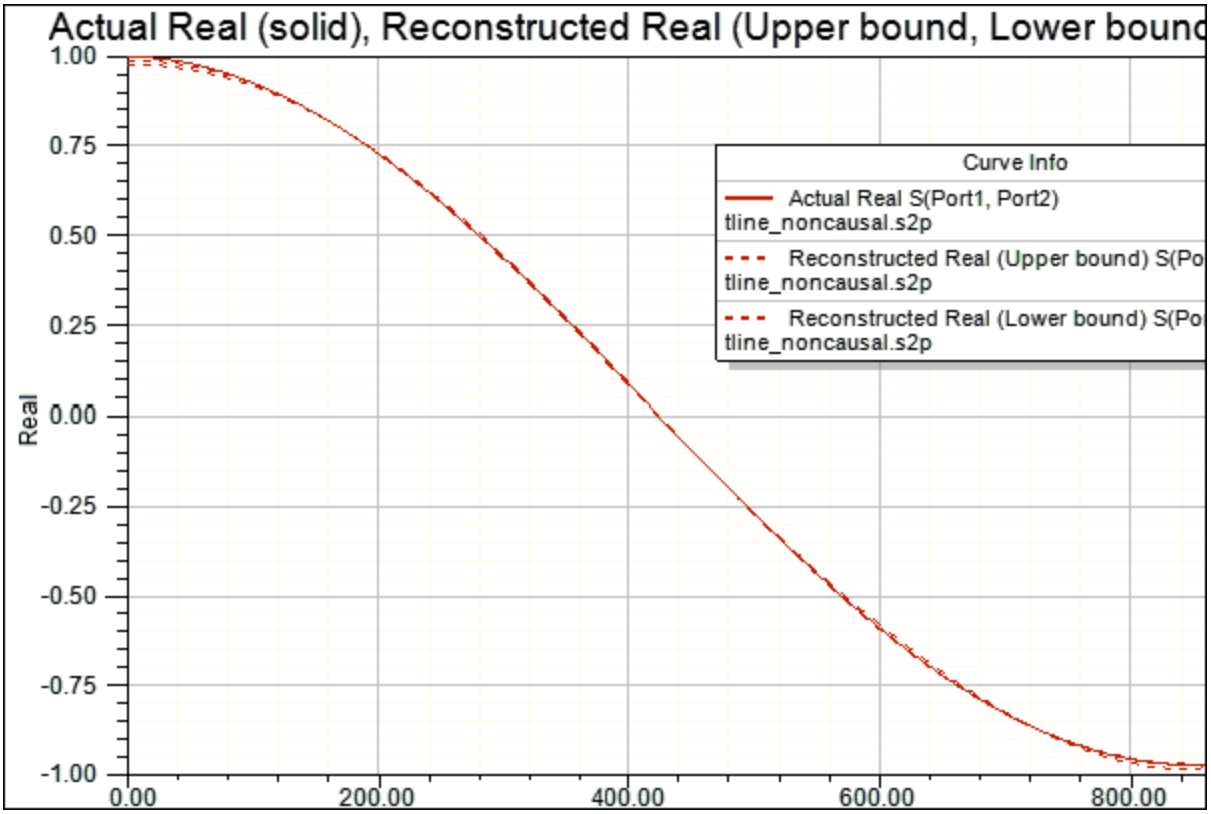
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The reconstruction error ratio for parameter S12 is positive for frequencies less than about 680MHz, indicating a broad range of noncausal behavior.

4. To compare the real part of the reconstructed data to the real part of the actual data, set the **Format** to **Real**.

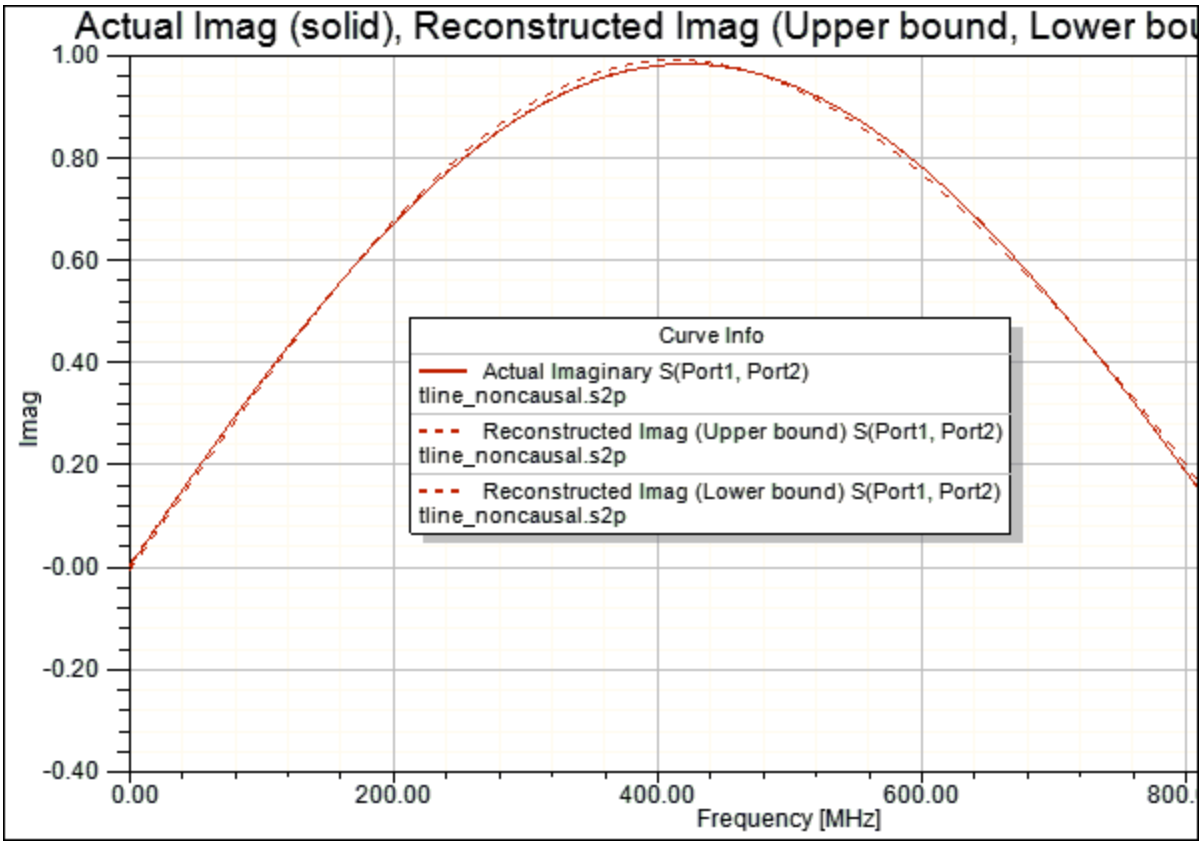
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For a causal frequency response, the actual data (solid line) will be within the upper and lower bounds of the reconstruction (dotted lines) at all frequencies.

5. To compare the imaginary part of the reconstructed data to the imaginary part of the actual data, set the **Format** to **Imaginary**.

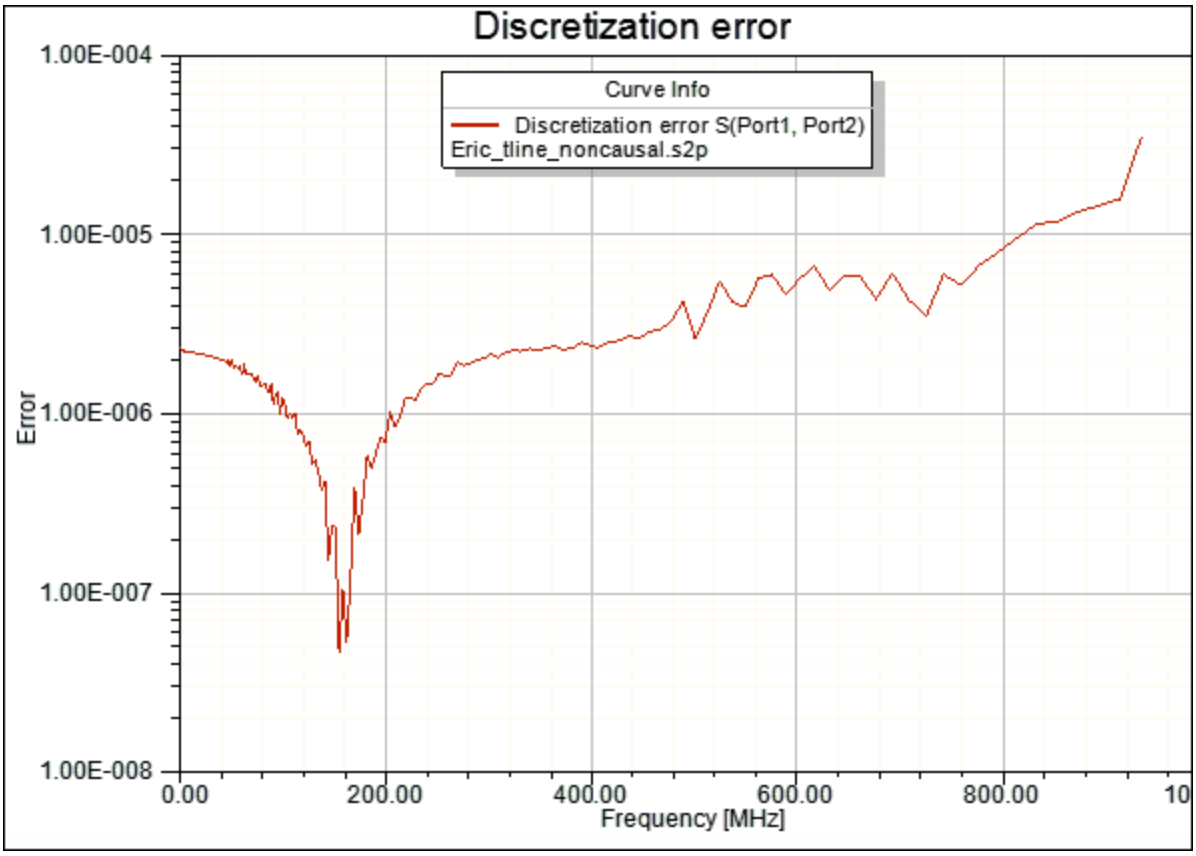
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For a causal frequency response, the actual data (solid line) will be within the upper and lower bounds of the reconstruction (dotted lines) at all frequencies.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. To view the frequency-dependent discretization error, set the **Format** to **Discretization**.



The discretization error is the error that is due to the fact that the data are available only at discrete frequencies rather than for a continuous spectrum. A discretization error near or greater than the causality tolerance renders the causality check inconclusive. Data at more frequencies could reduce the discretization error and render the analysis conclusive. This set of data exhibits low discretization errors (<<0.01) at all frequencies, and the causality check is conclusive (conclusively noncausal in this example).

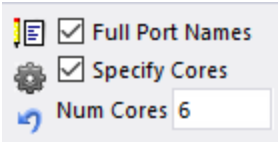
Multithreading

By default, multithreading (execution on multiple cores) is enabled for Causality Checking and Macro Model export. Multithreading saves significant time in the Causality Check calculation, and improves the time for other state-space fitting operations. See Technical Notes.

The **Cores** field in the Network Data Explorer Control Pane defaults to half the number of cores detected on your computer.

For best performance, disable hyper-threading on your computer. When hyper-threading is enabled, the number of cores includes the physical cores and an equal number of logical cores. With hyper-threading disabled, the display shows only half the number of physical cores.

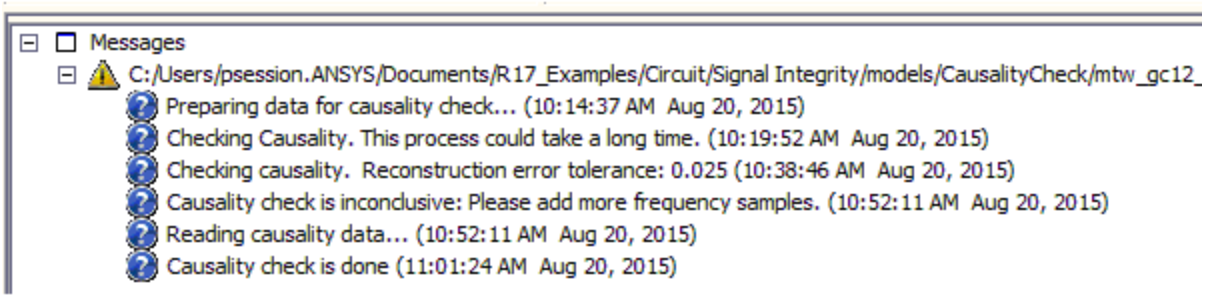
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To disable multithreading, uncheck the check box.

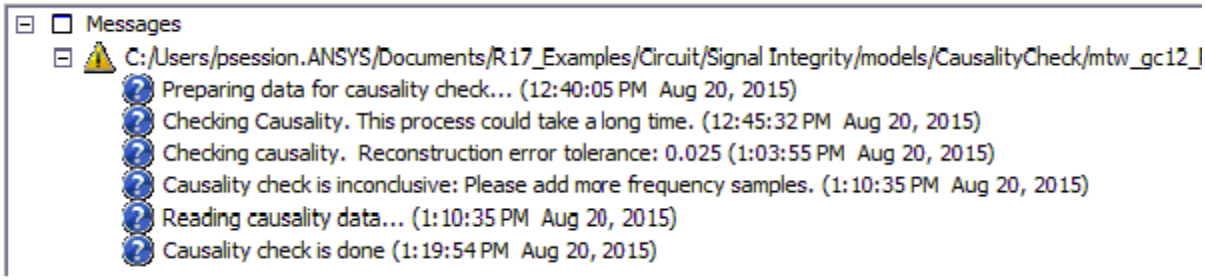
Multithreading Technical Notes

Multithreading can save significant time in the causality check calculation, especially for data sets with large number of ports or a large number of frequencies. The Network Data Explorer contributes a fixed amount of overhead time in preparing the data, and this overhead is not reduced by multithreading. Here are the messages from the causality check of a 278-port Touchstone file using two cores.



The causality check itself happens between the message `Checking causality. Reconstruction error tolerance: 0.025 (time 10:38:46)` and the message `Causality check is inconclusive: Please add more frequency samples (time 10:52:11)`. The difference between these two times is the time for the causality check after setup, 13:25 or thirteen minutes and twenty-five seconds. There is a fixed overhead of around 25 minutes involved in the overall time duration for this example.

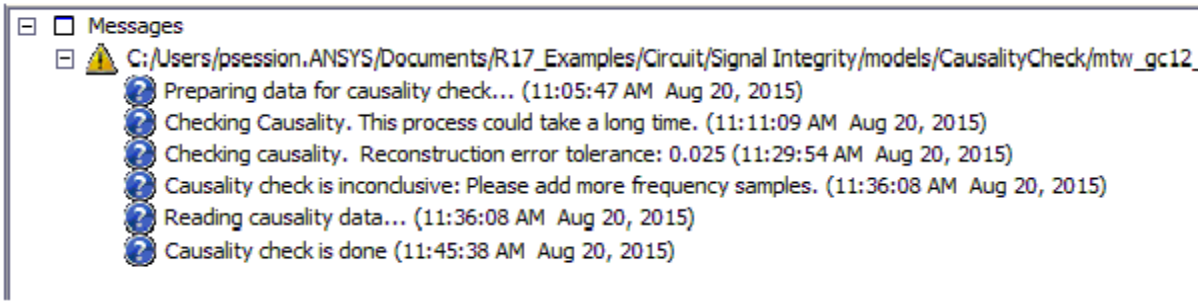
Here are the causality check messages for the same 278-port file using eight cores:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

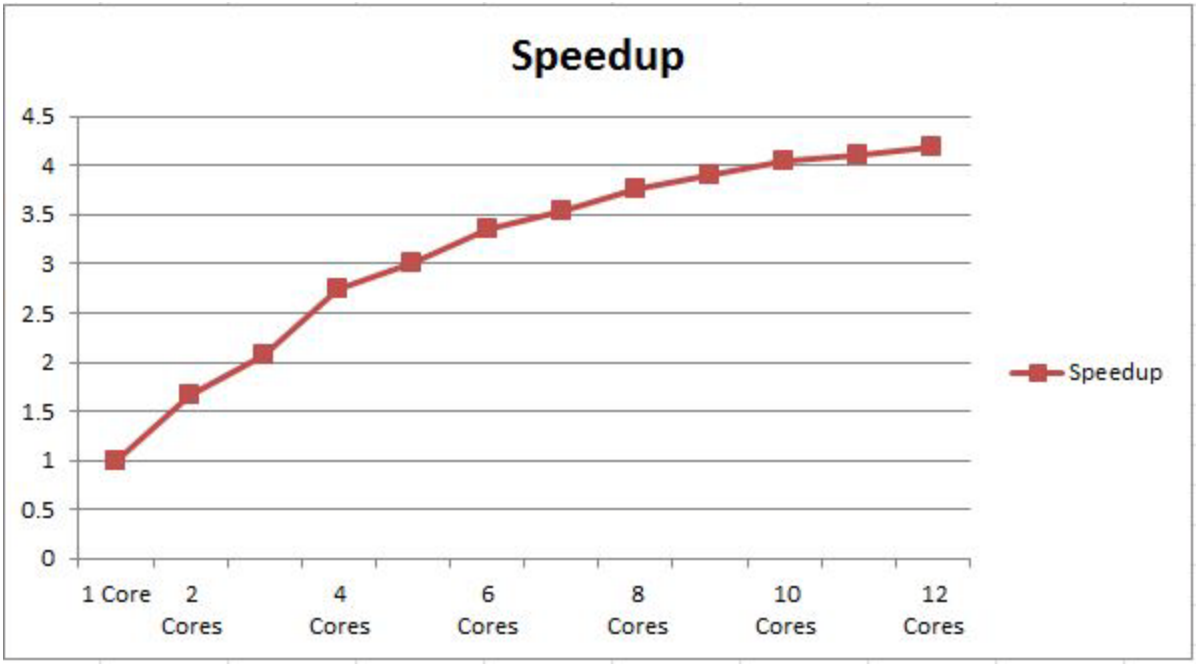
Now the causality check happens between times (1:03:55) and (1:10:35), a difference of (6:40) or six minutes and forty seconds, about half the time for the two-core example.

The speedup from multithreading is not linear in practice. Here are the messages from the same 278-port file using sixteen cores:



Now the causality check happens between times (11:29:54) and (11:36:08), a difference of (6:14) or six minutes and fourteen seconds, not significantly better than the eight-core performance.

Here is a graph showing the speedup of the causality checker using multithreading on this 278-port file. The plot is generated using data from runs with 1, 2, 4, 6, 8, 10, and 12 cores, then averaging the known times to approximate the speedup for 3, 5, 7, 9, and 11 cores. The speedup for N cores is the time with one core divided by the time with N cores.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24 - Postprocessing and Generating Reports

As Ansys Electronics Desktop completes a solution, you can display and analyze the results:

- View solution data including the following: convergence information, computing resources that were used during the solution process, mesh statistics, and matrices computed for the S-parameters, impedances, and propagation constants during each adaptive, non-adaptive, or sweep solution. For eigenmode solutions, you can view the real and imaginary parts of the frequency and quality factor Q computed for each eigenmode. Solution data can also be viewed while Ansys Electronics Desktop is generating a solution.
- View analysis results for Optimetrics solutions.
- Plot field overlays - representations of basic or derived field quantities - on surfaces or objects.
- Overlay Field Plots on Models
- Create 2D or 3D reports of S-parameters, basic and derived field quantities, and radiated field data.
- Plot the finite element mesh on surfaces or within 3D objects.
- Create animations of field quantities, the finite element mesh, and defined project variables.
- Scale an excitation's magnitude and modify its phase.
- Apply Derivative Tuning to Reports

> **Note:**
>
> Except in the case of non-model boxes drawn in the global coordinate system (CS), non-model objects cannot be used for any fields post processing operation You can use non-model boxes drawn in the global CS for post processing operations, including integration and solution domaining.

Viewing Solution Data

While Maxwell is generating a solution, or when it is complete, you can view the following types of information about the solution in the **Solutions** dialog box. The information actually available will vary with the solution type and setup of your design.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Convergence information

  > **Note:** Because adaptive mesh refinement is not performed for transient solutions, convergence data is not available for **Transient** analysis.

- Computing resources, or profile information, that were used during the solution process.
- Output Parameters (for example Force and Torque), and Matrix Data computed during each adaptive, non-adaptive, or sweep solution.
- Mesh data
- State of solved solutions
- Winding results (for AC magnetic solutions only)
- End Connection results (for AC magnetic solutions only)
- Loss results (for AC magnetic solutions only)

To access the **Solutions** dialog box, in which the information above can be viewed, do one of the following:

- Click **Maxwell 3D**, **Maxwell 2D,** or **RMxprt**, and then select **Results** > **Solution Data**.
- Right-click **Results** in the project tree, and then click **Solution Data** on the shortcut menu.

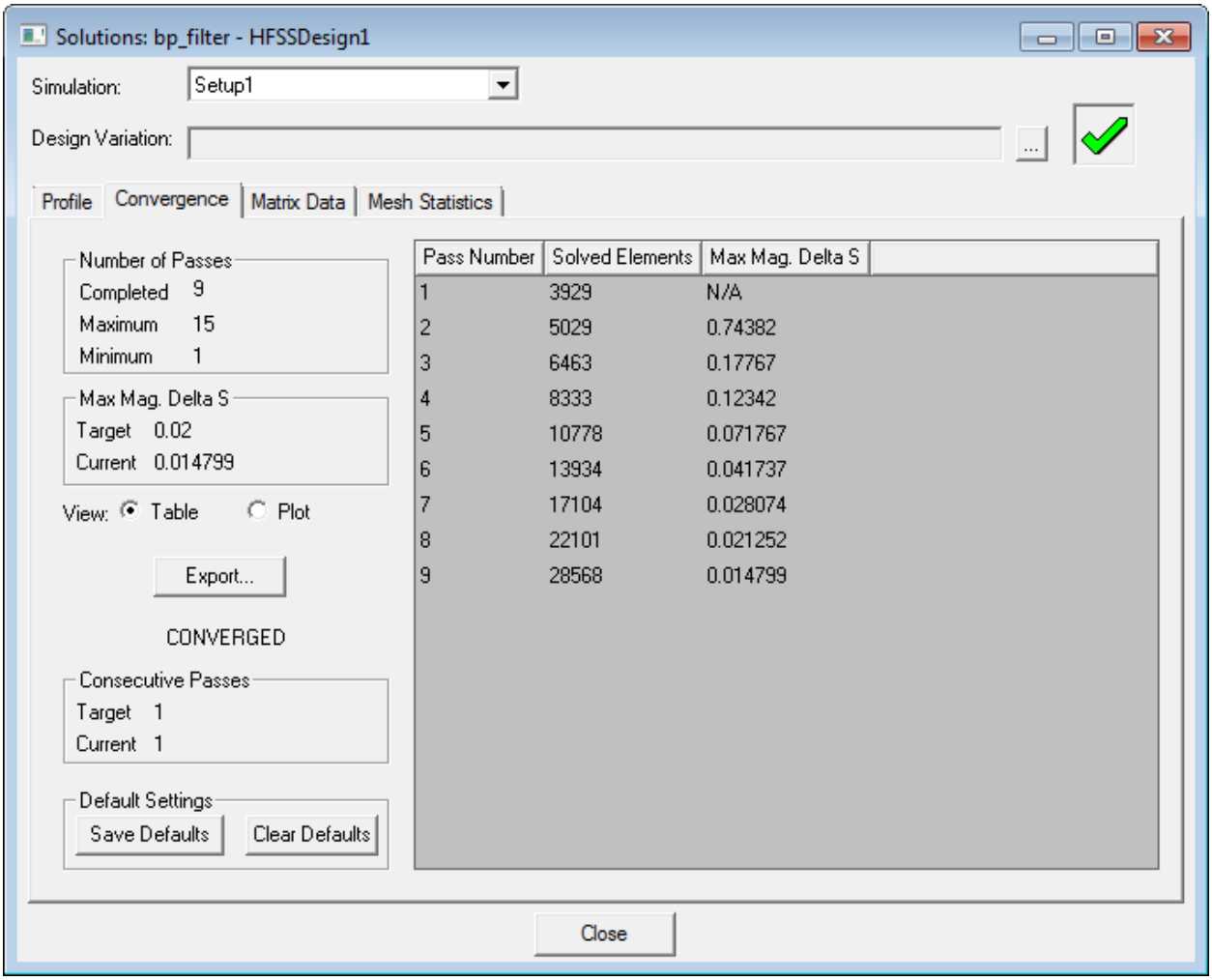
Viewing Convergence Data

To view an adaptive solution's convergence information, either during or after the solution process:

1. In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

   The **Solution Data** window appears. The **Convergence** tab is selected. You can also select the **Results** tab of the ribbon, click the **Solution Data** icon, and click the **Convergence** tab when the **Solution Data** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



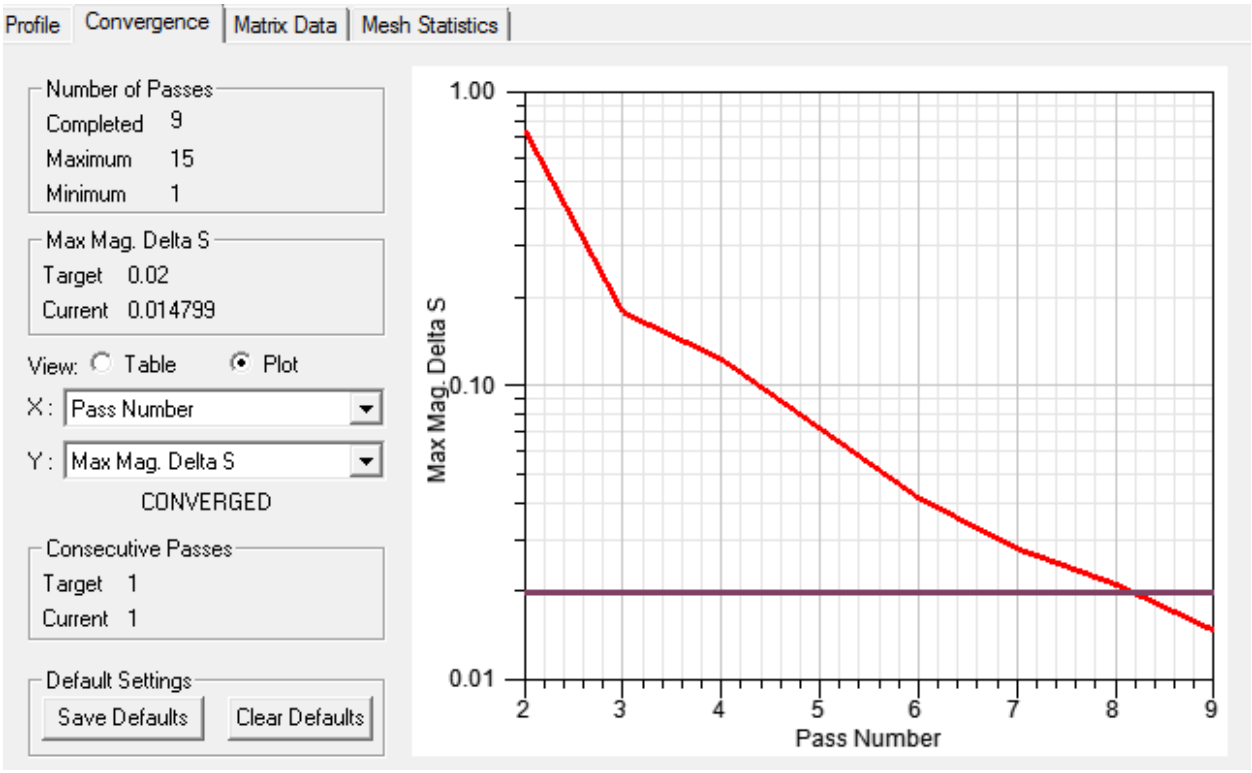
2. If you have run multiple simulations, from the **Simulation** list, select the solution setup for which you want to view convergence data.

   By default, the most recently solved solution is selected.

3. Under the **Convergence** tab, depending on your design setup, you can review the following convergence data:
   - Whether the solution is converged or not converged.
   - Number of adaptive passes completed and remaining.
   - The number of tetrahedra or triangles created at each adaptive pass.
   - Solution type dependent parameters such as Loss, Total Loss, Loss Error.
   - Max Delta (Mag S)
   - Max Delta (Phase S)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Select **Table** to display the convergence data in table format or **Plot** to plot the convergence data on a rectangular (X - Y) plot.



5. If you select **Save Defaults**, the settings for Table vs. Plot, and for Plot, the X and Y axis selections, those settings apply, based on the design-type, each time you open the desktop. You can also select **Clear Defaults** to reset to the original settings.

> **Note:**
>
> In the case of the 3D AC Conduction solver, even though the convergence columns names refer to the loss as a solution criteria, the convergence parameter considered is the magnitude of the apparent power.

Viewing the Number of Completed Passes

At any time during the solution process, you can view the number of adaptive passes (solve — error analysis — refine cycles) that have been completed and that have yet to be completed. When the solution is complete, you can view the number of adaptive passes that were performed. If the solution converged within the specified stopping criteria, fewer passes than requested may have been performed.

To view the number of passes:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

The **Solution Data** window appears. The **Convergence** tab is selected.

The number of completed and remaining passes is listed in the **Number of Passes** area.

Viewing Output Variable Convergence

At any time during or after the solution process, you can view the values of the output variable.

To view the output variable convergence:

- In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

The **Solutions** window appears. The **Convergence** tab is selected.

The **Output Var** column lists the value of the output variable for each pass. The **Output Var. Delta (%)** column lists the delta change between the current and previous passes as a percentage of the previous pass value. If output variable convergence is not used, the columns are not used in the table.

Plotting Convergence Data

To display convergence data vs. pass on a rectangular (x - y) plot:

1. In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

   The **Solution Data** window appears. The **Convergence** tab is selected.

2. In the lower-left corner of the window, select **Plot** as the view type.
3. Select the data you want to plot on the x-axis from the **X** drop-down menu.
4. Select the data type you want to plot on the y-axis from the **Y** drop-down list.

The x -y plot appears in the view window.

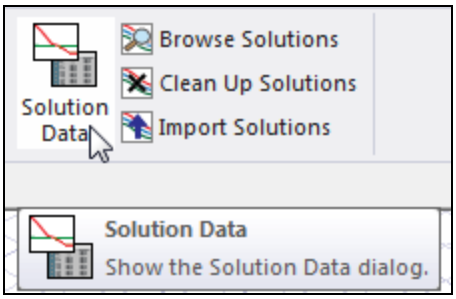
Viewing a Solution Profile

At any time during or after the solution process, you can examine the computing resources or profile data that were used by Ansys Electronics Desktop solvers during the analysis. The profile data is essentially a log of the tasks performed by Ansys Electronics Desktop during analysis. The log indicates the length of time each task took and how much physical and disk memory was required.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

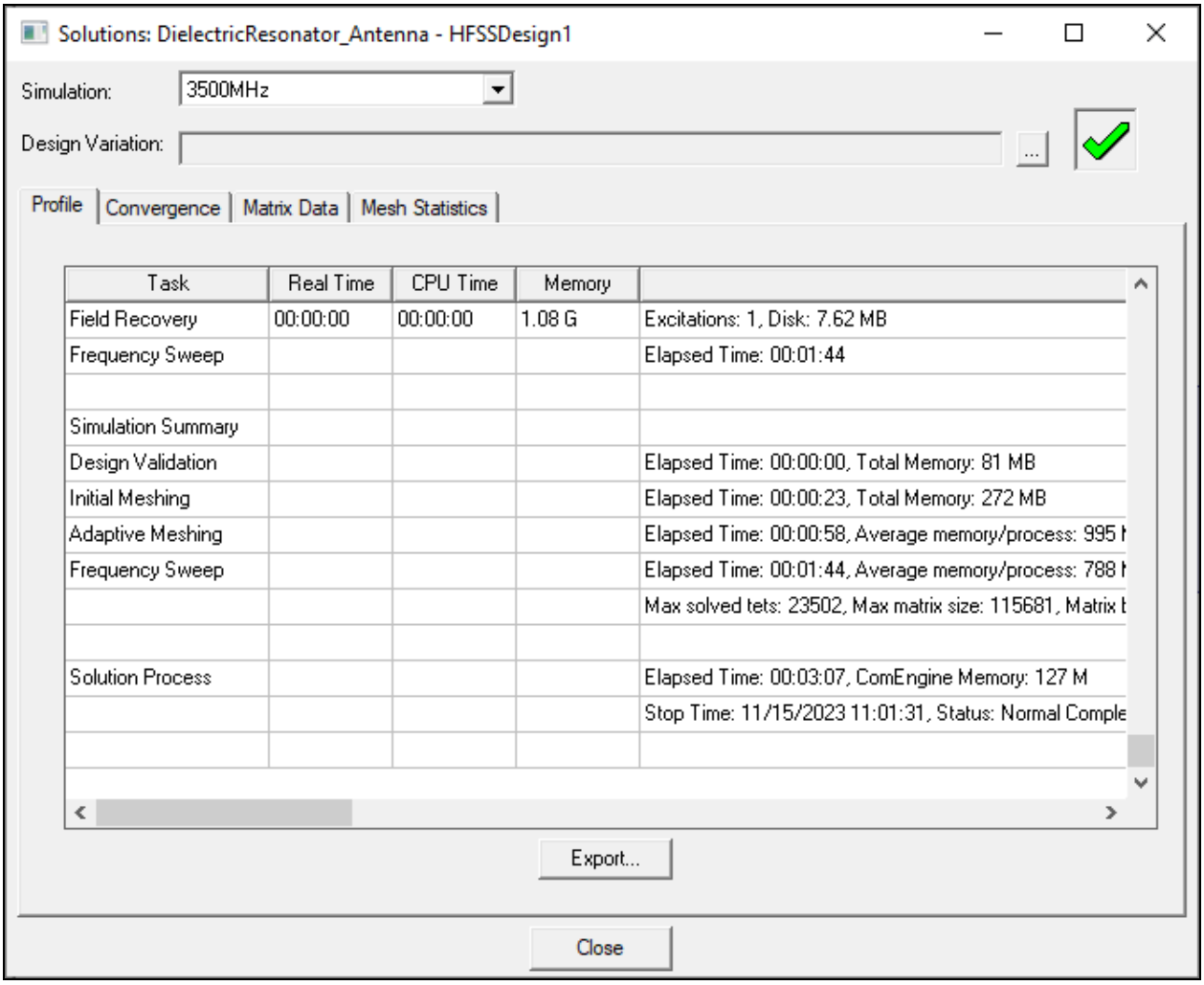
1. In the project tree, right-click a solution setup, and select **Profile**.

   The **Solutions** dialog box appears, with the **Profile** tab selected. You can also click the **Solution Data** icon on the ribbon to open the **Solutions** dialog box, and then select the **Profile** tab.



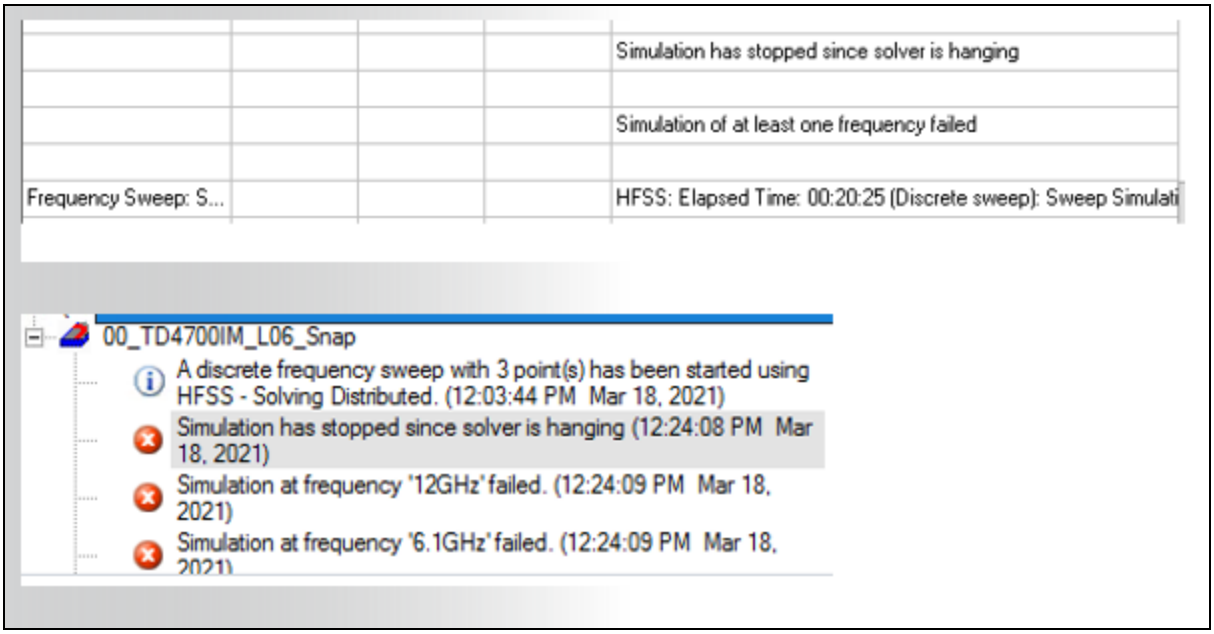
   The displayed data depends on the type of problem and solution setup.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If the simulation hangs, equivalent messages appear in the Message window and the Solution **Profile** tab.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. If one or more exist, you can select the desired profile information from the **Simulation** drop-down menu. In general, the information displayed includes:

Task	Lists the type of task that was performed. Tasks include Start, various Mesh tasks, Simulation Setup, Port Adaptation, Adaptive Pass tasks, including simulation setup, Matrix Assembly, Solver tasks, and Field Recovery, Sweep tasks, and Solution Process summary and Totals for time.
Real Time	The difference in time between the start of the task and the end of the task (elapsed time).
CPU Time	The amount of CPU time required to perform the task.
Memory	The peak amount of physical memory (RAM) used by the individual executable running the task. The memory is freed for other uses after each task is complete.
Information	General information about the solution (for example, the number of tetrahedra used in the mesh, disk use, solver information, sweep information, and totals).

The matrix solver writes specific information in some of these fields, as outlined below:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Task	The matrix solver task reports the type of solution performed by the solver, based on the physics of the problem. It has the form "Solver *pdsn*" (e.g., Solver MRS2, Solver DCS4-L2), where: • *p*, the precision type is: M (mixed for direct solver), or Mxx (for distributed direct solver), or D (double for direct and iterative solver) or E (extended precisions solve) • *d*, the matrix data type is: R (real) or C (complex) • *s*, the symmetry type is: S (symmetric), A (asymmetric), or H (hermitian) • *n*, the number of processors used. You specify the number of available processors on the local machine in the solver options. If a solve does not use all available processors (local or distributed), the number reported may be less than the number available. If a simulation uses the iterative solver, the Solver designation can include a level indicator appended. to an Iterative solver designation (L2 in the example above). The higher the Level number the lower the memory, you will never see L1 (this would be equivalent to direct solver. And a first order solve will only display L2 since it only has one level of order to go down for preconditioning. A second or mixed order solve may display L3 depending on the mesh quality. If the solver switches from the Iterative Solver to the Matrix solver, you see a Matrix solver warning: Switch from Iterative Solver to Direct Solver.
Information	The matrix solver information line includes, for example, Disk = 0 KBytes, matrix size 11137 , matrix bandwidth 20.3. • **Disk** – The amount of hard disk space used during the calculation of the matrix solution. If the disk usage for matrix solver is non-zero in profile, it usually indicates off-core matrix solver. If the matrix solver must solve off-core, smaller blocks of the data to be solved are created on disk, each block is then solved in physical memory, and then the matrix solution is reassembled. As a result of this additional processing, the time required to calculate a solution is higher. • **Matrix size** – The size of the matrix that was solved (the number of unknowns) • **Matrix bandwidth** – An FEM matrix is a sparse matrix. The solver only stores the non-zero entries. The matrix bandwidth is the average number of non-zeros per row. It gives an idea of the sparsity of a FEM matrix.Storage for the sparse matrix is proportional to the total number of nonzeros = #rows x bandwidth. The higher the bases order, the larger the bandwidth. • **Iterations** – For the Iterative Solver.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To export the profile data:

1. Open the **Solutions** dialog box and select the **Profile** tab.
2. Click **Export Profile**.

   This opens a file save dialog box that lets you provide a file name and location.

3. Click **Save**.

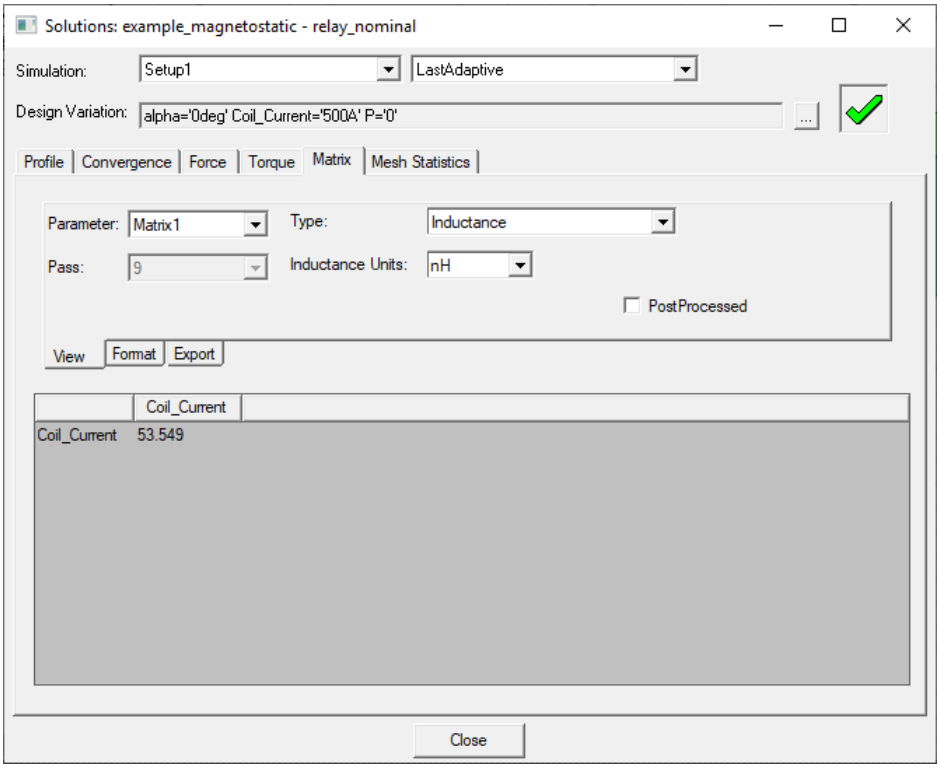
   The data is saved in a text file with a .prof extension.

Viewing Matrix Data

To view matrix parameter data:

1. In the Project Manager tree, right-click the solution setup of interest, and then click **Solutions** on the shortcut menu.

   The **Solutions** window appears. Select the **Matrix** tab. A typical example is shown below.



2. The **Design Variation** field shows the current design.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

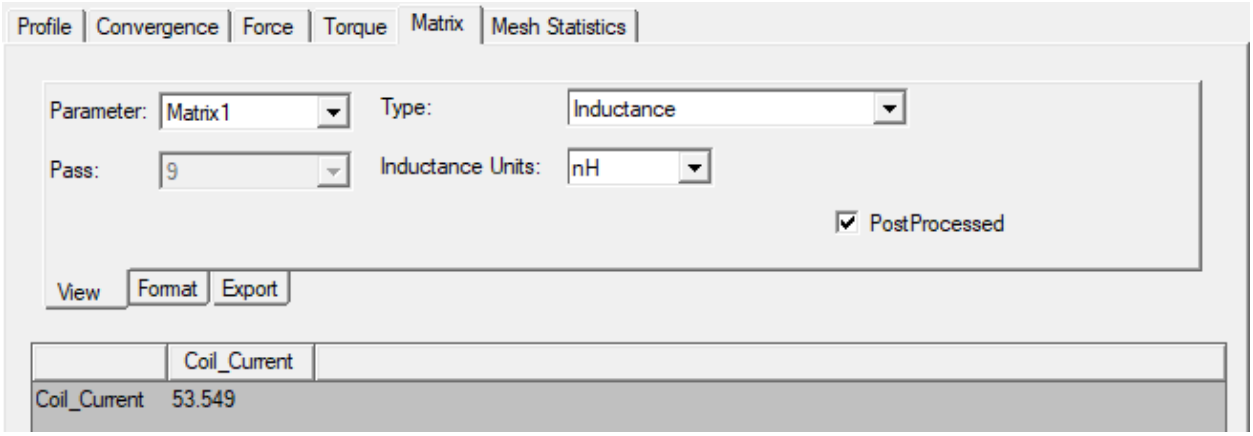
Optionally, click the **...** button, and choose a design variation solved during an optimization or parametric analysis from the **Set Design Variation** window, which lists all the solved variations in the design.

3. In the **Simulation** drop-down menus:
   - Select the solution setup from the left drop-down menu.
   - Select the solved pass data you wish to view from the right drop-down menu, either the **LastAdaptive** pass or an intermediate **AdaptivePass**. If you select **AdaptivePass** you must also select the desired intermediate pass form the **Pass** drop-down menu.
4. Select a **Parameter**.
5. Select one of the items from the **Type** drop-down menu. Items available in the **Type** list depend upon the design's solution type setting. For example:
   - **Inductance** (for magnetostatic and 3D transient A-Phi solutions)
   - **Re(Z), Im(Z)** (for AC Magnetic T-Omega solutions)
   - **R, L** (for AC Magnetic T-Omega and A-Phi solutions) The extracted resistance values of designs that use Litz wire windings take into consideration the stranded AC loss effects due to skin effect and proximity effects.)
   - **G,C** (for ACConduction and 3D AC Magnetic A-Phi solutions)
   - **Resistance** (for 3D transient A-Phi solutions)
   - **Capacitance** (for electrostatic and 3D transient A-Phi solutions)
   - **Conductance** (for DC Conduction solutions)
   - **Flux Linkage** (for magnetostatic and AC Magnetic solutions)
   - **Inductive Coupling Coefficient** (for magnetostatic and AC Magnetic T-Omega and A-Phi solutions)
   - **Capacitive Coupling Coefficient** (for electrostatic, AC Conduction, and AC Magnetic A-Phi solutions)
   - **Conductive Coupling Coefficient** (for DC Conduction solutions)

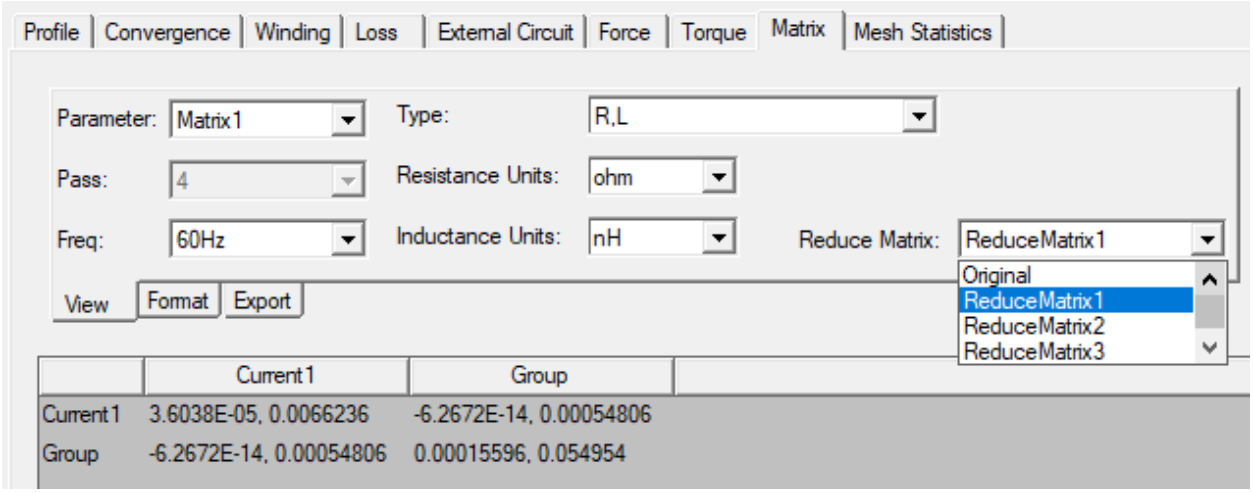
   Depending on the solution type you selected, you may have to specify the **Units** in which to display the information. The available units depend on the matrix type that is displayed.

   The bottom pane displays values for the selected type. The **Format** tab allows you to customize how values are displayed by setting the desired width, precision, and whether to use scientific notation.
6. (Optional) For Magnetostatic designs, the results of the postprocessing calculation on the matrix, including grouped sources defined on the Matrix window's **Post Processing** tab, can be seen by selecting **PostProcessed**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



7. (Optional) For 2D and 3D AC Magnetic designs, you can join (group) two or more excitations to one excitation in either a series or a parallel connection. The result of each set of join (group) operations is known as a Reduce Matrix. For AC Magnetic designs that include one or more reduce matrices, select the reduce matrix of interest from the **Reduce Matrix** drop-down menu. For reduced matrix selections, only the **R, L Type** values are available for viewing. (Refer to Assigning a Reduce Matrix for information on creating reduced matrices.)



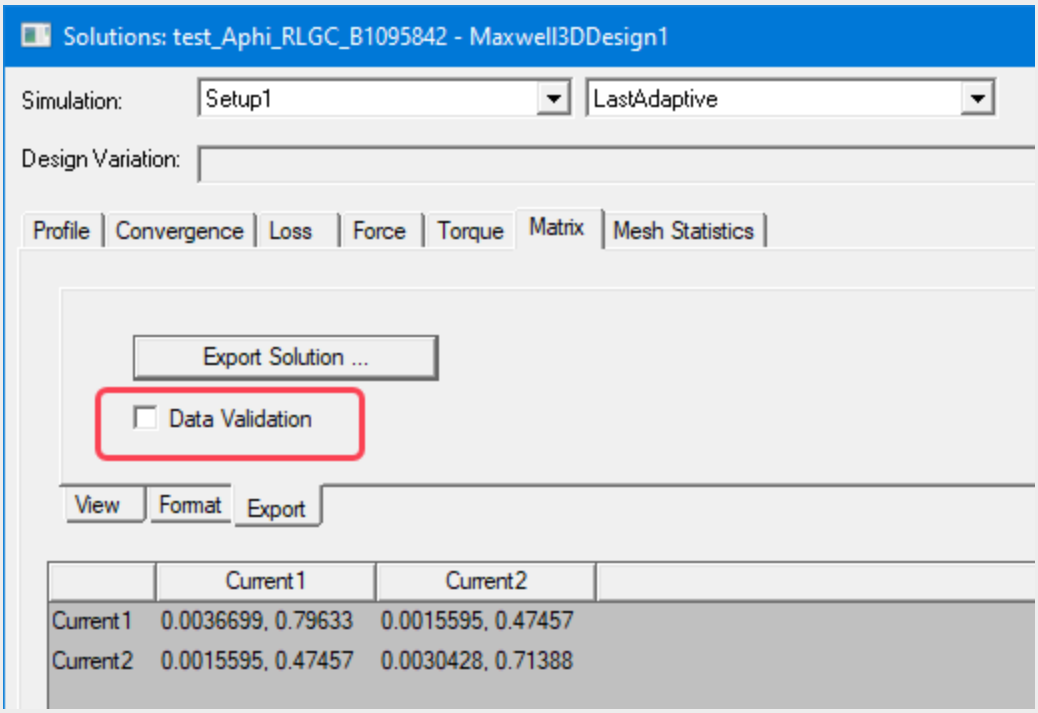
8. (Optional) To export the matrix data (as specified by the **Type** list):
   a. On the **Export** tab, click **Export Solution**.

      The **Export Solution** dialog box appears.

   b. Select a location for the solution data.
   c. Type a filename in which to store the data in the **File name** text box.
   d. Click **Save**. The solution is saved as a data table in the specified file.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: If you are writing out a spreadsheet with AC Magnetic A-Phi\RLGC data, uncheck the **Data Validation** option on the Solutions\Matrix**Export** option. Leaving this option checked may result in an error during export.



9. (Optional) For 2D and 3D Magnetostatic designs only, you may export a circuit based on the matrix parameter type by clicking **Export Circuit** on the **Export** tab. The **Export Circuit** button appears in the dialog box when a supported type is selected.

Exporting a Circuit Model

Note: This command is enabled only when the following two criteria are met:

- The solution type is **Magnetostatic** or **Electrostatic**.
- A Matrix parameter has been added.
- A parametric setup exists.

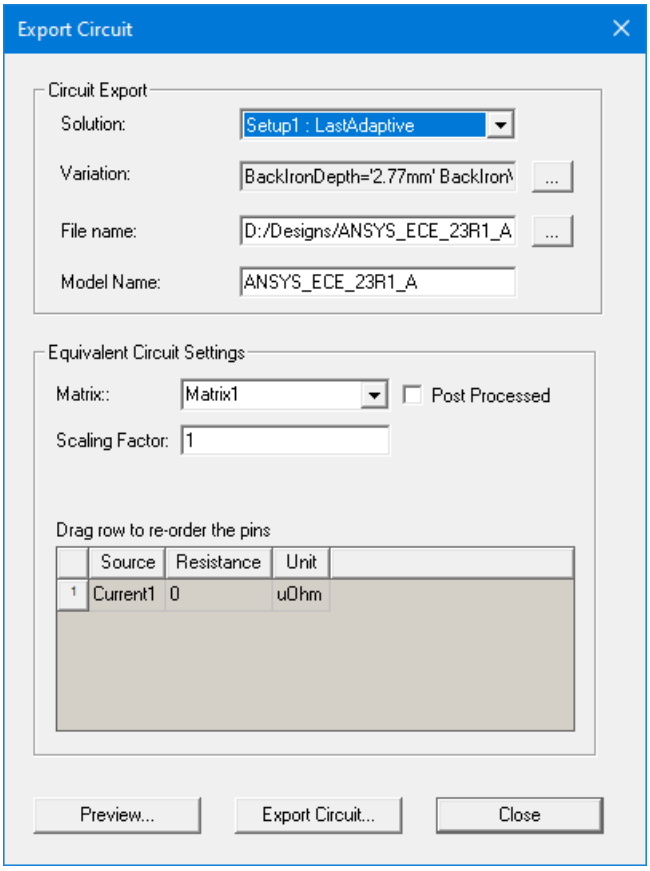
To export the circuit:

1. In the Project Manager tree, right-click the solution setup of interest, and then click **Solutions** on the shortcut menu.
2. In the **Solutions** window that opens, select the **Matrix** tab.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click the **Export** sub-tab.
4. Click **Export Circuit**.

   The **Export Circuit** dialog box appears.



5. In the **Equivalent Circuit Settings** section, select the matrix you want to export from the **Matrix** drop-down menu.
6. (Optional) Enter a scaling factor in the **Scaling Factor** text box. You may want to provide a scaling factor if you are using symmetry to analyze a portion of the model.
7. For an induction matrix, type a **Resistance** for each source listed in the table.
8. (Optional) If desired, drag table rows to re-order the circuit pins.
9. (Optional) The results of the post processing calculation (defined on the Matrix dialog's **Post Processing** tab) on the matrix can be included by checking **PostProcessed**.
10. In the **Circuit Export** section, select a setup from the **Solution Setup** drop-down menu.
11. (Optional) Click the **...** button to open the **Set Design Variation** dialog box to choose a **Variation**.
12. Type the path and file name in the **File name** text box, or click the **...** button to select the desired location for the exported circuit.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13. (Optional) To view the netlist:
    a. Click **Preview**.

       The **Circuit Model Preview** dialog box appears. You can also click **Export Circuit** from within this dialog box.

    b. Click **Close** to close the netlist.
14. To export the circuit to the specified file, click **Export Circuit**.
15. Click **OK** to close the **Export Circuit** dialog box.
16. Click **Close** to close the **Solutions** window.

Coupling Coefficient Matrix

For a matrix with entries M_{ij}, i, j = 1,..., N, the coupling coefficient for row i and column j is given by the following equation:

$$k = \frac{M_{ij}}{\sqrt{M_{ii} \times M_{jj}}}$$

This coefficient indicates how much flux in one coil is linked with the other coil.

If all the flux in one coil reaches the other coil, $|k| = 1$ (and coupling is 100%).

Inductors with $|k| > 0.5$ are tightly coupled. Inductors with $|k| > 0.5$ are loosely coupled.

The mutual inductance (M) falls between 0 and $\sqrt{M_{ii} \times M_{jj}}$ (the geometric mean of the inductances).

AC Loss Effects of Litz Wire on Impedance Matrix

The impedance matrix of polyphase windings is expressed as

$$[Z] = \begin{pmatrix} R_{11} + jX_{11} & \cdots & R_{1n} + jX_{1n} \\ \vdots & \ddots & \vdots \\ R_{1n} + jX_{1n} & \cdots & R_{nn} + jX_{nn} \end{pmatrix}$$

where n is the number of windings.

Litz wire eddy-current losses are computed in during postprocessing. When the total number of windings includes some Litz wire windings, the final AC impedance matrix, including an additional resistance matrix due to Litz wire eddy current losses, is expressed as

$$[Z'] = \begin{pmatrix} R_{11} + jX_{11} & \cdots & R_{1n} + jX_{1n} \\ \vdots & \ddots & \vdots \\ R_{1n} + jX_{1n} & \cdots & R_{nn} + jX_{nn} \end{pmatrix} + \begin{pmatrix} r_{11} & \cdots & r_{1n} \\ \vdots & \ddots & \vdots \\ r_{1n} & \cdots & r_{nn} \end{pmatrix}$$

where resistances denoted by r_{ij} are additional Litz wire eddy-current resistance components.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To compute Litz wire eddy-current resistance components, a permeability freezing algorithm is needed. After nonlinear iteration is convergent and permeabilities in all mesh components are frozen, the original impedance matrix *[Z]* is derived by injecting 1 A exciting current into each phase winding one by one. The self-impedance is obtained from the induced voltage in the excited winding, and all mutual impedances are obtained from the induced voltages in all other windings.

However, the Litz wire eddy-current resistance matrix is derived in a different way. When a winding is excited with 1 A rms current and all other windings with zero-value currents, the value of the total eddy-current loss (no matter in the excited winding, or in other zero current windings), is the value of the self-resistance. That is, if I_1 = 1 A rms current in winding 1 produces total eddy-current loss P_1, then the self-resistance of winding 1 is

$$r_{11}{I_1}^2 = r_{11} = P_1$$

Similarly, if I_2 = 1 A rms current in winding 2 produces total eddy-current loss P_2, then the self-resistance of winding 2 is

$$r_{22}{I_2}^2 = r_{22} = P_2$$

To calculate the mutual resistance between winding 1 and winding 2, 1 A currents in both winding 1 and winding 2 need to be injected. In such a case, the total eddy-current loss is

$$r_{11}{I_1}^2 + 2r_{12}I_1I_2 + r_2{I_2}^2 = r_{11} + 2r_{12} + r_{22} = P_{12}$$

where P_{12} is the total eddy-current loss in all Litz wire windings caused by 1 A currents in both winding 1 and winding 2. Therefore,

$$r_{12} = \frac{P_{12} - r_{11} - r_{12}}{2}$$

Viewing Parameter Data

To view parameter solution data such as force and torque:

1. In the Project Manager tree, right-click the solution of interest in the **Parameters** folder, and then click **View Solution** on the shortcut menu.

   The **Solutions** window appears.

2. The **Design Variation** box shows the current design.

   Optionally, click the **[...]** button, and choose a design variation solved during an optimization or parametric analysis from the **Set Design Variation** dialog box.

3. In the **Simulation** drop-down menus:

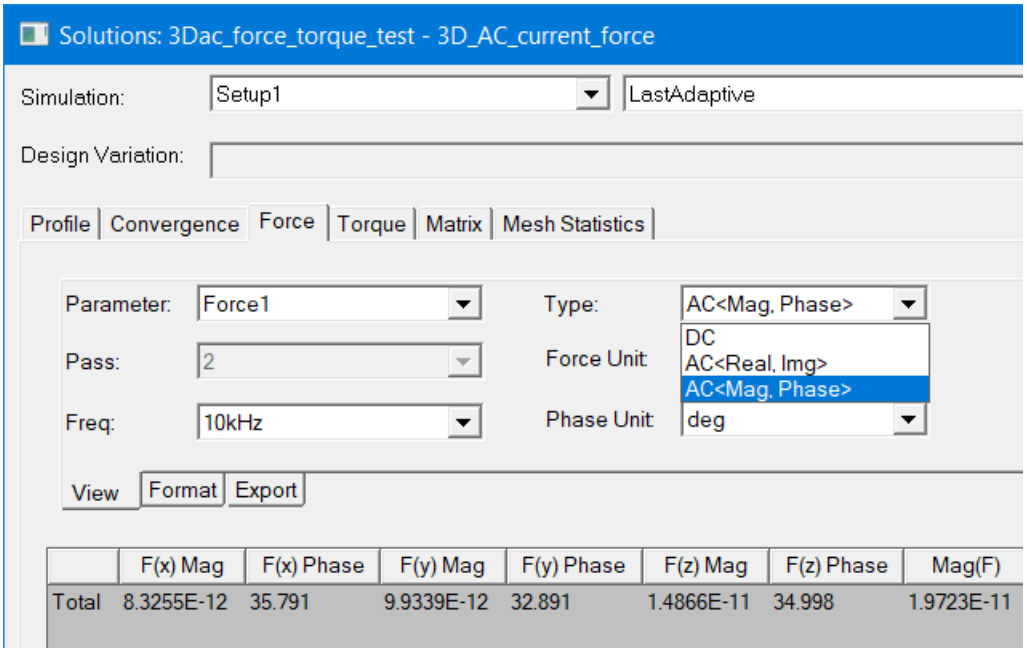
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select the solution setup from the first drop-down menu.
- Select the solved pass data you wish to view from the second drop-down menu, either the last adaptive pass or one of the intermediate passes.

4. On the desired parameter tab, such as **Force** or **Torque**, select a **Parameter** and optionally an adaptive **Pass** to view.

   Depending on the solution type you selected, you may have to specify

   - **Type** – In the screenshot the follows, the simulation was a 3D AC Conduction analysis, so the user has three types of simulation results to select from.
   - **Force Units** – Specifies the units in which to display the information. The available units depend on the matrix type being displayed.
   - **Phase Unit** – This setting is available if the **Type** of result includes phase data.



5. Optionally, to export the parameter solution data:
   a. Click **Export Solution** to open the **Export Solution** dialog box appears.
   b. Select a location for the solution data.
   c. Type a filename in which to store the data in the **File name** text box.
   d. Click **Save**.
6. Click **Close** to close the **Solutions** window.

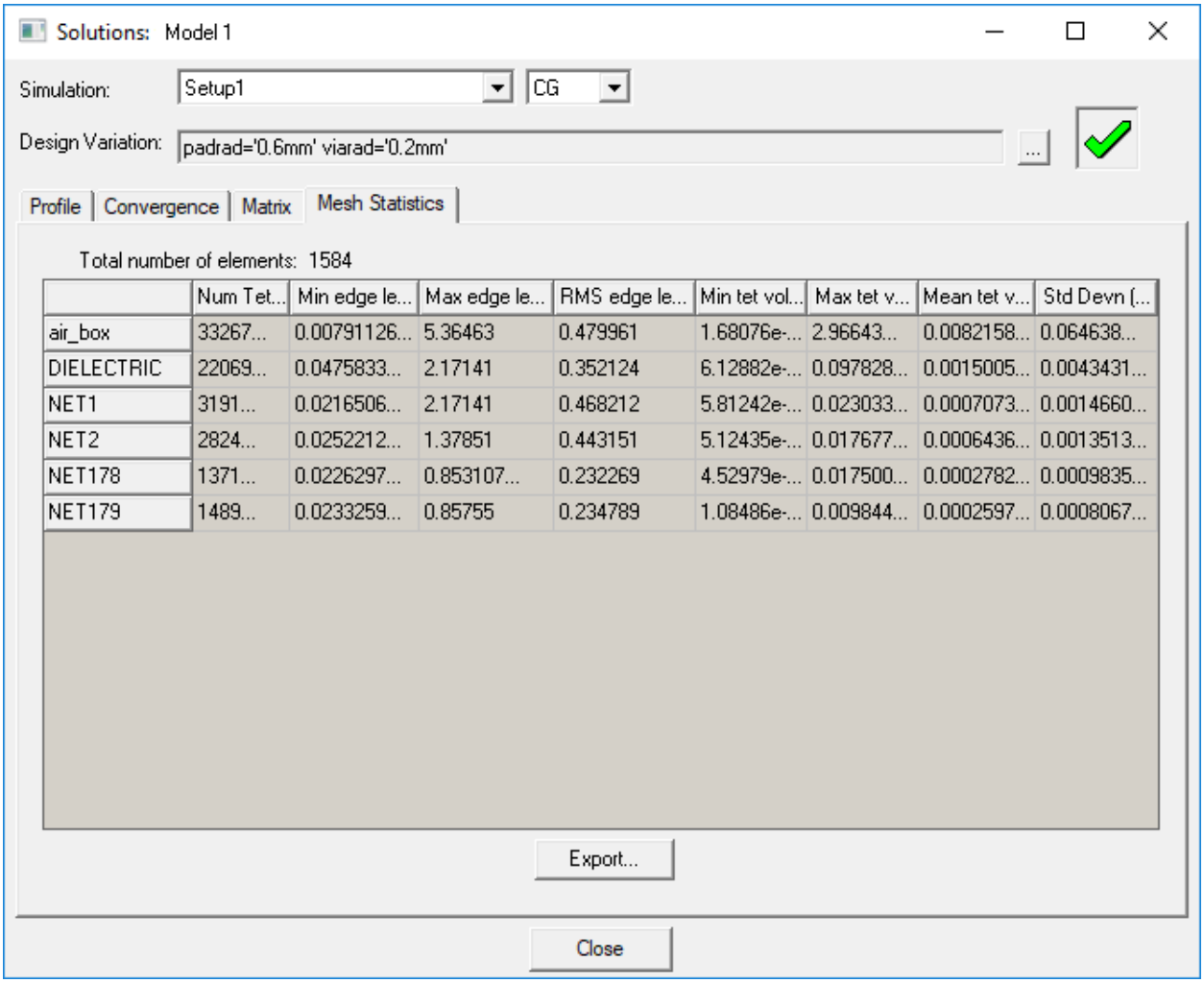
Viewing Mesh Statistics

To view an adaptive solution's mesh information, either during or after the solution process:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the Project Tree, right-click the solution setup of interest, and then click **Mesh Statistics**. Alternately, click the **Solution Data** icon on **Results** tab of the ribbon.

   The **Solutions** window appears with the **Mesh Statistics** tab displayed.



The table lists design elements and for each includes: Num Tets, Min edge length, Max edge length, RMS edge length, min tet vol., max tet vol., mean tet vol. and standard deviation (min elem area, max elem area, mean elem area and standard deviation for 2D designs).

If mesh repairs have been performed, two additional columns appear in the table: **Recovered %** and **Repaired %**. These columns indicate the fraction of an object that was successfully recovered, and the fraction that needed some repair.

To sort mesh statistics by column from low to high values or vice-versa, click a column header. This displays a shadowed triangle pointing down to indicate a list ordered from highest to lowest;

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and a triangle pointing up to indicate a list ordered from lowest to highest. Clicking again inverts the order.

Click on a blank cell above the object list to invert the order of objects. In this case, the cell does not display a directional triangle.

Viewing Winding Results for AC Magnetic T-Omega Solutions

To view winding results for AC Magnetic T-Omega solutions:

1. In the Project Manager tree, right-click the **Results** folder, and then click **Solution Data** on the shortcut menu.

   The **Solutions** window appears.

2. The **Design Variation** box shows the current design. Optionally, click the **[...]** button, and choose a design variation solved during an optimization or parametric analysis from the **Set Design Variation** dialog box.
3. In the **Simulation** drop-down menus:
   - Select the solution setup from the left drop-down menu.
   - Select the solved pass data you wish to view from the right drop-down menu – either the **LastAdaptive** pass, or an intermediate **AdaptivePass**. If LastAdaptive is selected, the **Freq** drop-down menu is enabled for you to choose frequency. If AdaptivePass is selected, the **Pass** drop-down menu is enabled for you to choose the desired intermediate pass.
4. If the design includes windings, the **Winding** tab displays the following columns: **Flux Linkage[Wb]**, **Induced Voltage[V]**, **Input Current[A]**, **Input Voltage[V]**, and **Current [A]**. Results listed in these columns are complex quantities. The **Format** tab allows you to customize how values are displayed by setting the desired width, precision, and whether to use scientific notation.
   - For Current type windings: Flux Linkage, Induced Voltage, and Input Current results are shown. The other quantities are shown as N/A (not applicable).
   - For Voltage type windings: Flux Linkage, Induced Voltage, Input Current, and Current results are shown. The other quantities are shown as N/A (not applicable).
   - For External type windings: Flux Linkage, Induced Voltage, and Current results are shown. The other quantities are shown as N/A (not applicable).
5. Click **Close** to close the **Solutions** window.

Viewing End Connection Results (2D AC Magnetic)

To view end connection results for Maxwell 2D AC Magnetic solutions:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the Project Manager tree, right-click the **Results** folder, and then click **Solution Data** on the shortcut menu.

   The **Solutions** window appears.

2. The **Design Variation** box shows the current design. Optionally, click the **[...]** button, and choose a design variation solved during an optimization or parametric analysis from the **Set Design Variation** dialog box.
3. In the **Simulation** drop-down menus:
   - Select the solution setup from the left drop-down menu.
   - Select the solved pass data you wish to view from the right drop-down menu – either the **LastAdaptive** pass, or an intermediate **AdaptivePass**. If **LastAdaptive** is selected, the **Freq** drop-down menu is enabled for you to choose frequency. If **AdaptivePass** is selected, the **Pass** drop-down menu is enabled for you to choose the desired intermediate pass.
4. If the design includes end connections, the **End Connection** tab displays columns for: **Induced Voltage[V]**, and **Current[A]**. Results listed in these columns are complex quantities. The **Format** tab allows you to customize how values are displayed by setting the desired width, precision, and whether to use scientific notation.
5. Click **Close** to close the **Solutions** window.

Viewing Loss Results (AC Magnetic)

To view loss results for AC Magnetic solutions:

1. In the Project Manager tree, right-click the **Results** folder, and then click **Solution Data** on the shortcut menu.

   The **Solutions** window appears.

2. The **Design Variation** box shows the current design. Optionally, click the **[...]** button, and choose a design variation solved during an optimization or parametric analysis from the **Set Design Variation** dialog box.
3. In the **Simulation** drop-down menus:
   - Select the solution setup from the left drop-down menu.
   - Select the solved pass data you wish to view from the right drop-down menu – either the **LastAdaptive** pass, or an intermediate **AdaptivePass**. If LastAdaptive is selected, the **Freq** drop-down menu is enabled for you to choose frequency. If **AdaptivePass** is selected, the **Pass** drop-down menu is enabled for you to choose the desired intermediate pass.
4. Review the loss results from the **Loss** tab.

   For AC Magnetic T-Omega results, the **Loss** tab displays the following columns: **Core Loss[W]**, **Solid Loss[W]**, **Stranded Loss[W]**, **Stranded Loss R[W]**and **Stranded Loss AC[W]**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For AC Magnetic A-Phi results, the **Loss** tab displays **Core Loss** and **Solid Loss**.

Results listed in these columns are real quantities. The **Format** tab allows you to customize how values are displayed by setting the desired width, precision, and whether to use scientific notation.

5. Click **Close** to close the **Solutions** window.

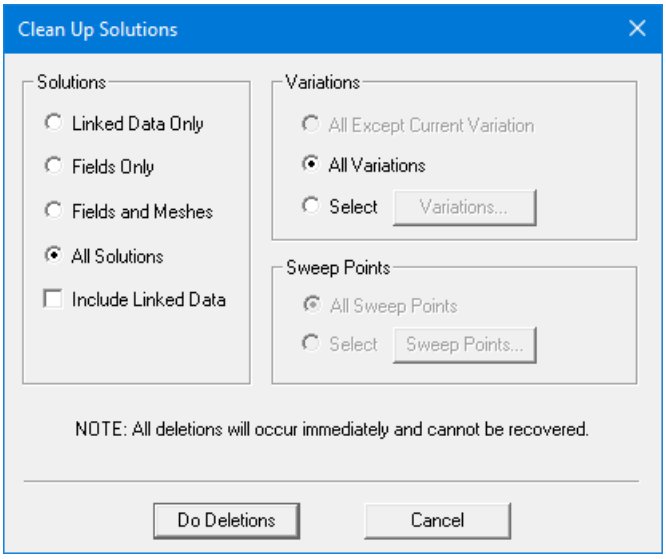
Deleting Solution Data

You can use **Clean Up Solutions** to selectively make deletions, or remove all solutions from the results.

To use **Clean Up Solutions**:

1. Click **Maxwell** > **Results** > **Clean Up Solutions**.

   The **Clean Up Solutions** dialog box appears.



2. Under **Solutions**, select whether you want to delete only fields data, only fields and mesh data, only linked data, or all solution data. Deleting all solution data erases all mesh, matrix, and fields data for all adaptive passes and frequency sweeps for the selected **Variations**.

   By option, you can **Include Linked Data** in the deletions.

   Linked data can be mesh, field or some other post-processing data that the source design generated. The target design for the link caches these data internally to minimize the need to activate the source design. If you don't need to resolve the source design, for example, if it has a lengthy solution time, you should not check **Include Linked Data**. If you need to

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

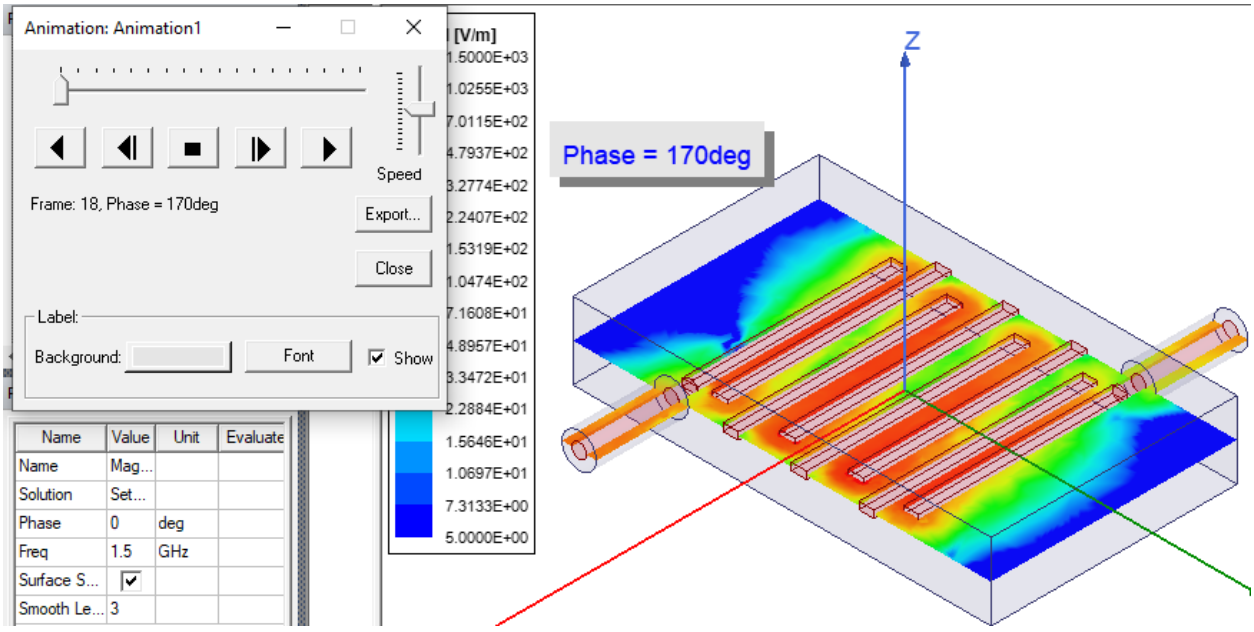
make some change in the source design that would affect the solution, and you want to ensure that the next solve result includes those changes, you should check Include **Linked Data**.

3. Under **Variations**, select which solution data you want to delete:
   - Select **All Except Current Variation** to delete all solution data that does not correspond to the current project and design variable values for the current design.
   - Select **All Variations** to delete all solution data for the current design.
   - Select **Select** to specify the variations you wish to delete. Click **Variations** to select the variations for deletion.
4. Click **Do Deletions**.

   The solution data you selected are deleted. Any post processing reports or field overlays you created that included data you deleted will be marked with an X in the project tree. They will be invalid until new solution data are generated.

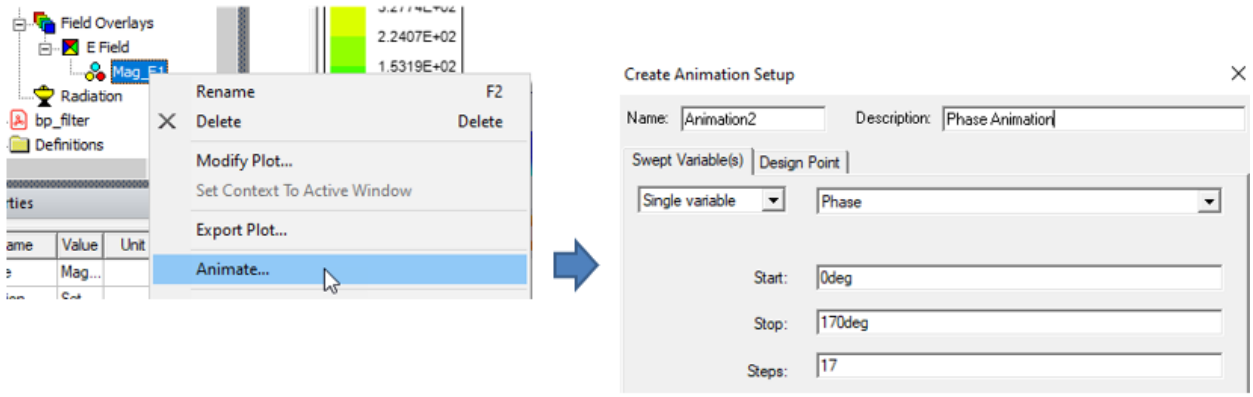
Creating Animations

An animated plot is a series of frames that displays a field, frequency, parametric value, mesh, virtual ray trace or geometry at varying values.



To create an animated plot, you specify the values of the plot that you want to include, just as an animator takes snapshots of individual drawings that make up a cartoon. Each value is a frame in the animation. You specify how many frames to include in the animation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following sections describe how to create different kinds of animations.

> **Note:**
>
> Each animation frame requires memory for storage which depends upon the mesh size and type of plot. Memory usage may become very large during plot animations. To reduce memory usage, specify the minimum number of frames possible. See General Options for more information.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Graphics System requirements for Optimal Performance

In order to obtain optimal performance improvements on fields overlay plot, a workstation-class 3D-capable graphics card with at least 512 MB of memory that supports OpenGL version 2.0 or higher is needed.

On Windows, the default OpenGL version support is v1.1, so you might need to update graphics driver to the latest version;

If you access the application through Windows Remote Desktop which only supports Generic GDI (functionally equivalent to OpenGL v1.0), the performance improvement will also not be available;

To view OpenGL version/extensions supported by your card, the OpenGL Extension Viewer tool is accessible via softpedia.com.

If animation is slow, especially for complex models, for some older graphics cards, you can improve performance by accessing **NVIDIA Control Panel**> **3D Settings**> **Manage 3D Settings – Global Settings** tab and choosing the **Workstation App – Dynamic Streaming** option from the **Global Presets** drop-down menu.

You can export the animation to animated Graphics Interchange Format (GIF), to Audio Video Interleave (AVI) format, or to WebM (.webm) format.

Creating Geometry Animations

You can create geometry animations to evaluate the effect of varying geometry variables on the model. You can also create more complex animations by creating time variables associated with object locations. This section describes the general procedure for geometry animation, and shows a geometry animation with a field overlay, and a second animation for an HFSS Antenna Example project that includes multiple objects moving with respect to a time variable.

Prerequisites

You must define at least one variable associated with the geometry before you create a geometry animation.

If you want to overlay a Field Solution on a geometry animation you must have solved fields for an Optimetrics setup using the same variable.

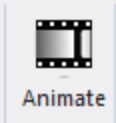
Procedure:

Following is the general procedure for creating an animation that varies a part of the model geometry.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

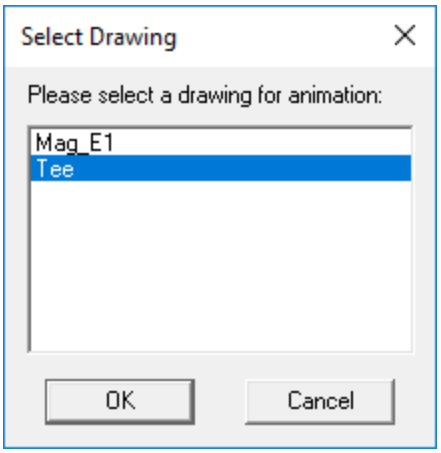
1. Right-click in the *Modeler* window, and click to **View** > **Animate...** or

   Select the **View** tab of the ribbon, and click the **Animate** icon.



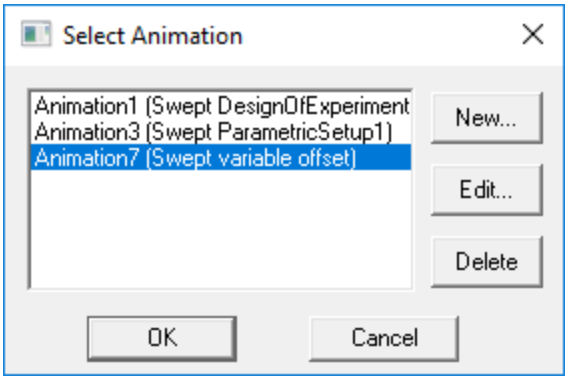
   If multiple geometries can be varied in the design, the *Select Drawing* dialog box appears; proceed to step 2. If only one geometry is variable, proceed to step 3.

2. In the *Select Drawing* dialog box, select the object that you want to animate.

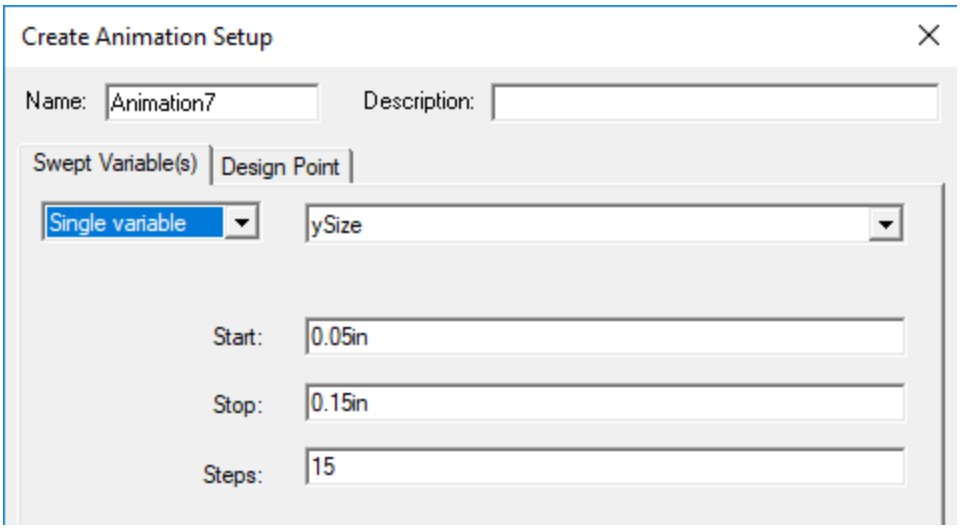


   If previous animations have been created for this project, the *Select Animation* dialog box will appear. Selecting an existing animation from that list starts it when you click **OK**. Click **Edit...** to open the *Modify Animation Setup* dialog box. You may choose an animation setup from the list if one is associated with the geometry variable of interest, and the animation will start. If no existing animation setup is acceptable, select **New** and continue at Step 3 below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

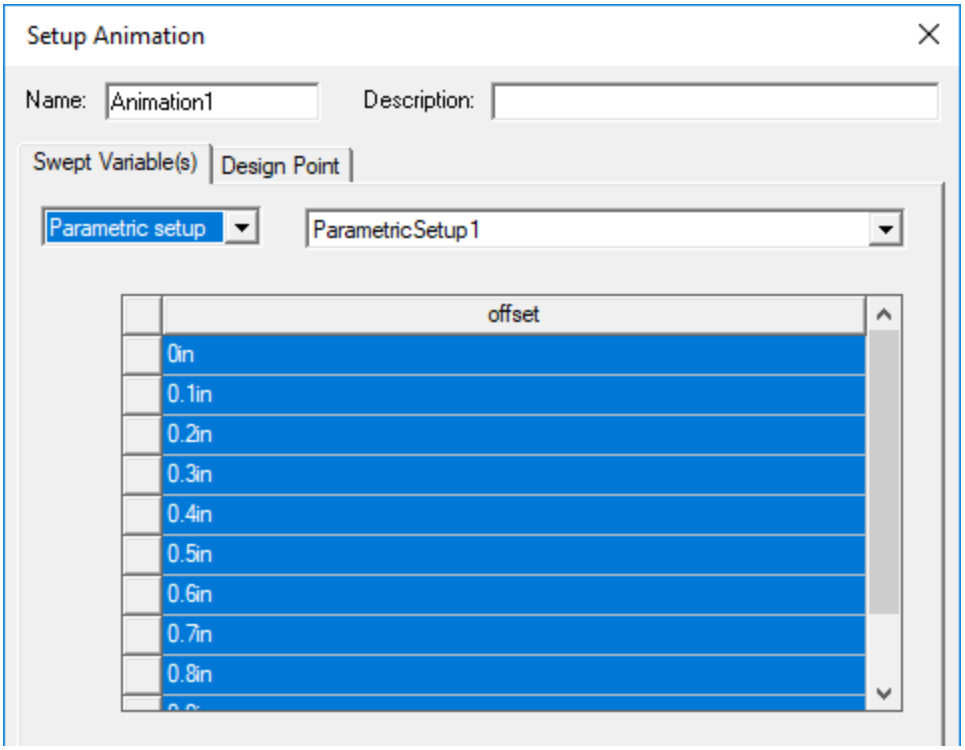


The *Create Animation Setup* dialog box appears:



3. For single variable:
    a. Select the geometry variable that you want to animate from the **Swept Variable** list.
    b. Specify the Start, Stop, and Steps values to include in the animation.
4. If your have created one or more Parametric Sweeps for one or more geometric variable, you can select, **Parametric setup**, the setup to use, and the variable values to animate.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. If you have created a **Design of Experiments** setup, you can select to define an animation based on that setup and values.
6. If the design has multiple project or intrinsic variables, you use the **Design Point** tab to set the values of the non-animated variables.
7. Click **OK**.

   The animation begins in the view window. It will display one frame for each variable value.

Creating Field Plot Animations

To animate an existing field plot:

1. Right-click on the plot name in the Project Manager tree, and then select **Animate**. The **Setup Animation** dialog box appears.

   > **Note:** If previous animations have been created for this project, the **Select Animation** dialog will appear. You may choose an animation setup from the list if one is associated with the geometry variable of interest and the animation will start. Otherwise select **New** and continue at Step 2 below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Type a name for the animation in the **Name** box, or accept the default name.
3. Optionally, type a description of the animation in the **Description** box.
4. Under the **Swept Variable** tab, the **Swept Variable** list includes all of the defined project and design variables. Select the variable that you want to animate from the **Swept Variable** drop-down menu.
5. The **Select Values** table lists all solved values for the variable selected in **Swept Variable**. Select the values to include in the animation. By default, all solved values will be included.
6. *(For Electric Transient Solver plots only)*
    a. Type the starting value of the variable in the **Start** box.
    b. Type the stopping value of the variable in the **Stop** box.
    c. Type the number of **steps** to include in the animation in the **Steps** box.
    For example, if the **Start** value is **0.15 in**, the **Stop** value is **0.45 in**, and the step size is **15**, the animation displays the geometry at 15 values between 0.15 inches and 0.45 inches. The animation also includes the start value, which is the first frame displayed, resulting in a total of 16 frames in the animation.
7. If the design has multiple project or intrinsic variables, click the **Design Point** tab to set the values of the non-animated variables.
    a. Deselect the **Use defaults** check box.
    b. In the table, select the row corresponding to the variable setting of interest.
    c. If intrinsic variables are available, select the value of interest in the **Intrinsic variables** section of the dialog.
8. Click **OK**.

The animation begins in the view window, displaying one frame for each variable value.

The play panel appears in the upper-left corner of the desktop, enabling you to stop, restart, and control the speed and sequence of the frames.

Creating Parametric Animations

Parametric animations can be based on parametric sweeps. Following is the general procedure for creating an animation of a plot based a parametric sweep.

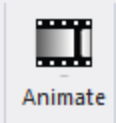
Prerequisites

- For animation of a field plot/overlay plot over a geometric variable, you must define and solve at least one optimetric sweep with same variable values and Save Fields selected before you can create a parametric animation of a field solution. See Creating Geometry Animations.
- For animation of geometry and VRT plots, solving parametric setup is not required.
- Animations can be based on time variables and objects whose location is linked to the time variable. See the animated example below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Procedure:

1. Right-click in the **Modeler** window, and click **View**> **Animate...**, or select the **View** tab of the ribbon, and click the **Animate** icon.

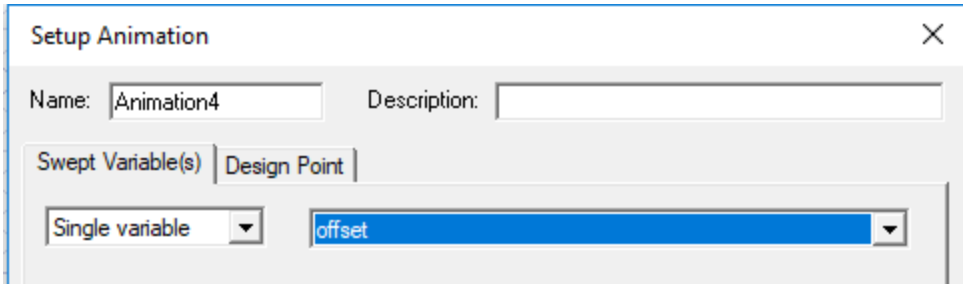


   If multiple geometries can be varied in the design, the *Select Drawing* dialog box appears; proceed to step 2. If only one geometry is variable, proceed to step 3.

2. In the *Select Drawing* dialog box:
   a. Select the geometry variable to vary in the animation.
   b. Select the object you want to animate.

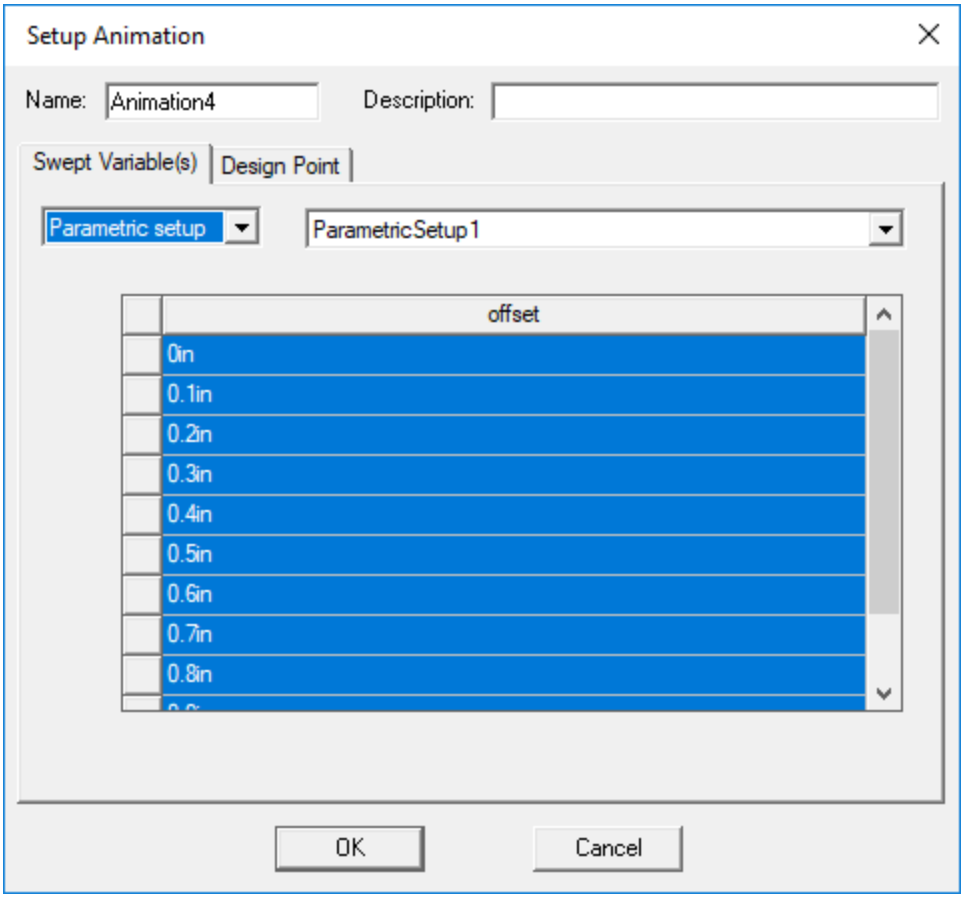
> **Note:**
>
> If previous animations have been created for this project, the *Select Animation* dialog box will appear. You may choose an animation setup from the list if one is associated with the geometry variable of interest, and the animation will start. If no existing animation setup is acceptable, select **New** and continue at Step 3 below.

   The *Setup Animation* dialog box appears.



3. If your have created one or more Parametric Sweeps for one or more geometric variables, you can select **Parametric setup**, the setup to use, and the variable values to animate.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

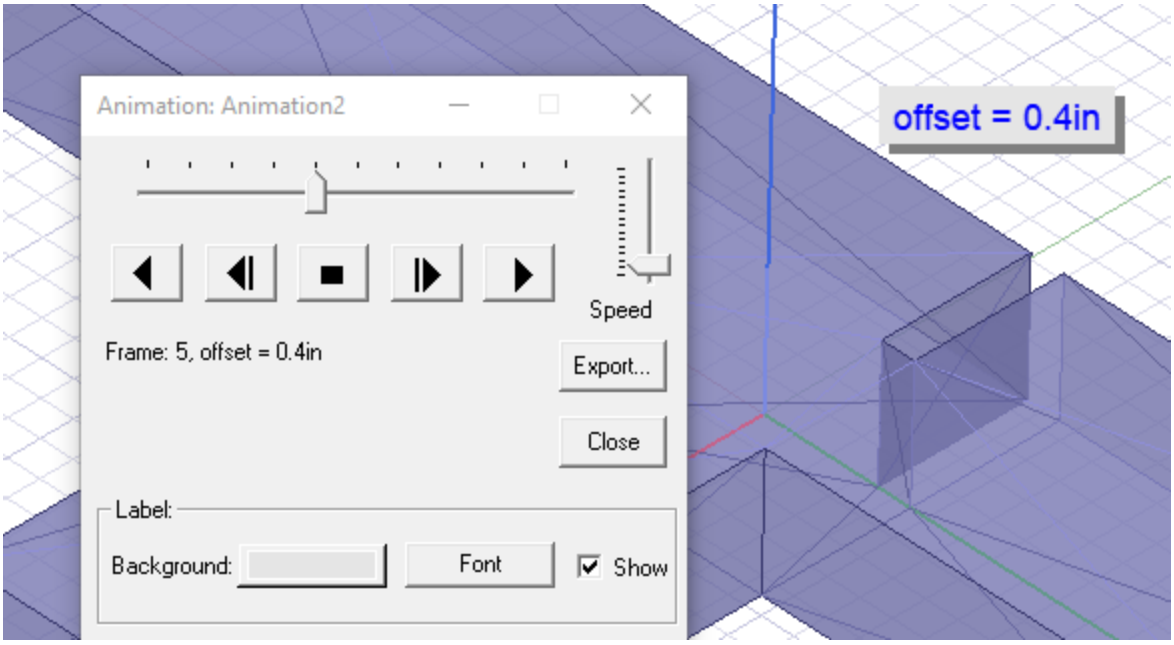


4. Click **OK**.

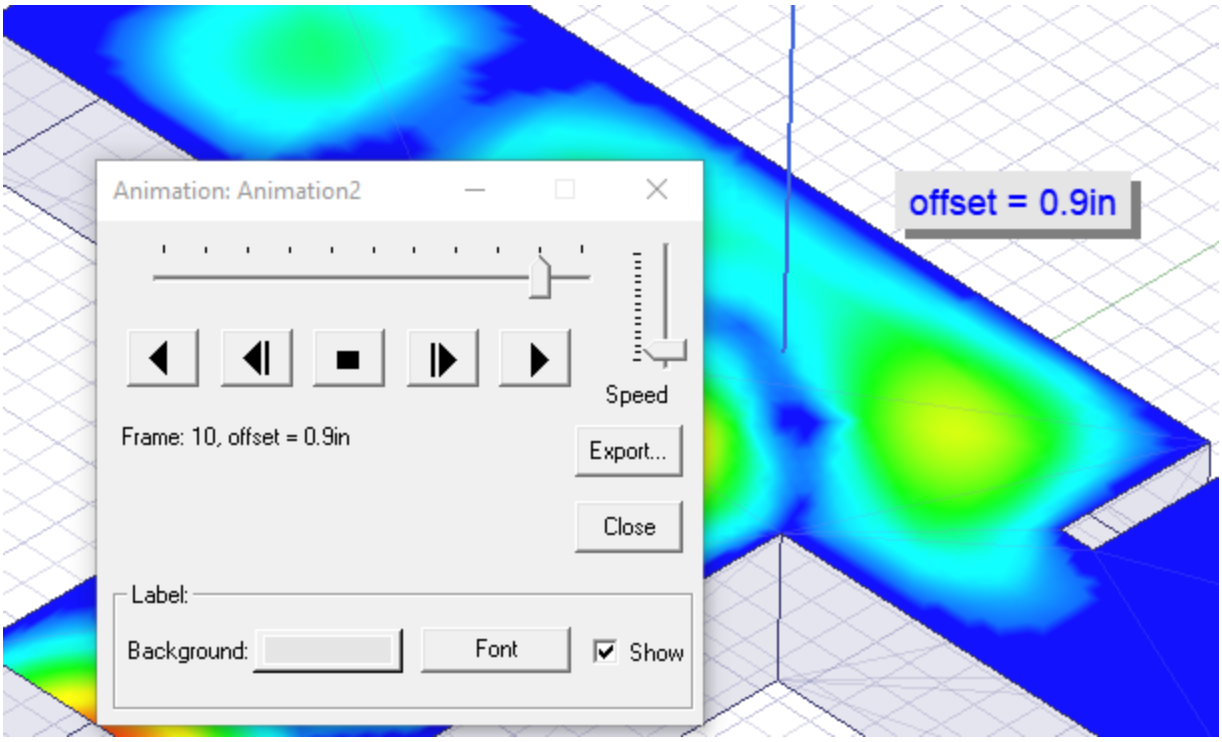
   The animation begins in the view window. It will display one frame for each variable value.

   The **Animation** control panel lets you to stop, restart, and control the speed and sequence of the frames.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



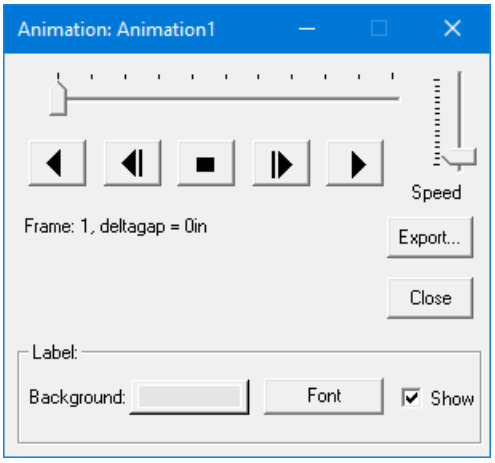
While a parametric animation is running, you can also turn on other field plots.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

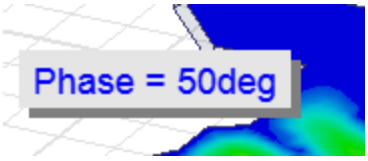
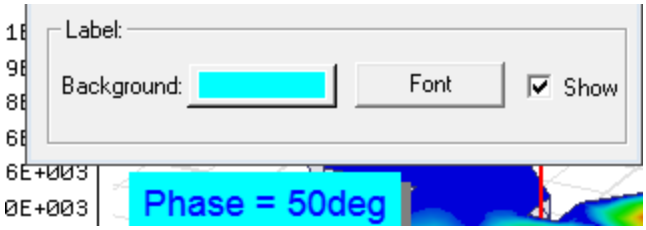
Controlling the Animation's Display

When an animation is displayed in the view window, the **Animation** window (also called the *play panel*) appears in the upper-left corner of the desktop. It has buttons that enable you to control the speed and sequence of the frames, start and stop the animation, and export the animation. Click an area of the window image below to learn its function.



Animation slider	Each dot on the slider represents a frame in the animation. Drag the slider to the right to display the next frame in the animated plot. Drag the slider to the left to display the previous frame in the animation.
	Plays the plot's animation sequence backwards.
	Steps backward through the animated plot one frame at a time.
	Stops the animation.
	Steps forward through the animated plot one frame at a time.
	Plays the plot's animation sequence forwards.
	Drag the **Speed** slider to the top to increase the speed of the animation. Drag the **Speed** slider to the bottom to decrease its speed.
Frame information	The current frame and phase at which the plot is being displayed is listed below the control buttons.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Export...	Enables you to export the animation to an animated Graphics Interchange Format (GIF) or to Audio Video Interleave (AVI) format.
Close	Closes the animation window.
Show label check box	If you select the Show check box, a label showing the swept variable value appears in the animation. You can select the label with the mouse and drag it to a location. 
Background button.	Click the background button to open a color pallet dialog that lets you set the background for the swept variable label. 
Font	The font button opens a font selection dialog that you can use to set the Font, Font Style, and Size for the label. The Default is Arial Narrow 14pt.

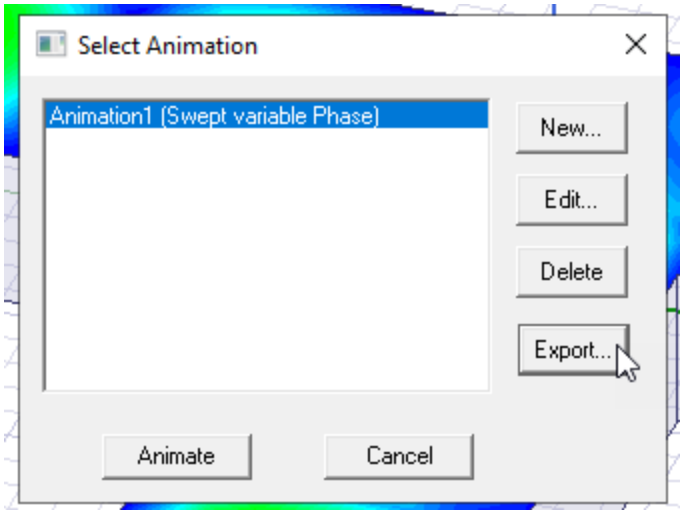
Exporting Animations

You can export animations from the **Animation** control dialog box, or, for Ensight format, from the **Select Animation** dialog box.

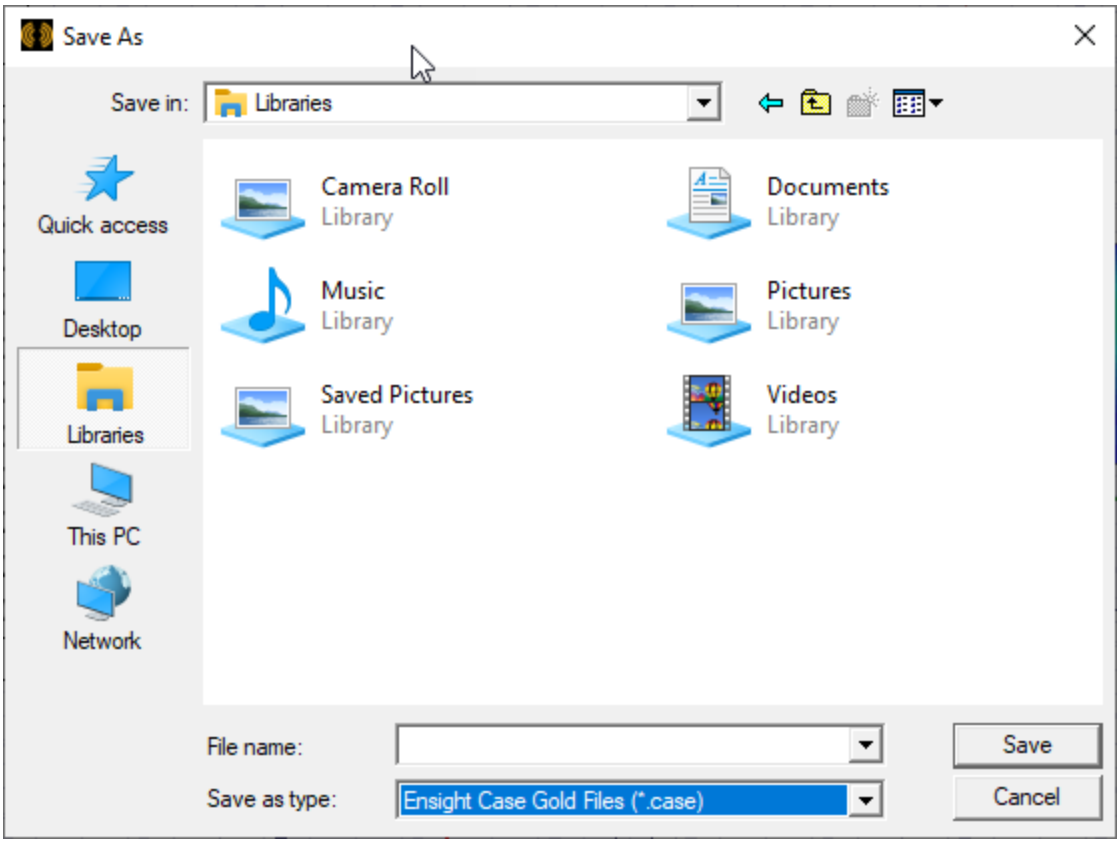
Exporting Ensight from the Select Animation Dialog Box

1. Create the animation you want to export.
2. In the **Select Animation** dialog box, select **Export...**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



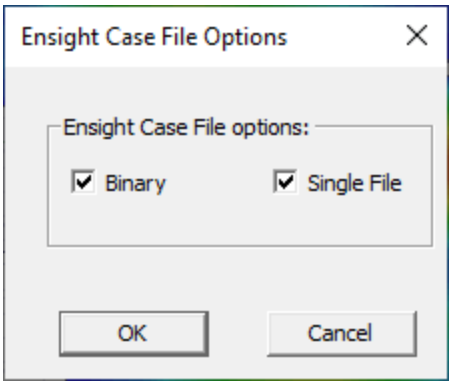
3. This opens the Browser window.



Navigate to the desired location and provide a file name. The format will be Ensight .case files.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

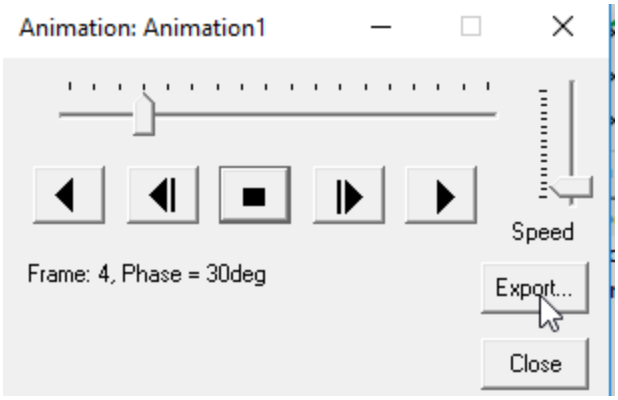
4. When you select **Save**, the **Ensight Case File Options** dialog box opens.



The binary option permits a more compact and efficient animation creation process in Ensight. The Single File options leverages the Ensight transient dataset capability to export all frames of field/geometry data to a single case file. On **OK**, every visible and animatable field plot should generate a case file with name in the format of "UserTypedName_fieldPlotName.case"

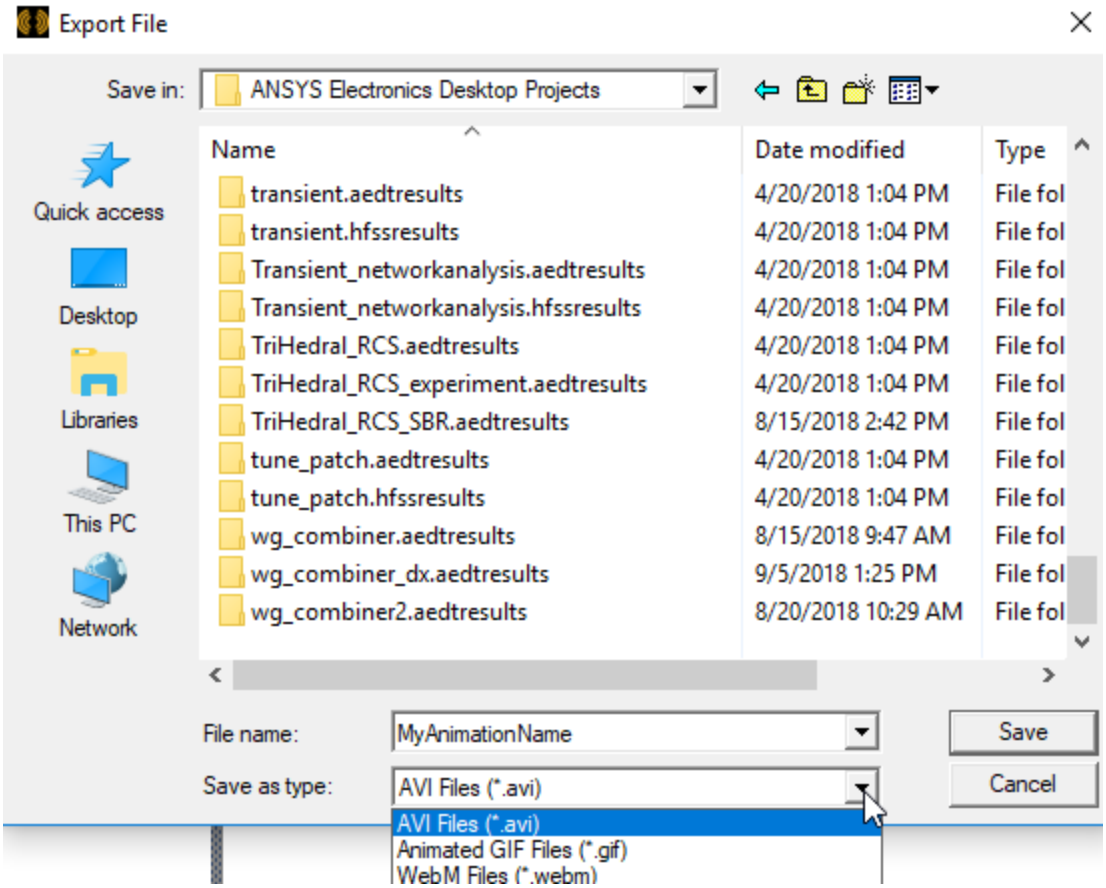
Exporting from the Animation Control Dialog Box

1. Create the animation you want to export.
2. In the play panel, click **Export...**.



The *Export File* dialog box appears.

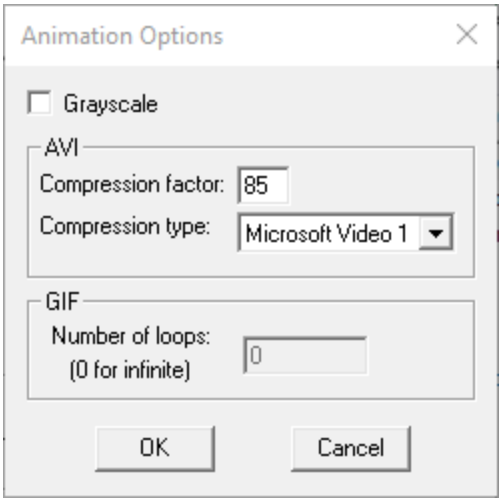
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Specify the directory to **Save in**, the **File name**, and use the **Save as type** drop-down menu to select **Animated GIF File (.gif), AVI File (.avi),** or **WebM File (.webm)**.

   The *Animation Options* dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. To replace colors in the file with 256 shades of gray, select **Grayscale**.

   Grayscale animations tend to use less memory than full color animations.

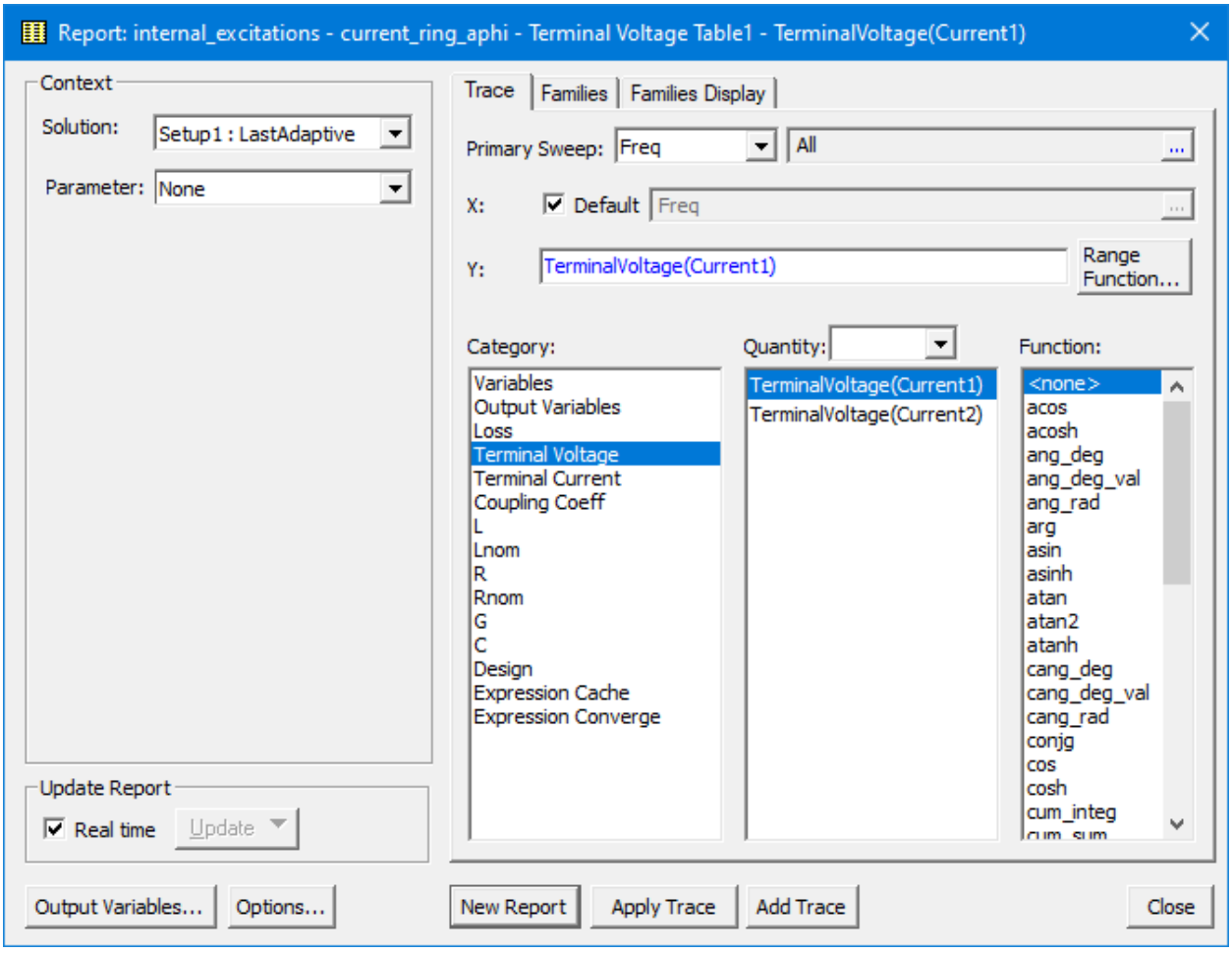
5. For AVI format export, specify the **Compression factor** (the default is 85) and one of the following **Compression types**:
   - INTEL Indeo
   - Cinepak
   - Microsoft Video 1
   - None
6. For GIF format export, specify the number of loops. The default "0" denotes infinite loops.
7. Click **OK** to close the *Animation Options* dialog box.

   The animation is exported to the file format you specified.

Terminal Current/Voltage Output

For Eddy and Transient A-Phi solvers, it is possible to compute the current and voltages on terminals.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The terminal current is computed by

$$I = \int J \cdot n ds$$

where ***n*** is the surface normal pointing inwards to the domain in the case of single potential excitation models, and it is the excitation direction for double potential excitation models.

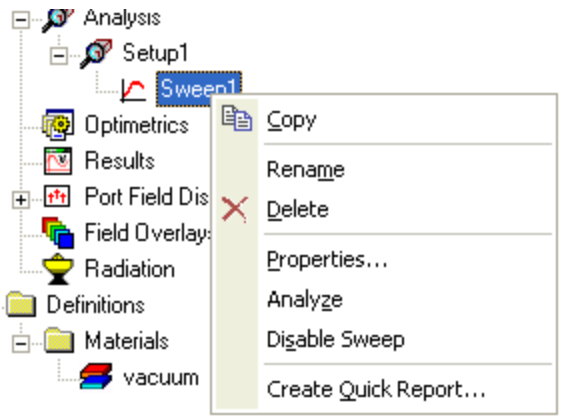
The terminal voltage is the electric potential on the surface in the case of single potential excitation models and the voltage drop in the case of double potential excitation models.

Creating a Quick Report

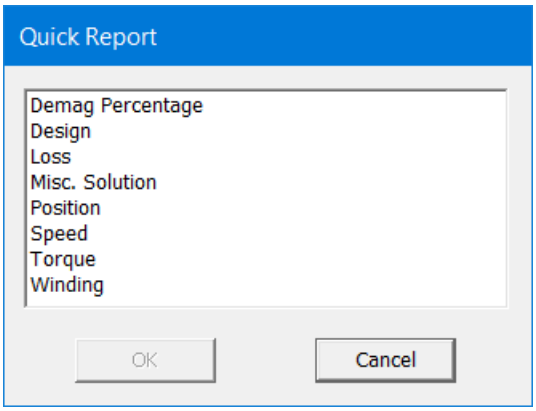
Following is the procedure for creating a quick report.

1. On the Project tree under Analysis, select a **setup** or **sweep icon**, or the **Results** icon.
2. Right-click to display the shortcut menu and select **Create Quick Report**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Quick Report** dialog box appears.



The list of available reports differs depending on the Solution type.

3. Select the one or more categories for the report from the list and click **OK**.

A rectangular plot for each selected category displays. The new plot or plots appear in the Project tree under the Results icon. The default Report Name that appears derives from the report type specified in the **Quick Report** dialog box.

Quick Report Categories

When using the Quick Reports function for Solutions, the following report categories may be available depending upon the solution parameters requested, solution type, etc:

Category	Description
Coupling Coeff	Reports values of the coupling coefficient.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Design	Reports values of design variables.
Force	Reports values of Force parameters such as Force Magnitude and Components, Load Force.
Torque	Reports values of Torque parameters such as Torque Magnitude and Components
L, Lnom	Reports values of Inductance Matrix calculations.
MagFlux, MagFluxNom	Reports values of magnetic flux calculations.
C	Reports values of Capacitance Matrix calculations.
R	Reports values of Resistance Matrix calculations.
Z	Reports values of Impedance Matrix calculations.
Loss	Reports values of Loss parameters such as CoreLoss, EddyCurrentLoss, ExcessLoss, HysteresisLoss, SolidLoss, StrandedLoss, StrandedLossR, and StrandedLossAC.
Misc Solution	Reports values for the solution process such as DeltaT, Energy Error, and Temporal Error
Position	Reports Position of moving objects.
Speed	Reports Speed of moving objects.
Winding	Reports Flux Linkages, currents and voltages in coils.

Creating a New Report

Following is the general procedure for creating a new report:

1. On the **Maxwell** menu or the Project Manager, point to **Results**, select **Create *<type>* Report**, and select the **Display Type** template.

   The **Results** tab for the Ribbon will show icons and drop-down menus for available report types for the active project.

   If you have created custom report templates (for example, including your company name or other format changes), you can also create a report based on that template by selecting **Maxwell or RMxprt** > **Results** > **Report Templates** > **PersonalLib** > *<templateName>*. You can also make such changes the default for new reports by right-clicking a modified report and selecting **Report Templates** > **Save Settings as default**.

   When you have selected the report and display type from the **Results** menu, the **Report** dialog box appears, with the **Trace** tab selected by default.

2. In the **Context** section, make selections depending on the design and solution type.
   - **Solution** field with a drop-down selection list. This lists the available setups and sweeps. As a minimum, the LastAdaptive solution and AdaptivePass solution is

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

available to choose.

The AdaptivePass solution context can be selected to allow any value or parameter to be plotted versus the adaptive pass. This function is usually used to evaluate the convergence of the solution.

- *For Transient projects only*:

  Domain field with a drop-down menu containing options for plotting vs time (Sweep Domain) or plotting vs frequency (Spectral Domain).

- *For Transient projects only*:

  Domain field with a drop-down menu containing options for **Average and RMS** and **Transient D-Q** for electric machines. Click **Machine Options** to setup the machine specific parameters.

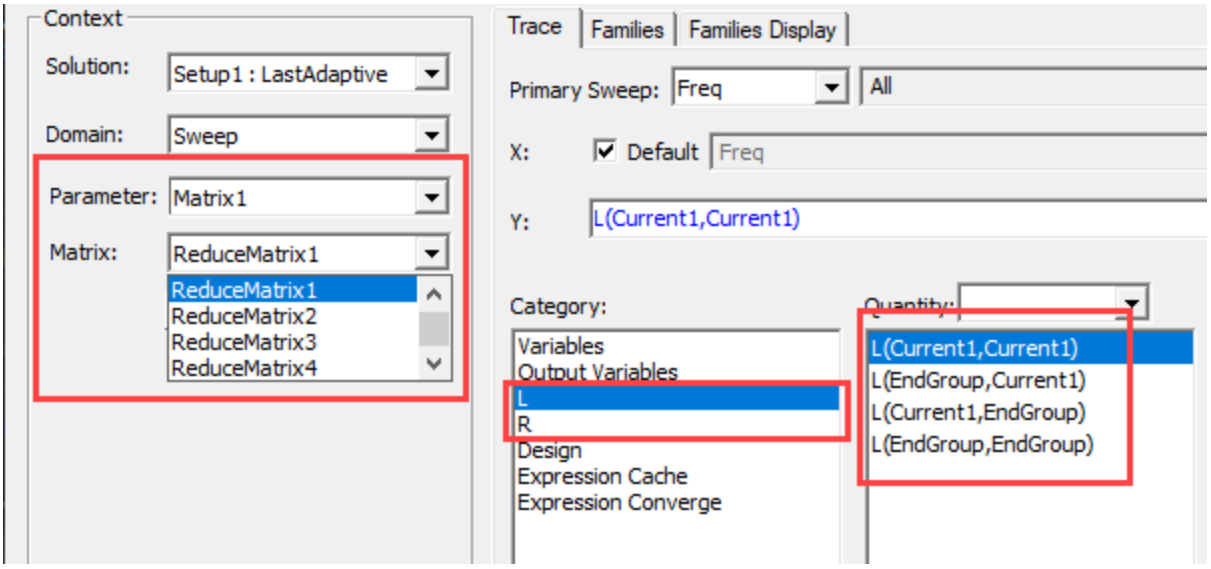
- *For AC Magnetic projects only*:

  Domain field with a drop-down menu containing options for plotting vs frequency (Sweep Domain) or plotting vs time (Time Domain).

  Matrix field with a drop-down menu containing options for plotting *matrix* and *reduce matrix* parameters.

  > **Note:**
  >
  > In Maxwell AC magnetic designs, the user can create *matrix* parameters, which will cause the solver to produce an impedance *matrix* for the selected excitations. In addition, the user can group (wire) two or more excitations to one excitation in either a series or parallel connection referred to as a *reduce matrix*. The **Matrix** field appears only if a matrix entry is selected in the **Parameter** field. (Refer to *Assigning a Matrix* for information on creating a *matrix* and *Assigning a Reduce Matrix* for information on creating *reduced* matrix parameters for AC magnetic designs.)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Parameter** field with a drop-down menu. Whether this field appears, and the parameters listed depend on the Solution type and the ***<type>*** selected.
- *For Fields reports*: **Geometry** field with a drop-down menu. This applies the quantity to a specific geometry.

3. In the **Y** Component section of the dialog box, make selections for the following:
   a. Categories – depend on the Solution type and the design. For example, Magnetostatic categories include: Torque, Output Variables, Inductance, and other user-selectable solution parameters. Transient categories include: Loss, Output Variables, Variables, and others. AC magnetic categories include: Loss, Winding, End Connection, L (inductance), R (resistance) Z (impedance), and others.
   b. Quantities for Y are relative to the selected category. For example, Loss quantities include: CoreLoss, EddyCurrentLoss, HysteresisLoss, and others. Winding quantities include: Flux Linkage, Induced Voltage, and others - depending on the winding type.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> The Quantity text field can be used to filter the Quantity list by typing in text, or by using the four predefined selections. This is useful if the Category selected produces a lengthy Quantities list. See Filtering Quantity Selections for the Reporter.
>
> When the matrix is very large, the number of quantities can be correspondingly huge. Therefore, the Quantities field can optionally use a tree structure to divide matrix quantities into groups by their first element name. The initial display shows groups, without initially listing group members. See Report Setup Options.

c. **Function** list to apply to the Y quantities.

d. Value field displays the currently specified Quantity and Function. You can edit this field directly.

> **Note:**
>
> Color shows valid expression.

e. **Range Function** button – opens the **Set Range Function** dialog box. This applies to the currently specified Quantity and Function.

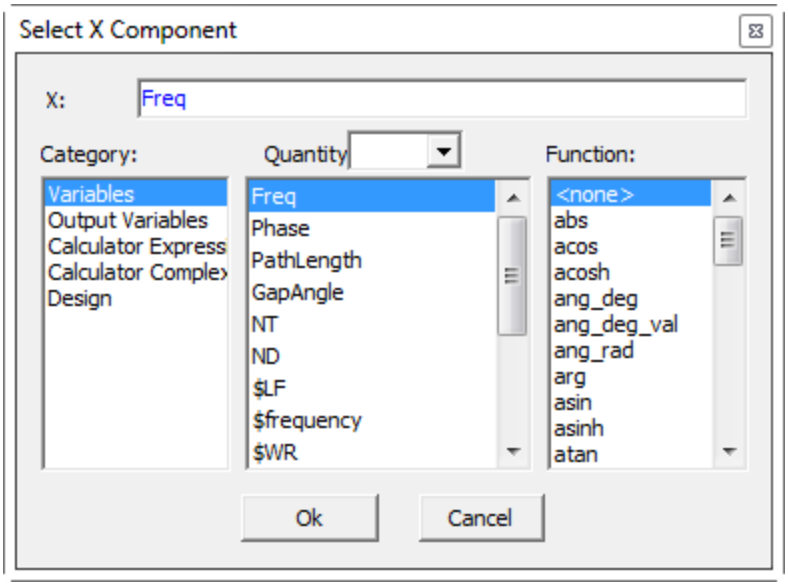
f. To use a dataset in this report, set the **Y** component to a Piecewise Linear Function `pwl(dataset_expression, primary_sweep)` where is the `primary_sweep` is what you set in the next step.

4. In the **X (Primary Sweep)** section, make selections for the following:

   a. Select the Primary value(s) from the drop-down menu.

      To select an X component that is different from the Primary Sweep, uncheck the Default field to enable the X field and **Browse** [...] button. Click **Browse** [...] to display the **Select X Component** dialog.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This lets you specify the X component as you do the Y; that is, in terms of Categories which define the selectable Quantities, and Functions to apply. After making selections, **OK** the dialog box to assign the X component.

b. If sweeps are available, you can select **Browse** [...] to display a panel that lets you select Use all values, or selected sweep or sweeps, or access an Edit Sweep dialog box with further editing options. Post-Processing variables are Post-Processing sweeps/editable sweeps, so you can use the **Edit Sweep** dialog box to create your own sweep.

c. The **Families** tab provides a way to select from valid solutions for sweeps where a simulation has multiple variables defined (for example, for a parametric sweep). If so, the variables other than the one chosen as the **X (Primary sweep)**, are listed under the **Families** tab with columns for the variable, the value, and an Edit column with an ellipsis [**...**] button. See Using Families tab for Reports.

5. **Update Report** setting
   - **Real Time** checked – enable real time updates for all reports while the reports are being edited.
   - **Real Time** unchecked – enables drop down menu to **Update All Reports** or **Update Report**. Reports will only be updated with one of these user selectable update options or upon exiting the report dialog box. This can be useful if you expect a trace to take time to display. You can then add additional traces without having to wait.
6. The **Report** dialog command buttons permit you create a new report with the settings you provide, or to modify an existing report.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Output Variables** – opens the **Output Variables** dialog box.
- **Add Trace** – this is enabled when you have created or selected a report. Add one or more traces to include in the report.
- **Update Trace** – updates the selected traces in a report based on further processing or changes.
- **New Report**. Adds a report to the Project tree under the Results icon. The new Report is displayed in the Project window.
- **Options** – opens the **Report Setup Options** dialog box. This contains a check box for using the advanced mode for editing and viewing trace components. This mode is automatic if the trace requires it. It also contains a field for setting the maximum number of significant digits to display for numerical quantities.
- **Close** – closes the **Report** dialog box.

7. Click **New Report** to create a new report in the Project Manager.

   The report appears in the view window. It will be listed in the project tree under Results, with the default name based on the Report Category you selected, for example, S Parameter Plot *n* or Output Variables Plot *n*. You can edit the plot names in the project tree and the plot header text in the report synchronizes. Traces within the report also appear in the project tree.

   Some plots may take time to complete. Performing a **File** > **Save** in such cases after the plot has been created will permit you to review the plot later without having to repeat the calculation time when you reopen the project later.

8. To speed redraw times for changed plots, perform a **Save**. This saves the data that comprises expressions. For example if re(S_{11})*re(S_{22}) is requested over multiple widths, each of the S_{11} and S_{22} are stored when you save. If you do not do a save of a changed plot, the changed version is not stored.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Remember that the evaluated value of an expression is always interpreted in SI units. However, when an angle quantity is plotted in a report, you have the option to plot values in units other than SI. If you want to plot the polar angle of a complex simulation result, S11 say, you can choose between ang_deg(S_{11}) and ang_rad(S_{11}). Both of these return the exact same angle quantity but in degree and radian units respectively.

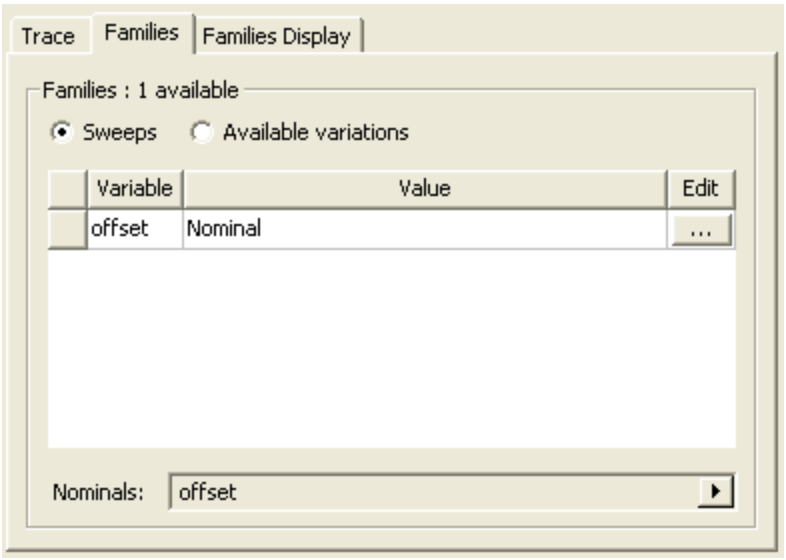
Note that when used in expressions, some surprising outcomes might result. For example, the expression "1+ang_deg(S11)" represents an ‘angle’ and the number "1" is treated as "1 rad". The angle SI unit is attached to any unitless number that is added/subtracted from an angle value. If you want to treat "1" as degrees, make it explicit and use "1deg + ang_deg(S_{11})" instead.

If you are interested in unitless degree values, two additional functions exist: ang_deg_val(S_{11}) and cang_deg_val(S_{11}). These return simple numbers and are treated as such by any expression. If the complex S_{11} lies on the positive Y axis say, ang_deg_val (S_{11}) would be 90 and "1 + ang_deg_val(S_{11})" will be 91.

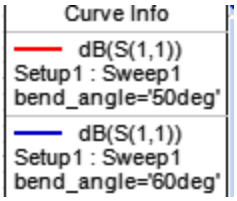
Using Families Tab for Reports

The **Families** tab of the **Report** dialog box provides a way to select from valid solutions for sweeps where a simulation has multiple variables defined and solutions exist for multiple variable values (for example, for a parametric sweep or re-running an analysis with a different variable value). If no variables are defined, or none have solutions for different values, 0 families will be available. If so, the variables other than the **X (Primary sweep)**, are listed under the **Families** tab with columns for the variable, the solution value (which may be All, Nominal, or a Specific value), and an Edit column with an ellipsis [**...**] button. Families gives the number available. If an existing variable is specified as Nominal, only that value is currently available. You can set any solved variables as Nominal, All, or select from values provided for Available solutions.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



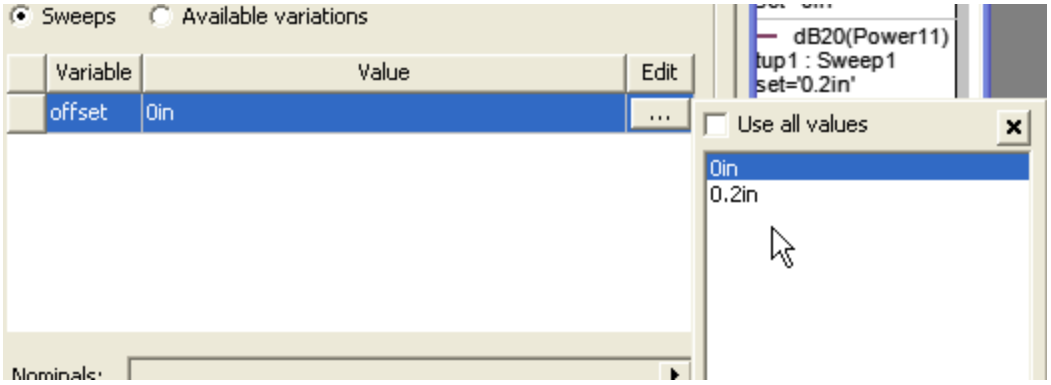
When you select a variable with multiple solved values, a trace for each solved value appears in the Report, with the variable value appended to the trace name in the Report legend.



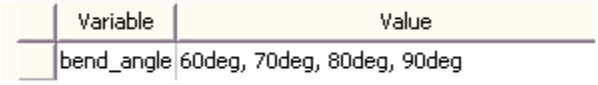
When families are available, you can make selections for the following:

1. Select the **Sweeps** radio button (the default) to list the swept variables you can select or the Available variations button to list and select variation values for which solutions exist.
2. With the **Sweeps** radio button selected, click the ellipsis [**...**] button to display a list of variable values for a particular variable. If many variables exist, you can use a scroll bar to navigate the list.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

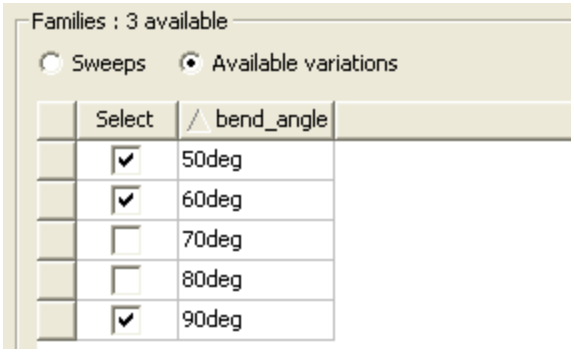


- To select all values, click the check box for **Use all** values. This writes "All" in the value field for that variable. You can also select individual values by clicking on them.
- To select a range of values, hold down the shift key, and click again.
- To select intermittent additional values, hold the Ctrl key and click additional. The values you select are highlighted in the list, and are also listed in the Values column for that variable.
- To select all, use the **Select All** button. This highlights the complete list, as well as listing all values for the variable in the Value field.



- To clear the selections, use the **Clear All** button.

Select the **Available variations** radio button to list the choices that derive from variable combination.

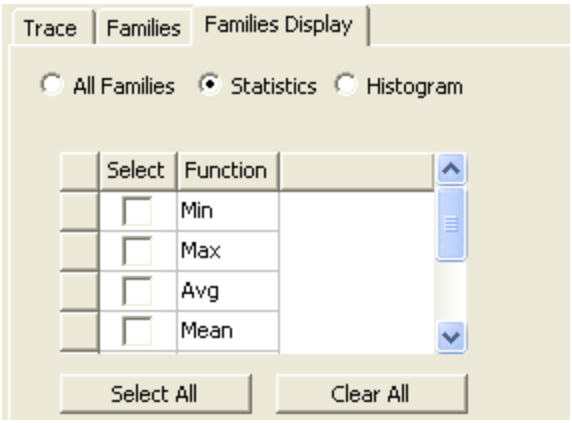


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

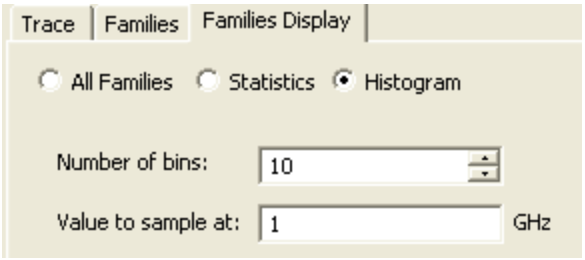
a. To select individual variations, check the select box.
b. To check or clear all variations at once, click the **Select** button at the top of the column.
c. To invert the list order, click the triangle beside the variable name.

The **Families Display** tab has three radio button selections.

a. **All Families**
b. **Statistics** lists a table statistical functions that you can select to apply to the plot. The functions include Min, Max, Avg, Mean, Variance, Std Dev, and Sum. You can use the Select check boxes or the Select All and Clear All button.



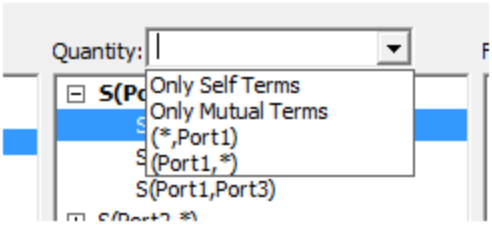
c. **Histogram** lets you select the number of bins to use for a histogram plot, and the sampling frequency to use.



Filtering Quantity Selections for the Reporter

When a two-port quantity Category is selected, four predefined filters are added to the combo box. "Port1" is the first matrix element name found in the quantity list.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Only Self Terms – Only display quantities when the first and second port are same.
- Only Mutual Terms – Only display quantities when the first and second port are different.
- (*,Port1) – Only display quantities when the second element name is "Port1". You can edit the element name to display quantities for other elements.
- (Port1,*) – Only display quantities when the first element name is "Port1". You can edit the element name to display quantities for other element.

Creating a Report from a Report Data File

Using the Report Data File format, **.rdat**, a report can be saved (exported) from one design, and subsequently imported into a different design (or even imported into the same design). This allows you to take a static snapshot of a set of simulation data and view it at a later stage of the design process, or view the same data set in one or more different designs.

To save a report to a Report Data File:

1. Create the report that you wish to save.
2. Right-click in the report window (or right-click on the desired report icon in the project tree) and select **Export** to open the **Export Report** dialog box.
3. Select the Report Data files option (**.rdat**) in the **Save as type** selection drop-down menu.
4. Navigate to the desired location and either create a file name or select an existing file for the exported report; and then click **Save**.

To create a report from an existing Report Data File:

1. Right-click the **Results** folder in the project tree and select **Create Report From File**.
2. Browse to select the desired Report Data file (**.rdat**), and then click **Open**.
3. The selected report is added to the design and opens in a new report window.

Modifying Reports

To modify the data that is plotted in a report:

1. In the Project Manager, click the report you want to modify.
2. Right-click **Modify Report**.

   The **Report** dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. The **Report** dialog box command buttons permit you create a new report with the settings you provide, or to modify an existing report.
    - **Output Variables** – opens the **Output Variables** dialog box.
    - **Add Trace** – this is enabled when you have created or selected a report. Add one or more traces to include in the report.
    - **Update Trace** – updates the selected traces in a report based on further processing or changes.
    - **New Report** – adds a report to the Project tree under the Results icon. The new Report is displayed in the main window.
    - **Options** – opens the **Report Setup Options** dialog box. This contains a check box for using the advanced mode for editing and viewing trace components. This mode is automatic if the trace requires it. It also contains a field for setting the maximum number of significant digits to display for numerical quantities.
    - **Close** – closes the **Report** dialog box.

    The updated report appears in the view window.

4. **Update Report** setting
    - **Real Time** checked – enable real time updates for all reports while the reports are being edited.
    - **Real Time** unchecked – enables drop-down menu to **Update All Reports** or **Update Report**. Reports will only be updated with one of these user selectable update options or upon exiting the report dialog box. This can be useful if you expect a trace to take time to display. You can then add additional traces without having to wait.
5. In the **Context** section you make selections depending on the design and solution type.
    - **Solution** field with a drop-down selection list. This lists the available setups and sweeps. As a minimum, the LastAdaptive solution and AdaptivePass solution is available to choose.

      The AdaptivePass solution context can be selected to allow any value or parameter to be plotted versus the adaptive pass. This function is usually used to evaluate the convergence of the solution.

    - *For Transient projects only*:

      Domain field with a drop-down selection list containing options for plotting vs time (Sweep Domain) or plotting vs frequency (Spectral Domain).

    - *For Transient projects only*:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

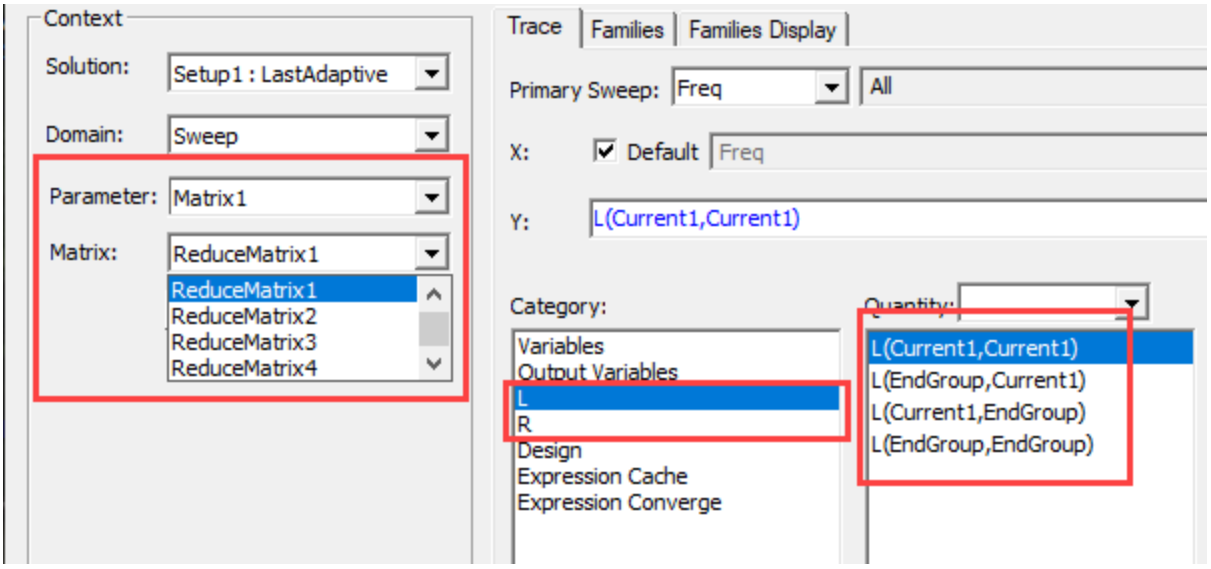
Domain field with a drop-down selection list containing options for **Average and RMS** and **Transient D-Q** for electric machines. Click **Machine Options** to setup the machine specific parameters.

- *For AC Magnetic projects only*:

  Domain field with a drop-down selection list containing options for plotting vs frequency (Sweep Domain) or plotting vs time (Time Domain).

  Matrix field with a drop-down selection list containing options for plotting *matrix* and *reduce matrix* parameters.

> **Note:** In Maxwell AC magnetic designs, the user can create *matrix* parameters, which will cause the solver to produce an impedance *matrix* for the selected excitations. In addition, the user can group (wire) two or more excitations to one excitation in either a series or parallel connection referred to as a *reduce matrix*. The **Matrix** field appears only if a matrix entry is selected in the **Parameter** field. (Refer to *Assigning a Matrix* for information on creating *matrix* and *reduced* matrix parameters for AC magnetic designs.)



- **Parameter** field with a drop down selection list. Whether this field appears, and the parameters listed depend on the Solution type and the ***<type>*** selected.
- *For Fields and Noise Vibration reports*: **Geometry** field with a drop down selection list. This applies the quantity to a specific geometry.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. The Families tab provides a way to select from valid solutions for sweeps where a simulation has multiple variables defined (for example, for a parametric sweep). If so, the variables other than the one chosen as the **X (Primary sweep)**, are listed under the **Families** tab with columns for the variable, the value, and an Edit column with an ellipsis [**...**] button. See Using Families tab for Reports.
7. In the **Y** Component section of the dialog make selections for the following:
    a. Categories – those depend on the Solution type and the design. For example, Magnetostatic categories include Torque, Output Variables, Inductance, and other user-selectable solution parameters. Transient categories include: Loss, Output Variables, Variables, and others. The selected category provides the default name of the plot, for instance S Parameter Plot *n*. You can edit the plot names in the project tree and the plot header text in the report synchronizes.
    b. Quantities for Y are relative to the selected category.

> **Note:**
>
> The Quantity text field can be used to filter the Quantity list by typing in text, or by using the four predefined selections. This is useful if the Category selected produces a lengthy Quantities list. See Filtering Quantity Selections for the Reporter.
>
> When the matrix is very large, the number of quantities can be correspondingly huge. Therefore, the Quantities field can optionally use a tree structure to divide matrix quantities into groups by their first element name. The initial display shows groups, without initially listing group members. See Report Setup Options.

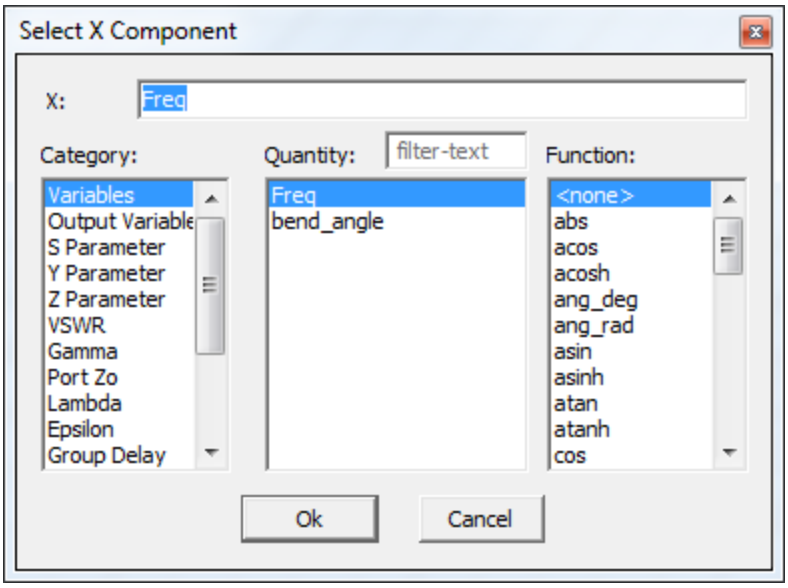
    c. **Function** list to apply to the Y quantities.
    d. Value field displays the currently specified Quantity and Function. You can edit this field directly.

> **Note:**
>
> Color shows valid expression.

    e. **Range Function** button – opens the **Set Range Function** dialog box. This applies currently specified Quantity and Function.
8. In the **X (Primary Sweep)** section, make selections for the following:
    a. Select the Primary value(s) from the drop-down menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To select an X component that is different from the Primary Sweep, uncheck the Default field to enable the X field and **Browse** [...] button. Click **Browse** [...] to display the **Select X Component** dialog box.



This lets you specify the X component as you do the Y; that is, in terms of Categories which define the selectable Quantities, and Functions to apply. After making selections, **OK** the dialog to assign the X component.

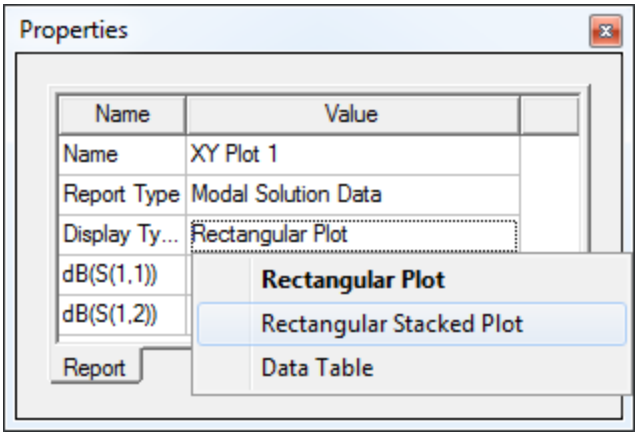
b. If sweeps are available, you can select **Browse** [...] to display a panel that lets you select Use all values, or selected sweep or sweeps, or access an Edit Sweep dialog box with further editing options. Post-Processing variables are Post-Processing sweeps/editable sweeps, so you can use the *Edit Sweep* dialog box to create your own sweep.

c. The **Families** tab provides a way to select from valid solutions for sweeps where a simulation has multiple variables defined (for example, for a parametric sweep). If so, the variables other than the one chosen as the **X (Primary sweep)**, are listed under the **Families** tab with columns for the variable, the value, and an Edit column with an ellipsis [**...**] button. See Using Families tab for Reports.

You can also view and edit the properties of Reports and their traces via their Properties windows. See Modifying the Background Properties of a Report.

You can also modify the display type of an existing plot from the Properties dialog box for that plot. Select the Report icon in the Project Manager to display the **Properties** dialog box. Selecting the Display Type field displays a menu with selections available for that plot.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Once you make a selection, the plot display updates for the current selection.

> **Note:**
>
> Remember that for many excitations of interest for plotting, you can control the default base names through the dialog box described here: Setting Default Boundary/Excitation Base Names. This may save you the need to edit individual names in the plots.

Modify Report: Using the Edit Sweep Dialog Box

Clicking **Browse** [...] at the lower right corner of the **Use all values** pane opens the **Edit Sweep** dialog box for Modify Report. This lets you edit the current Primary sweep variable values, including radio button selections for Single value, Linear step or count, and Decade, Octave, or Exponential Counts.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can specify start, stop and step values and units, and add specific values to the list of current sweep values. The **Add>>** and **Update>>** buttons let you edit the value list.

You can use the mouse click, drag, and Alt+click to select values. You can also edit individual values.

Zooming and Fitting Reports

The standard Zoom and Fit commands operate on reports. After clicking in an open report, you can also use a mouse wheel to zoom in and out.

Modifying the Background Properties of a Report

To modify the appearance of a report, or to modify the display properties any object in a report, including traces, axis labels, grids, colors, fonts, legends, color maps, contour color, and so forth:

1. Open the report you want to modify.
2. Select an editable object in the report to be able to edit its properties. Click on an object to select it and to view its properties in the docked **Properties** window. To open a floating **Properties** window, either double-click the selected object, or select **Edit** > **Properties** from the menu bar.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Properties** tabs and options displayed for editable plot objects varies depending on the report type (for example, whether 2D rectangular, 2D polar, Smith, Stacked, or 3D), but can include the following:

- **Cartesian** – controls the scroll bar and thumb properties for 2D rectangular plots.
- **Header** – controls the properties for the text displayed at the top of the report, including the Title font, Company Name, Show Design Name, Subtitle Font. The plot title is tied to the report's name and is not a Header property. If you change the report name in the Project tree, plot title synchronizes. The Company Name and the Show Design Name check box are grouped in the **Properties** dialog box as Subtitle. Edits to the Subtitle Font Property affects both of them.
- **General** – controls the background color (the perimeter around the trace display) for the plot, the contrast color (the trace display background), the Field width, the Precision, and whether to use scientific notation for marker and delta marker displays. (X and Y notation display is set separately, in the Axis property tabs.) An Auto Scale Fonts property is on by default and scales text in plots and colorkey (contour plot, field plots in 3D modeler) for high resolution screens.
- **Legend** – controls the properties for whether to include a Legend Name, Show Trace Name, Solution Name, and Variation Key. At least one of these must be selected. Legend Name is blank by default. When non-empty, a header row for the Legend in plot shows up with that string. You can also specify the File Name Display as Full Path, File Name without Path, or as an Array Index. You can also edit the Font, the background color of the Legend box, the Border Color, the Border Width, Grid Color (for the lines between Trace descriptions), and the Grid line width. See: Modifying the Legend in a Report.
- **Color Key** – for 3D plots, controls the appearance of the color key (colors, transparency, border appearance, fonts, number format, field width and precision.
- **Contour** – for 3D plots, controls the appearance of the color map, including map type, ramp color, spectrum, IsoValType, levels, number of contours, and values shown.
- **Radiation Pattern** – for 2D polar plots, controls whether to show the circular grid and angle lines.
- **Stacked** – for stacked plots, controls properties for X scrollbar, thumb properties, and stack layout, auto fit, and stack height.
- **Smith** – for Smith charts, controls whether to show grids for Imp., Adm., Cir, and angle lines.
- **Traces** – controls the properties for traces, including: Color, Line Style, Line Width, Trace Type, whether to Show a symbol, Symbol Frequency, Symbol style, whether to Fill symbol, symbol color, and whether to show arrows. You can select traces either in the Legend or on the plot. See: Editing the Display Properties of Traces.
- **Axis** for X, Y or Z, or for Phi, Theta or Rho, and circular – the defaults for most of these values (applying to 2D and 3D both) are set in the **Report 2D Options**Axis

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tab.

- Display name – check box for whether to display the axis name.
- Specify name – check box for specifying the Axis name.
- Name – this describes the axis to which the following properties/options refer. These are selected in the **Report** dialog box.
- Axis Color – set the color by double-clicking to display the **Set color** dialog box. Select a default or custom color and click OK.
- Axis Font – click the cell to display the **Edit Text Font** dialog box. The dialog box lets you select from a list of available fonts, styles, sizes, effects, colors, and script. The dialog box also contains a preview field. OK the selections to apply the font edits and to close the dialog box.
- Display Units – this specifies whether to display the axis units.
- **Window (section):**
  - Window Mode – can be Axis range, Continuous moving window, or Step moving window.
  - Window Width (in) – provide an integer value for the previous selection.
- **Manual Format** (section):
  - Number format – select from the drop down menu, Auto, Decimal, or Scientific notation.
  - Field Width – enter a real value.
  - Field Precision – enter a real value.

■ X or Y or Z **Scaling Tab** – These properties provide control over scaling:

- Axis Scaling – use the drop-down menu to select scaling as Linear or Log. For the Y axis, all zero or negative values are discarded before log scaling is applied. For 3D plots, scaling is on the Axis tabs
- Specify Min – check box
- Min – text entry in same units as axis units. Saved as SI internally.
- Specify Max – check box
- Max – text entry in same units as axis units. Saved as SI internally.
- Specify Spacing – check box
- Spacing – text entry in same units as axis units. Saved as SI internally
- **Manual Units** (section):
  - Auto Units – use the check box compute the correct units for the axis.
  - Units – click on the cell to select from a menu of available units if you have not checked Auto Units.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Infinity Visualization** (section):
  - Map Infinity Mode – check box.

    Each axis can be set to treat infinity values in a user defined way. When you check the Map Infinity Mode, any infinity values in the input data get the infinityMap value (negative infinity get the value*-1 and positive infinity the positive value specified). This can be useful if there are zeros, or very small values that Maxwelltreats as zero, in the data, for example, dB Gain.

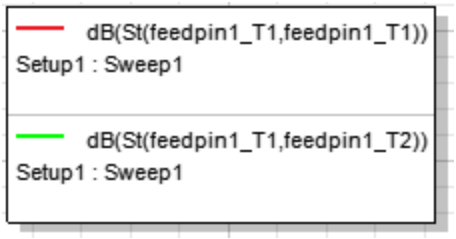
  - Map Infinity To – enter a real value for the Map Infinity Mode.

- **Grid** – properties for grid labels and grid style, appearance, line styles, color, major and minor lines, major and minor circles on polar grids, and scaling. For the 3D rectangular plots, there are separate tabs for the XY, YZ and ZX axes, and for 3D Polar plots, tabs for phi-rho, or theta-rho grids.

3. Edit the properties, and **OK** the dialog box to apply the changes.

Modifying the Legend in a Report

The legend in a report is a list of the curves being plotted. For each curve, the legend gives the name, shows the line color, and lists the setup and the adaptive pass used to generate the curve.



To show or hide a legend in a report:

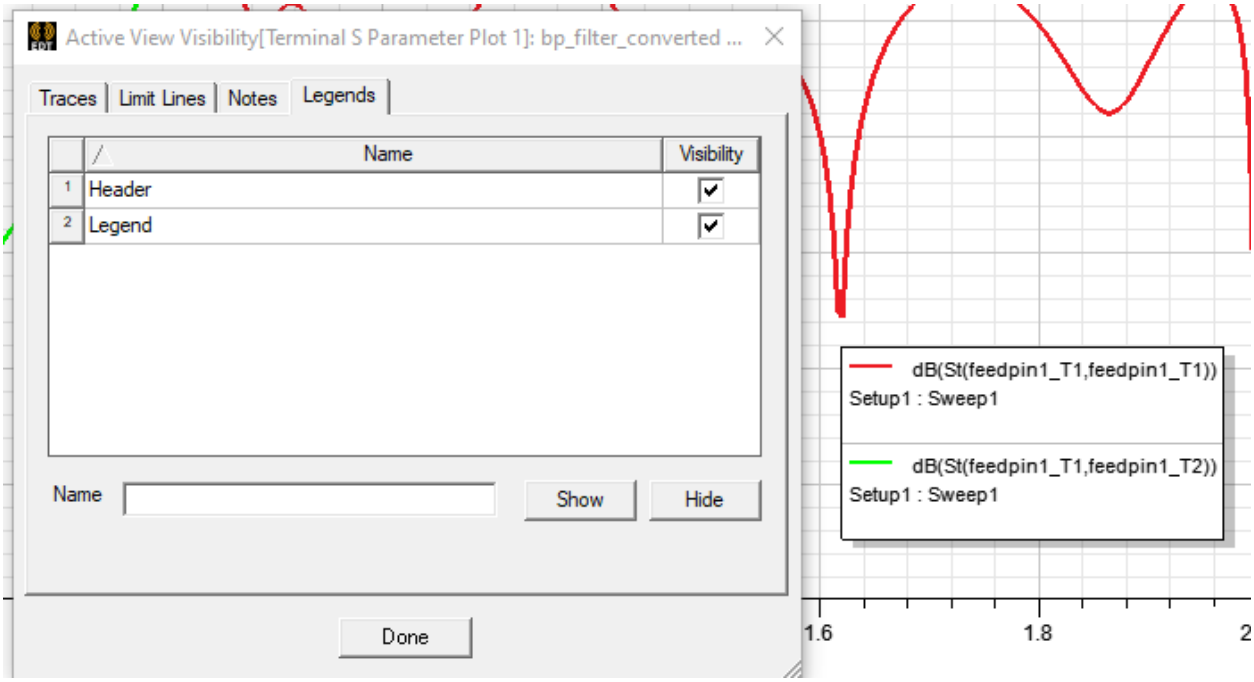
1. Make the report the active view.
2. Use **View** > **Active View Visibility** or the **Show/Hide** icons on the toolbar to display or hide the report.

   Either command displays the **Active View** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select the **Legend** tab.

   This lists the legend (or legends) in the report.



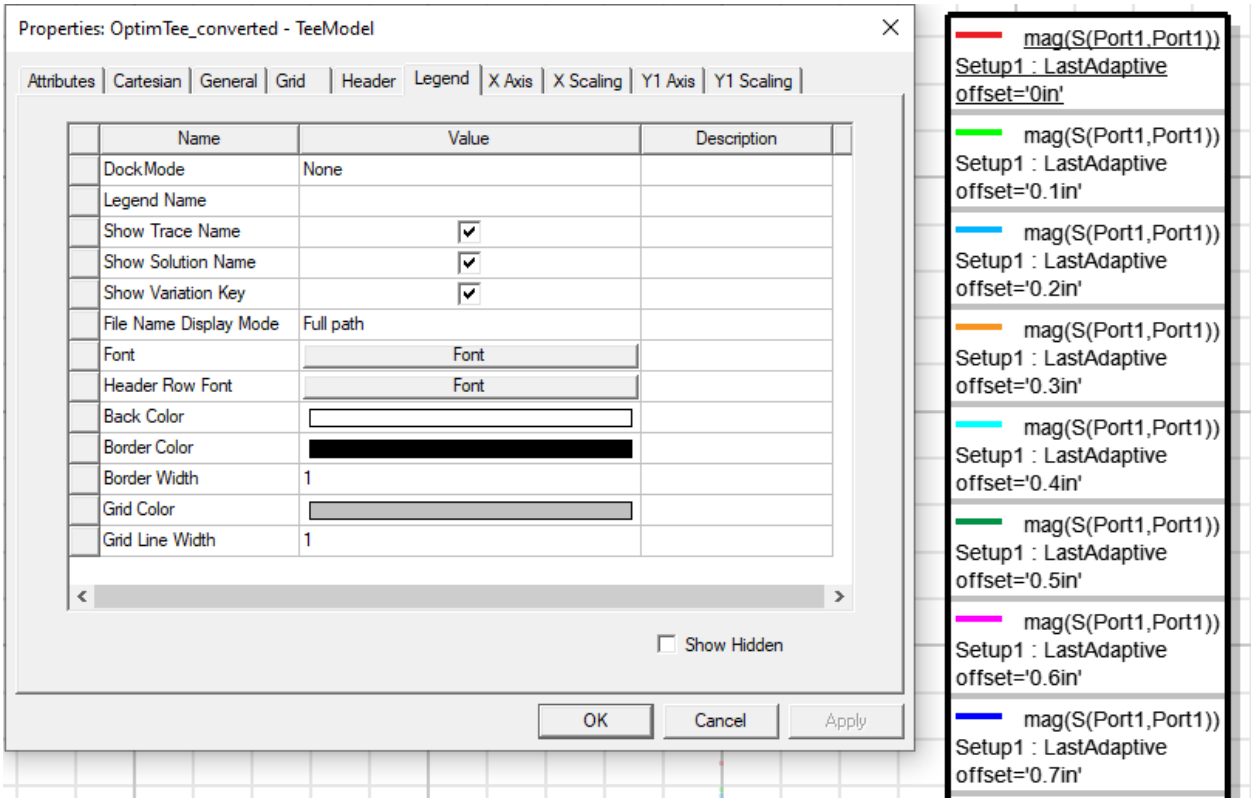
4. Enable the visibility check box, and click **OK** to close the dialog box and apply the change.

To edit the display properties of a legend:

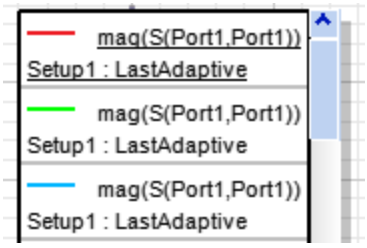
1. Double-click a legend in a report to display a docked properties window, or right-click on the legend and select **Edit** > **Properties** to display the floating **Properties** dialog box.

   This lets you edit the Properties for Dock Mode, Legend Name (default is no name. When non-empty, a header row for the Legend in plot shows up with that string.), whether to Show Trace Name, Solution Name, and Variation Key (which applies to parametric variables, if present). If none of these three are selected, only a trace color shows.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Here is an example with Variation Key off.



You can also edit the Font by clicking the Font cell to display the **Edit Text Font** dialog box. The dialog box lets you select from a list of available fonts, styles, sizes, effects, colors, and script. The dialog also contains a preview field. OK the selections to apply the font edits and to close the dialog box.

You can also edit the background color of the Legend box, the Border Color, the Border Width, Grid Color (for the lines between Trace descriptions), and the Grid line width.

2. Click **OK** to close the **Properties** dialog box and apply the selections.

To change the display name for traces, see: Editing Trace Properties.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To move a legend in a report:

1. Click and hold on the legend.

   The cursor changes to crossed lines with arrow tips.

2. Still holding, drag the legend to a new location and release.

   The legend is released and the crossed lines change back to a mouse pointer.

To resize a legend in a report:

1. Position the mouse tip over the edge you want to resize.

   The mouse pointer changes to a horizontal or vertical line with arrow tips.

2. Click and drag the horizontal or vertical edge to the desire size.
3. Release.

Creating Custom Report Templates and Defaults

You can edit properties from any report type and save it as a template or as the default. This can save repeated editing of properties (for example, the company name, or color schemes and plot attribute settings) when you create other reports. You can prepare a template by copy/pasting plots settings from one plot to another of the same display type. Once you create templates, you can access them from the **Results** > **Report Templates** > menu and the **Report2D** > **Report Templates** > **Apply Settings** menu.

See Modifying the Background Properties of a Report for a discussion of format changes you can make to any report.

To save an edited report as a template:

1. In the Project Tree, right-click the report name of interest to display the shortcut menu and click **Report Templates** > **Save...**. You can also click **Report2D** > **Report templates** > **Save...** or **Report3D** > **Save As Template...**

   This displays the **Save As Report Template** file browser. By default, the directory is your AnsysEM\<*productName*>\userlib\ReportTemplates directory. You can also save to the SysLib directory.

2. Typically, you accept the directory.
3. You must provide a file name, which will be given an *.rpt extension.

   It is good practice to give the template a descriptive name, showing both the kind of format you begin with (such as XY Plot or 3D Plot) and apt description of the distinguishing edits (such as for company name, or color scheme). Once, saved, this name will appear on the *PersonalLib* menu.

   The **Save As Type** field currently supports the Ansoft Report Format (*rpt) format.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **Save** to save the template to the PersonalLib menu.

All *.rpt templates in the userLib directory appear on the **Results> Report Templates> PersonalLib** menu. Selecting a report from the PersonalLib menu opens a report that you can then Modify to add traces or perform other edits. Templates in the SysLib directory appear on the **Report Templates** menu.

To save an edited report as a default:

1. In the Project Tree, right-click the report name of interest to display the shortcut menu and click **Report Templates** > **Save Settings as default**. You can also click **Report2D** > **Report templates** > **Save Settings as default** or **Report3D** > **Save Settings as default**.

Selecting the Report Type

The **Report Types** available for creating a report depends on the simulation setup. The Report Category provides the default report name for the project tree and the text displayed in the report header. For example, Force as the Category type causes the default report name to be Force Plot *n*. Depending on the design setup, you can make a selection from the following report types.

Select from the following report types:

Magnetostatic	Project variables, user-specified Output Variables and Design data are available to plot. Depending upon the solution parameters requested, torque data may be available to plot, as well as Inductance, and magnetic flux. Quantities such as Lnom, and MagFluxNom may also be available for plotting. See Post processed Quantities for more details.
Electrostatic	Project variables, user-specified Output Variables and Design data are available to plot. Depending upon the solution parameters requested, torque data may be available to plot, as well as Capacitance.
AC Magnetic T-Omega	Project variables, user-specified Output Variables and Design data are available to plot. Depending upon the solution parameters requested, torque data may be available to plot, as well as Inductance, Resistance, Impedance. Quantities such as Lnom, MagFlux, and various Winding, Loss, and End Connection quantities may also be available for plotting, depending on the design. See Postprocessed Quantities for more details.
AC Magnetic A-Phi	Project variables, Output Variables, and Design data are available for plotting, as well as Force, Torque, Loss, and solution process data. Terminal Voltage, Terminal Current, Inductance, Resistance, Conductance, and Capacitance quantities may also be available for plotting, depending on the design.See Postprocessed Quantities for more details.
DC Conduction	Project variables, Output Variables, and Design data are available for plotting, as well as Resistance matrix data.

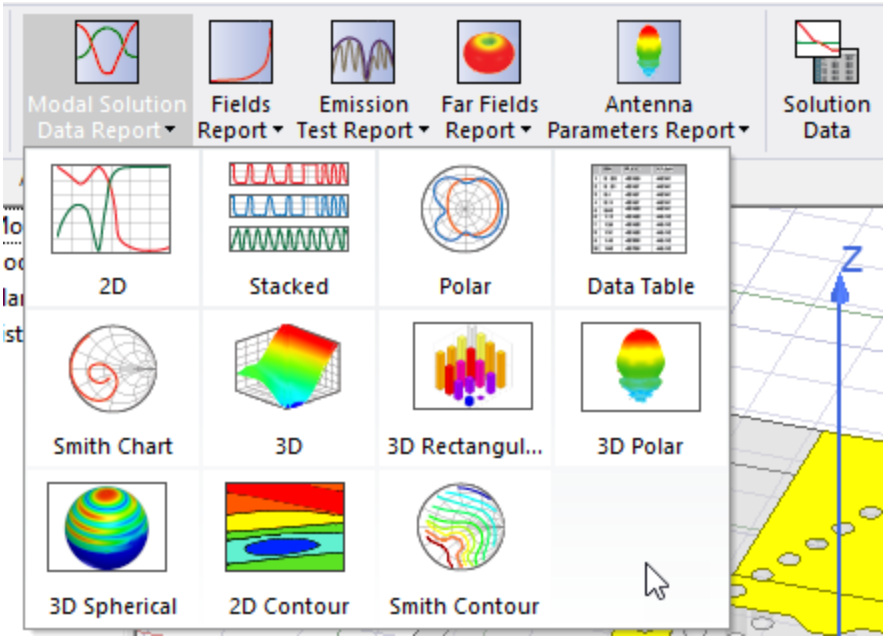
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AC Conduction (2D Only)	Project variables, Output Variables, and Design data are available for plotting. Depending upon the solution parameters requested, Capacitance, Conductance, and Admittance quantities may also be available for plotting.
Transient	Project variables, Output Variables, and Design data are available for plotting, as well as Force, Torque, Loss, Winding, and solution process data.
Transient A-Phi Formulation (3D only)	Project variables, Output Variables, and Design data are available for plotting, as well as Force, Torque, Loss, Winding, and solution process data. Terminal Voltage, Terminal Current, Inductance, and Capacitance quantities may also be available for plotting, depending on the design.
Electric Transient (3D Only)	Project variables, Output Variables, and Design data are available for plotting, as well as Force, Torque, Loss, and solution process data.
Fields	Project variables, Output Variables, and Design data are available for plotting, as well as calculator expressions.
Noise Vibration	Project variables, Output Variables, and Design data are available for plotting, as well as calculator expressions.
Harmonic Force	Project variables and Output Variables, are available for plotting, as well as harmonic force data (radial, tangential, and axial complex values of harmonic force are available for transient 3D designs – only radial and tangential for transient 2D designs).

Selecting the Display Type

The information in a report can be displayed in several formats. When you select the **Results** tab in the ribbon, depending on reports available for a given design and Solution type, you see a selection of report type groups (for example, Modal Solution Data, or Fields Report, Emission Test, Antenna Parameters, Near Fields, Terminal Solution Report, Far Fields Report, Characteristic Mode Data, etc.), with drop-down menus that displays the kinds of reports available for the design in that group:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

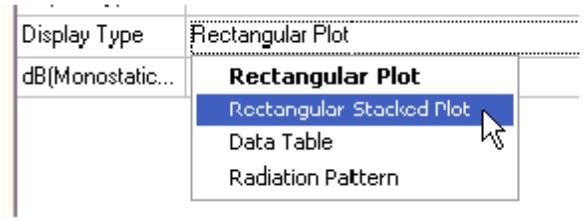


For the initial plot, you can also select from the following **Display Type** formats in the **Create<*type*> Report** submenu:

Rectangular Plot	A 2D rectangular (x-y) graph.
Rectangular Stacked Plot	This choice puts each trace into its own 2D rectangular plot, and stacks each plot, rather than overlaying the traces on the same plot.
3D Rectangular Plot	A 3D rectangular (x-y-z) graph.
3D Rectangular Bar Plot	A 3D plot of rectangular bars.
Rectangular Contour Plot	A rectangular (x-y-z) graph. Contour plots are useful to visualize surfaces (e.g., Directivity as a function of phi/theta).
Data Table	A grid with rows and columns that displays, in numeric form, selected quantities against a swept variable or another quantity.

You can also modify the display type of an existing plot from the **Properties** dialog for that plot. Select the Report icon in the Project tree to display the Properties dialog box. Selecting the **Display Type** field displays a menu with selections available for that plot.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Once you make a selection, the plot display updates for the current selection.

Creating 2D Rectangular Plots

A rectangular plot is a 2D, x-y graph of results.

1. On the **Results** menu (under the **Maxwell** menu, or right-click **Results** in the Project Manager), click **Create < *type* > Report**, and select **Rectangular Plot**.

   The *Report* dialog box appears.

2. In the **Context** section make selections from the following field or fields, depending on the design and solution type:
   a. Solution field with a drop down selection list. This lists the available solutions, whether sweeps or adaptive passes.
   b. Domain field with a drop down selection list. Whether this field appears, and the domains listed, depend on the Solution type and the report <*type*> selected.
   c. Geometry field with a drop-down selection list. For field and radiated field reports, this applies the quantity to a geometry or radiated field setup.
3. Under the **Trace** tab, **Y** component section, specify the information to plot along the y-axis:
   a. In the **Category** list, click the type of information to plot. The category selected provides the default plot name.
   b. In the **Quantity** list, click the value to plot.
   c. In the **Function** list, click the mathematical function of the quantity to plot.
   d. Value field displays the currently specified Quantity and Function. You can edit this field directly.

      > **Note:**
      >
      > Color shows valid expression.

   e. **Range Function** button -- opens the **Set Range Function** dialog box. This applies currently specified Quantity and Function.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

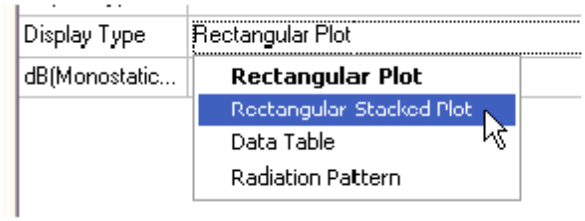
4. On the **Trace** tab, **X** (Primary sweep) line, specify the quantity to plot along the x-axis in one of the following ways:
   - Select the sweep variable to use from the drop-down list.
   - If sweeps are available, you can select the browse button to display a dialog that lets you select particular sweep or sweeps, or all sweeps. The quantity will be plotted against the primary sweep variable listed.
5. On the **Families** tab, confirm or modify the sweep variables that will be plotted.
6. Click **New Report**.

   This creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog.

   The function of the selected quantity will be plotted against the swept variable values or quantities you specified on an x-y graph. The plot is listed under **Results** in the project tree and the traces are listed under the plot. The default name is based on the Report Category you selected, (for example, S Parameter Plot *n* or rE Plott *n*). You can edit the plot names in the project tree and the plot header text in the report synchronizes. When you select the traces or plots, their properties are displayed in the Properties window. These properties can be edited directly to modify the plot.

7. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

You can also modify the display type of an existing plot from the Properties dialog for that plot. Select the Report icon in the Project tree to display the Properties dialog box. Selecting the Display Type field displays a menu with selections available for that plot.



Once you make a selection, the plot display updates for the current selection.

Creating a 2D Rectangular Stacked Plot

A rectangular stacked plot is a 2D, x-y graph of results, with each trace displayed on a separate plot.

1. On the **Results** menu (Maxwell menu or right-click on **Results** on the Project tree), click **Create <*type*> Report**, and select **Rectangular Stacked Plot**.

   The **Report** dialog appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the **Context** section make selections from the following field or fields, depending on the design and solution type.

    a. **Solution** field with a drop down selection list. This lists the available setups and sweeps. As a minimum, the LastAdaptive solution and AdaptivePass solution is available to choose.

    The AdaptivePass solution context can be selected to allow any value or parameter to be plotted versus the adaptive pass. This function is usually used to evaluate the convergence of the solution.

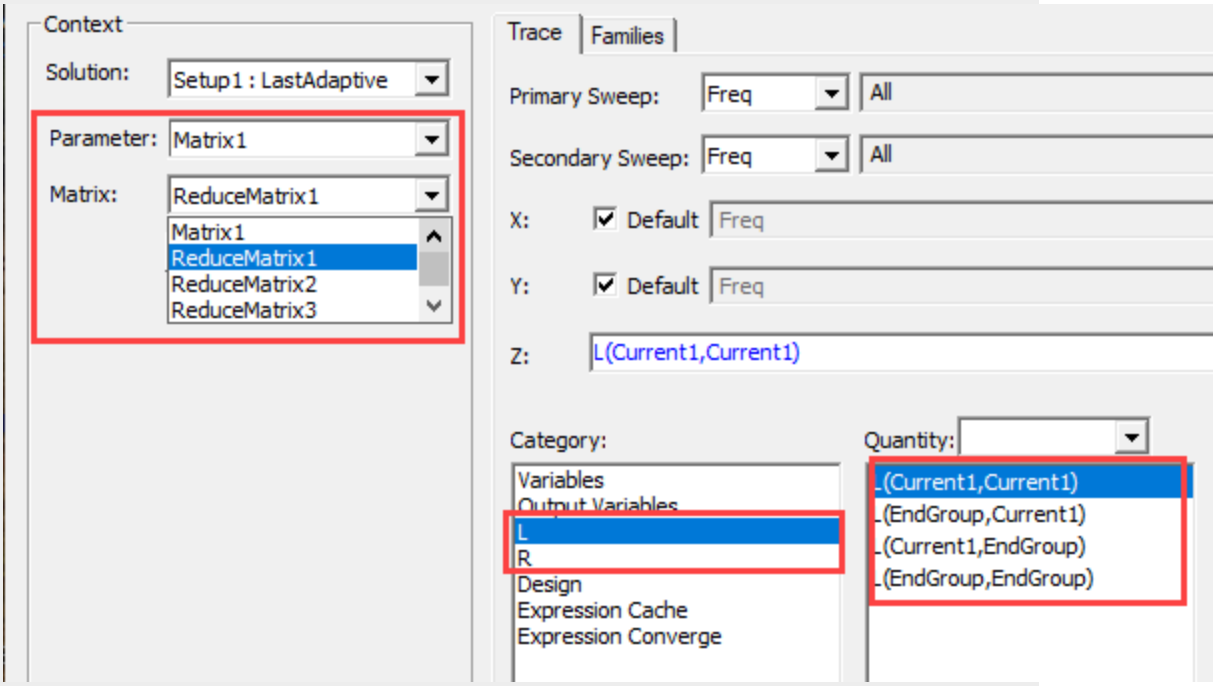
    b. **Parameter** field with a drop down selection list. Whether this field appears, and the parameters listed depend on the Solution type and the ***<type>*** selected.

    c. *For AC Magnetic projects only*:

    Matrix field with a drop-down selection list containing options for plotting *matrix* and *reduce matrix* parameters.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: In Maxwell AC Magnetic designs, the user can create *matrix* parameters, which will cause the solver to produce an impedance *matrix* for the selected excitations. In addition, the user can group (wire) two or more excitations to one excitation in either a series or parallel connection referred to as a *reduce matrix*. The **Matrix** field appears only if a matrix entry is selected in the **Parameter** field. (Refer to *Assigning a Matrix* for information on creating a *matrix* and *Assigning a Reduce Matrix* for information on creating *reduced* matrix parameters for AC Magnetic designs.)



d. *For Fields and Noise Vibration reports*: **Geometry** field with a drop down selection list. This applies the quantity to a specific geometry.

3. Under the **Trace** tab, **Y** component section, specify the information to plot along the y-axis:
   a. In the **Category** list, click the type of information to plot.
   b. In the **Quantity** list, click the value to plot.
   c. In the **Function** list, click the mathematical function of the quantity to plot.
   d. Value field displays the currently specified Quantity and Function. You can edit this field directly.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** Colors are used to indicate a valid expression.

e. **Range Function** button opens the **Set Range Function** dialog box, which applies the currently specified Quantity and Function.

4. On the **Trace** tab, **X** (Primary sweep) line, specify the quantity to plot along the x axis in one of the following ways:
   - Select the sweep variable to use from the drop down list.
   - If sweeps are available, you can also select the browse [...] button to display a dialog box that lets you select particular sweep values, specify a range of sweep values (for Time sweeps), or **Use all values** (the default setting). The quantity will be plotted against the primary sweep variable listed.
5. On the **Families** tab, confirm or modify the sweep variables that will be plotted.
6. Click **New Report.**

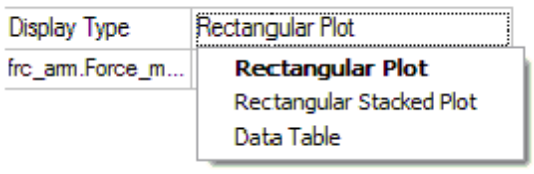
   This creates a new report in Project Manager tree, displays the report with the defined trace, and enables the **Add Trace** button on the **Report** dialog box. The default name is based on the Report Category you selected, (for example, Force Plot *n* or Output Variables Plot *n*). You can edit the plot names in the project tree and the plot header text in the report synchronizes.

   The function of the selected quantity will be plotted against the swept variable values or quantities you specified on an x-y graph. The plot is listed under **Results** in the project tree and the traces are listed under the plot. When you select the traces or plots, their properties are displayed in the Properties window. These properties can be edited directly to modify the plot.

7. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

The new trace will appear as a separate plot “stacked” above the first plot. The stacked plots share a common x axis scale. However, each plot has its own independent y axis scale.

You can also modify the display type of an existing plot from the Properties window for that plot. Select the Report icon in the Project tree to display the Properties window. Selecting the Display Type field displays a menu with selections available for that plot.

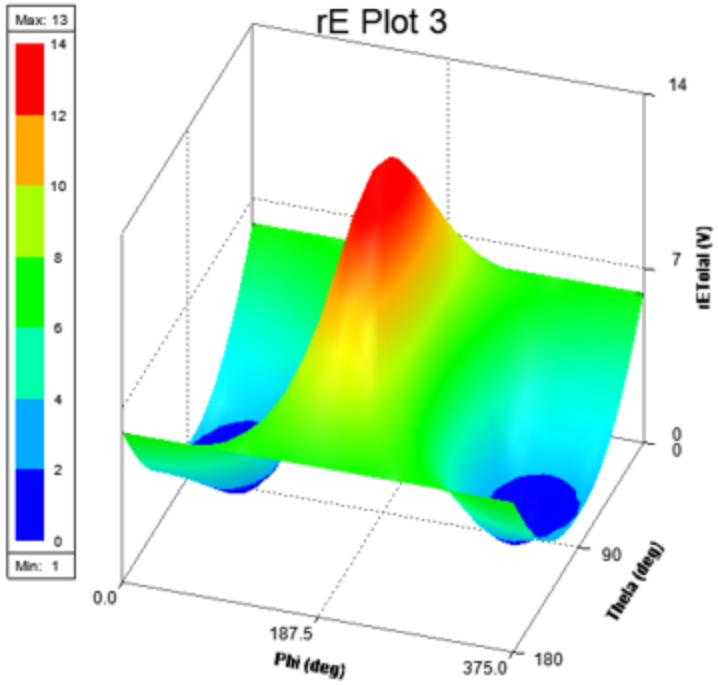


Once you make a selection, the plot display updates for the current selection.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating 3D Rectangular Plots

This is a 3D, x-y-z graph of results.



Working with a 3D Rectangular Plot

You can Rotate, Zoom, and Pan a plot. When you rotate, the Cartesian grid responds so that the curve always remains in front and the grids behind.

Clicking on a plot entity selects it, highlighting the selected entity in bold.

Double-clicking anywhere in the plot brings up the Properties dialog box. The properties are grouped appropriately under various tabs, which correspond to plot entities:

- General: For general plot properties such as Visual Detail level and background color
- Header: Properties related to plot Header/Title.
- Axis [X|Y|Z]: Properties related to the 3 axes
- Grid [XY|YZ|ZX]: Properties related to the 3 grids
- ColorKey: Properties related to ColorKey, including borders, background, Min and Max, as well as number format and precision.
- Contour: Properties related to contouring of all curves/surfaces
- Surface: Properties related to the curve

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting a property also displays its properties in the Property window. You can edit the properties to customize the appearance of the plot. See "Controlling Visual Detail in a 3D Plot" on the facing page.

Creating a 3D Rectangular Plot

1. On the **Results** menu (under the **Maxwell** menu or right-click **Results** in the Project Manager), click **Create *<type>* Report**, and select **3D Rectangular plot** from the report type menu.

   The *Report* dialog box appears.

2. In the **Context** section make selections from the following field or fields, depending on the design and solution type.
   a. Solution field with a drop-down selection list. This lists the available solutions, whether sweeps or adaptive passes.
   b. Geometry field with a drop-down selection list. For field and radiated field reports, this applies the quantity to a geometry or radiated field setup.
3. Under the **Trace** tab **Z** Component area, specify the information to plot along the z-axis:
   a. In the **Category** list, click the type of information to plot. The category you select provides the default plot name.
   b. In the **Quantity** list, click the value to plot.
   c. In the **Function** list, click the mathematical function of the quantity to plot.
   d. The **Value** field displays the currently specified Quantity and Function. You can edit this field directly.

      > **Note:**
      >
      > Color shows valid expression.

   e. **Range Function** button -- opens the *Set Range Function* dialog box. This applies currently specified Quantity and Function.
4. On the **Trace** tab **Y** (Secondary sweep) lines, specify the information to plot along the y-axis in one of the following ways:
   - Select the sweep variable to use from the drop-down list.
   - If sweeps are available, you can select the browse button to display a dialog that lets you select particular values. The quantity will be plotted against the primary sweep variable listed.
5. On the **Trace** tab **X** (Primary sweep) lines, specify the information to plot along the x-axis in one of the following ways:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select the sweep variable to use from the drop-down list.
- If sweeps are available, you can select the browse button to display a dialog that lets you select particular values. The quantity will be plotted against the primary sweep variable listed.

6. On the **Families** tab, confirm or modify the sweep variables that will be plotted.
7. Click **New Report.**

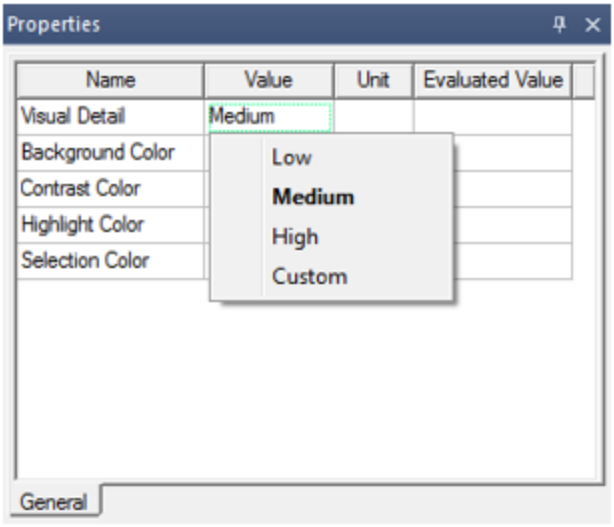
   This creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog box. The default name is based on the Report Category you selected, (for example, S Parameter Plot *n* or rE Plot *n*). You can edit the plot names in the project tree and the plot header text in the report synchronizes.

   The function of the selected quantity or quantities will be plotted against the values you specified on an x-y-z graph. The plot is listed under **Results** in the Project Manager. When you select the traces or plots, axis or grid labels, plot header, color key, or variable labels, their properties are displayed in the Properties window. The properties for each plot element can be edited directly to modify the plot content and appearance. See Modifying the Background Properties of a Report.

8. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

Controlling Visual Detail in a 3D Plot

If a particular plot seems busy with information, you can edit plot properties, such as Axis and Grid Attributes for discrete levels of visual detail to improve readability. Double-click anywhere on a plot to display the Properties dialog box. The Visual Detail property on the General tab also provides control suited to different screen and plot sizes.

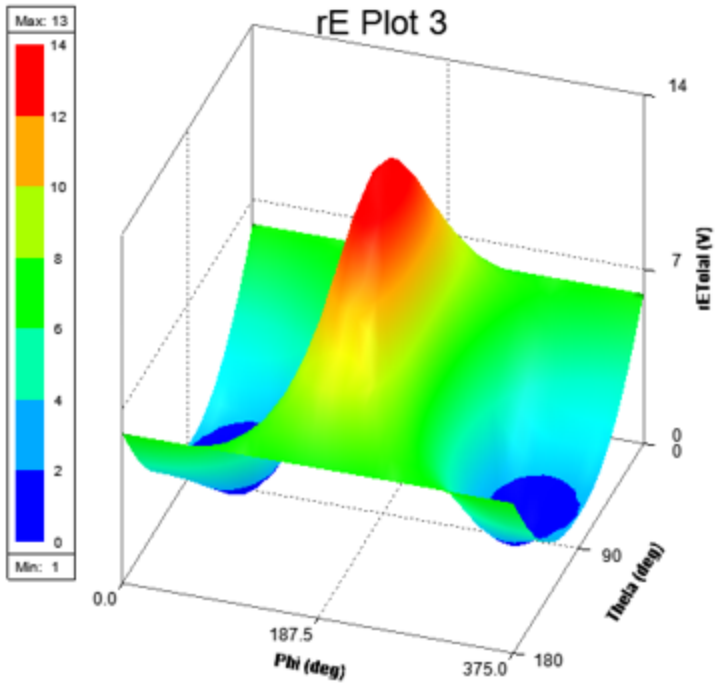


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Visual Detail menu has four options: Low, Medium (the default), High, and Custom. If you select any Visual Detail, the 3D plot is rendered according to the selected Visual Detail level and the properties reflect the values chosen for the selected visual detail level. From this predefined visual detail level, if you modify any properties, Visual Detail is automatically set to Custom (or to another predefined visual detail level if the edits happen to match the settings for that level).

You can also manually set Visual Detail to Custom. In such a case, Custom will inherit property values corresponding to the previous level. This ensures that you can customize settings starting from a baseline provided by the preconfigured Low, Medium or High Visual Detail levels.

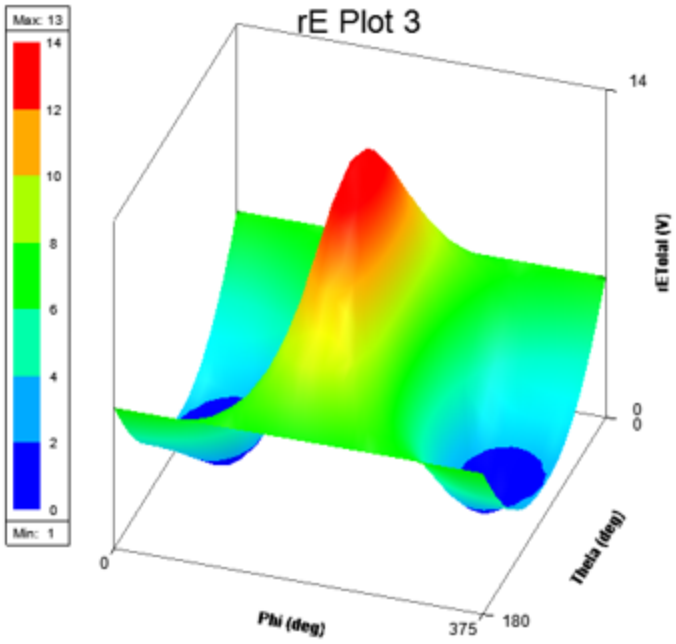
3D Rectangular Plot with Medium Visual Detail



On creation, a 3D Rectangular Plot has Visual Detail set to Medium and looks and feels as shown above. Specifically, under Medium Visual Detail level, a 3D Rectangular Plot has 3 ticks per axis (X, Y, Z axis) which will show min, max and middle value. This setting also shows axes labels.

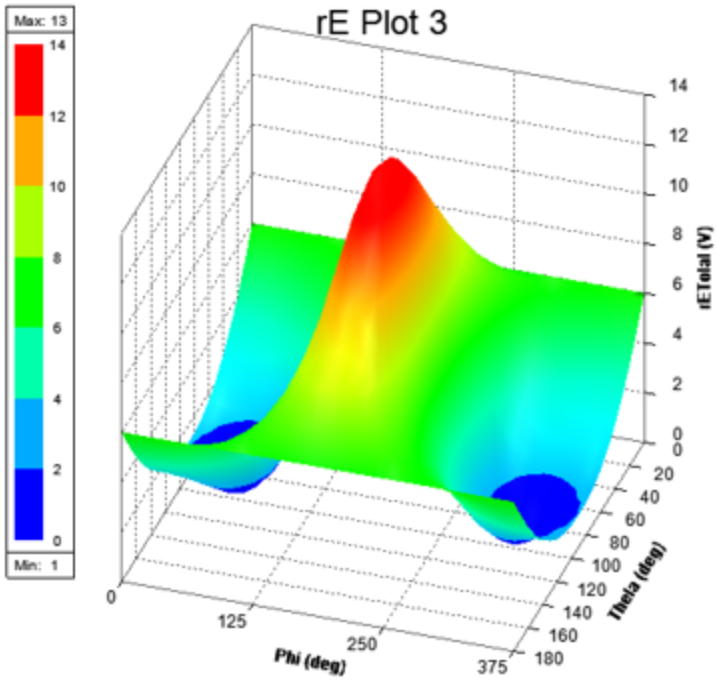
3D Rectangular Plot with Low Visual Detail

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



With Visual Detail set to Low, a 3D Rectangular Plot shows axes with 2 ticks corresponding to min and max values. It also shows axes labels and grid borders.

3D Rectangular Plot with High Visual Detail

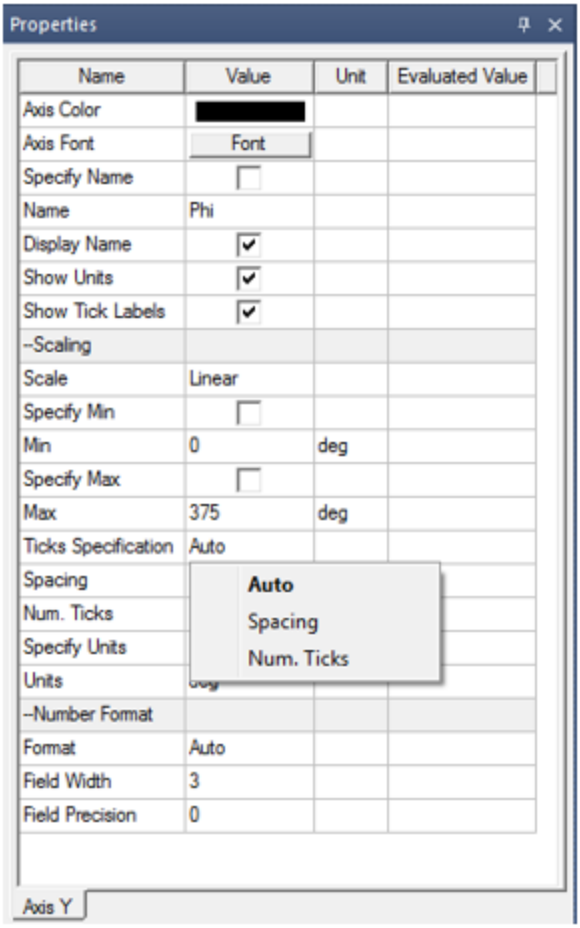


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

With Visual Detail set to High, a 3D Rectangular Plot shows all Cartesian axes and grids together with all ticks and axes labels.

Axis Properties: Ticks Specification and Num. Ticks

Ticks Specification is available on Axis properties, as shown below:



Ticks Specification is a menu with possible values as Auto, Spacing, and Num. Ticks, with Auto being the default value. If Ticks Specification is Auto, then a spacing value is automatically calculated and used to calculate and display the tick labels. **Spacing** shows the calculated value, and Num. Ticks shows the number of ticks based on this spacing value, as shown below:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

--Scaling			
Scale	Linear		
Specify Min	☐		
Min	0	deg	
Specify Max	☐		
Max	375	deg	
Ticks Specification	Auto		
Spacing	125	deg	
Num. Ticks	2		

You can edit the **Spacing** field when Ticks Specification is set to Spacing; otherwise, it is read only.

You can edit the **Num. Ticks** field when Ticks Specification is Num. Ticks; otherwise, it is read only.

Valid Num. Ticks are between 0 and 100, including 0 and 100. If you enter an invalid value, an error message is shown. If you enter a spacing value that results in number of ticks greater than 100, then an appropriate value is shown.

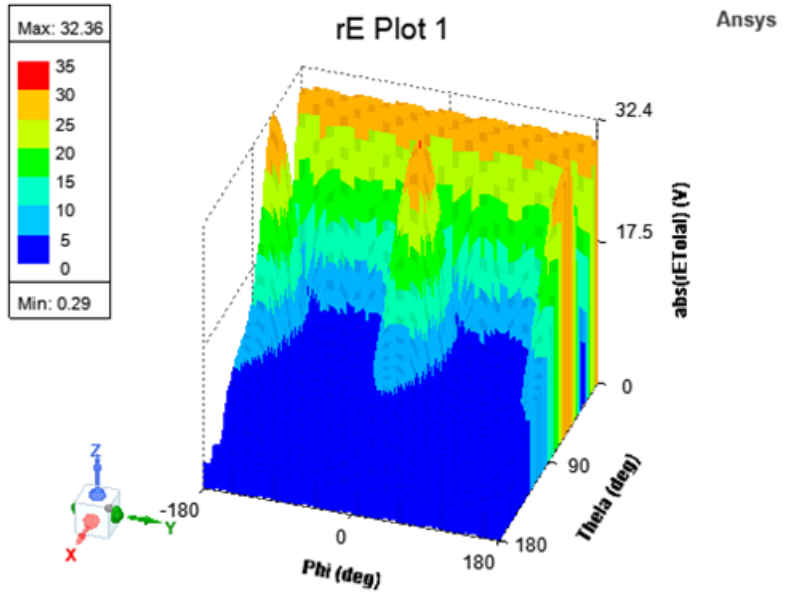
- If Num. Ticks is 0, then no ticks are shown on the axis.
- If Num. Ticks is 1, then only the max value tick is shown on the axis.
- If Num. Ticks is 2, then only the min and max value ticks are shown on the axis.
- If Num. Ticks is greater than 2, then evenly spaced ticks (including min and max) are shown on the axis.

> **Note:**
>
> With the addition of the Ticks Specification property to Axis properties, the **Specify Spacing** property was removed as an Axis property.
>
> - If an R18.0 or R18.1 project is opened with **Specify Spacing** as Unchecked, Ticks Specification is set to Auto.
> - If an R18.0 or R18.1 project is opened with **Specify Spacing** as Checked, Ticks Specification is set to Spacing.

Creating 3D Rectangular Bar Plots

This is a 3D graph of results as rectangular bars.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

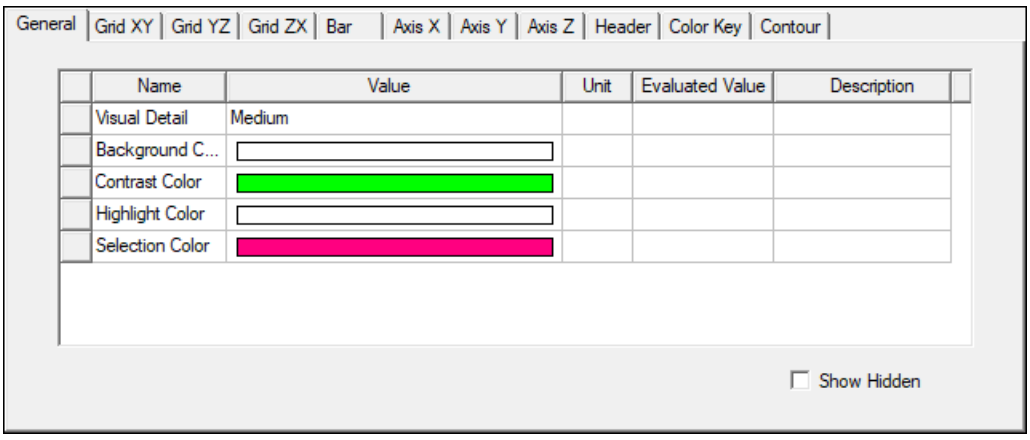


Working with a 3D Rectangular Plot

You can Rotate, Zoom, and Pan a plot. When you rotate, the Cartesian grid responds so that the curve always remains in front and the grids behind.

Clicking on a plot entity selects it, highlighting the selected entity in bold.

Double-clicking anywhere in the plot brings up the **Properties** dialog box.



The properties tabs for a 3D Rectangular Bar plot are

- **General**: For general plot properties such as Visual Detail level and background color
- **Grid [XY|YZ|ZX]**: Properties related to the three grids

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Bar**: Properties related to rectangular bars.
- **Axis [X|Y|Z]**: Properties related to the three axes
- **Header**: Properties related to plot Header/Title.
- **ColorKey**: Properties related to ColorKey, including borders, background, Min and Max, as well as number format and precision.
- **Contour**: Properties related to contouring of all surfaces

Selecting a property also displays its properties in the Property window. You can edit the properties to customize the appearance of the plot. See "Controlling Visual Detail in a 3D Rectangular Bar Plot" on page 24-82 .

Creating a 3D Rectangular Bar Plot

1. On the **Results** menu (under the **Maxwell** menu or right-click **Results** in the Project Manager), click **Create *<type>* Report**, and select **3D Rectangular Bar plot** from the report type menu, or select the Results tab of the ribbon, and select the 3D Rectangular Bar icon for the Report type you want to create.

   The *Report* dialog box appears.

2. In the **Context** section make selections from the following field or fields, depending on the design and solution type.
   a. Solution field with a drop-down selection list. This lists the available solutions, whether sweeps or adaptive passes.
   b. Geometry field with a drop-down selection list. For field and radiated field reports, this applies the quantity to a geometry or radiated field setup.
3. Under the **Trace** tab **Z** Component area, specify the information to plot along the z axis:
   a. In the **Category** list, click the type of information to plot. The category you select provides the default plot name.
   b. In the **Quantity** list, click the value to plot.
   c. In the **Function** list, click the mathematical function of the quantity to plot.
   d. The **Value** field displays the currently specified Quantity and Function. You can edit this field directly.

      > **Note:** Color indicates a valid expression.

   e. **Range Function** button -- opens the *Set Range Function* dialog box. This applies currently specified Quantity and Function.
4. On the **Trace** tab **Y** (Secondary sweep) lines, specify the information to plot along the y-axis in one of the following ways:
   - Select the sweep variable to use from the drop-down list.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If sweeps are available, you can select the browse button to display a dialog that lets you select particular values. The quantity will be plotted against the primary sweep variable listed.

5. On the **Trace** tab **X** (Primary sweep) lines, specify the information to plot along the x-axis in one of the following ways:
   - Select the sweep variable to use from the drop-down list.
   - If sweeps are available, you can select the browse button to display a dialog that lets you select particular values. The quantity will be plotted against the primary sweep variable listed.
6. Click **New Report.**

   This creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog box. The default name is based on the Report Category you selected, (for example, S Parameter Plot *n* or rE Plot *n*). You can edit the plot names in the project tree and the plot header text in the report synchronizes.

   The function of the selected quantity or quantities will be plotted against the values you specified on an x-y-z graph. The plot is listed under **Results** in the Project Manager. When you select the traces or plots, axis or grid labels, plot header, color key, or variable labels, their properties are displayed in the Properties window. The properties for each plot element can be edited directly to modify the plot content and appearance. See Modifying the Background Properties of a Report.

7. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

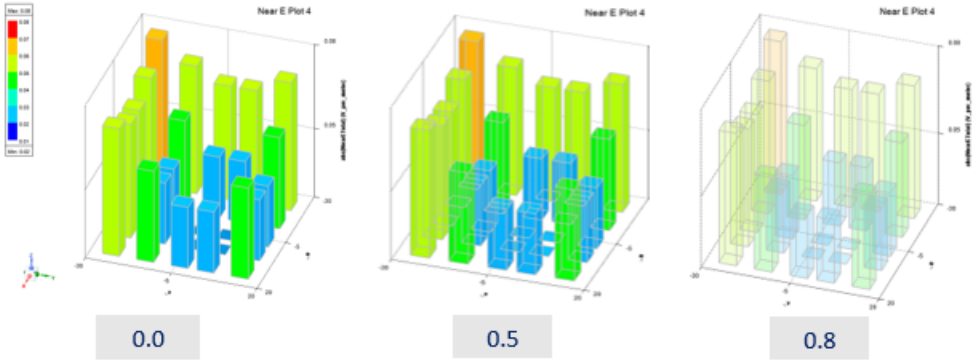
3D Rectangular Bar Customization

Selecting a bar in the plot shows the customizable attributes of bars in a 3D Rectangular Bar plot.

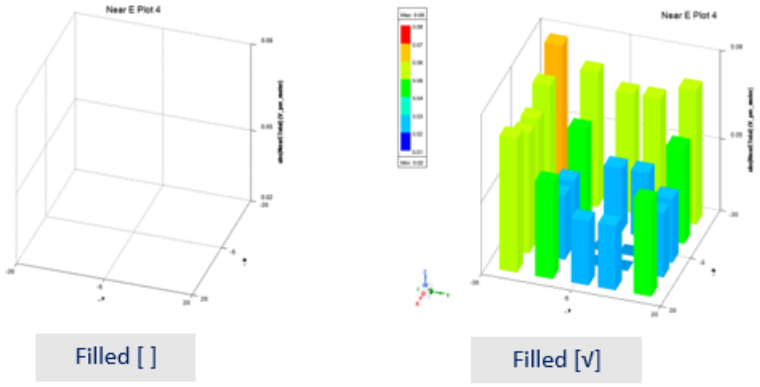
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The customizable attributes of the bars in a 3D Rectangular Bar plot include

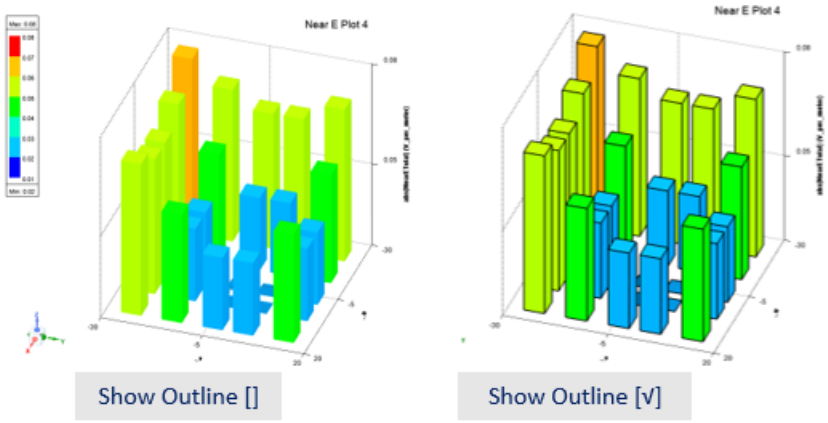
- Transparency



- Filled



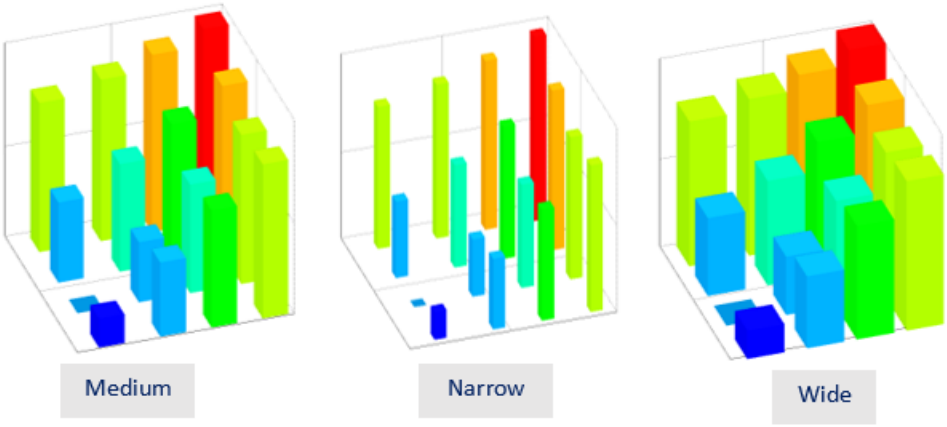
- Show Outline



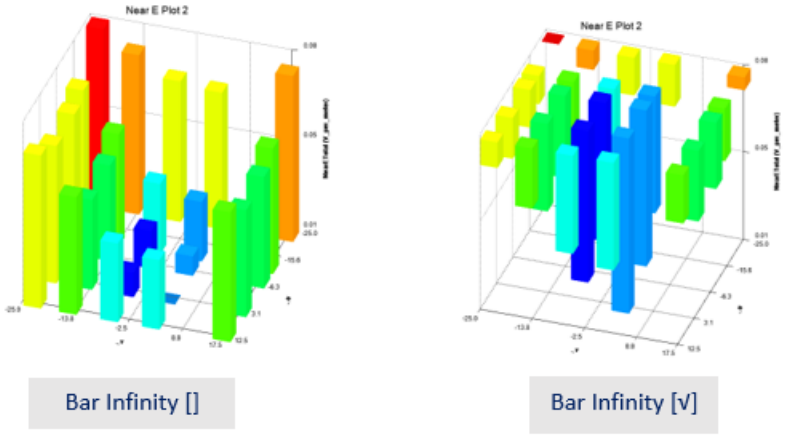
- Outline Color (customizable, see plot above)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Outline Width (customizable, see plot above)
- Bar Width, controls the thickness of the bar. Medium (default), Wide and Narrow are the width that equals to ½, ¾ and ¼ of the min spacing between adjacent points respectively.



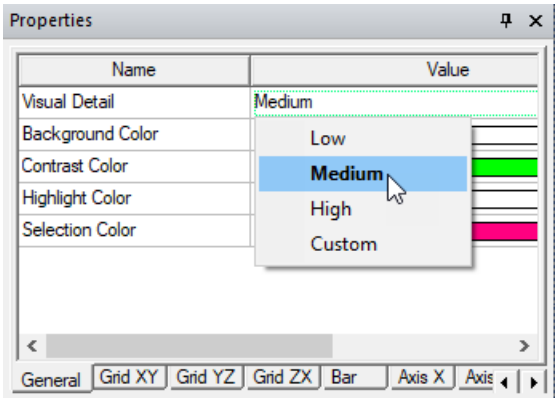
- Bar Infinity represents the third coordinate (z) using the bottom of the bar.



Controlling Visual Detail in a 3D Rectangular Bar Plot

If a particular plot seems busy with information, you can edit plot properties, such as Axis and Grid Attributes for discrete levels of visual detail to improve readability. Double-click anywhere on a plot to display the Properties dialog box. The Visual Detail property on the General tab also provides control suited to different screen and plot sizes.

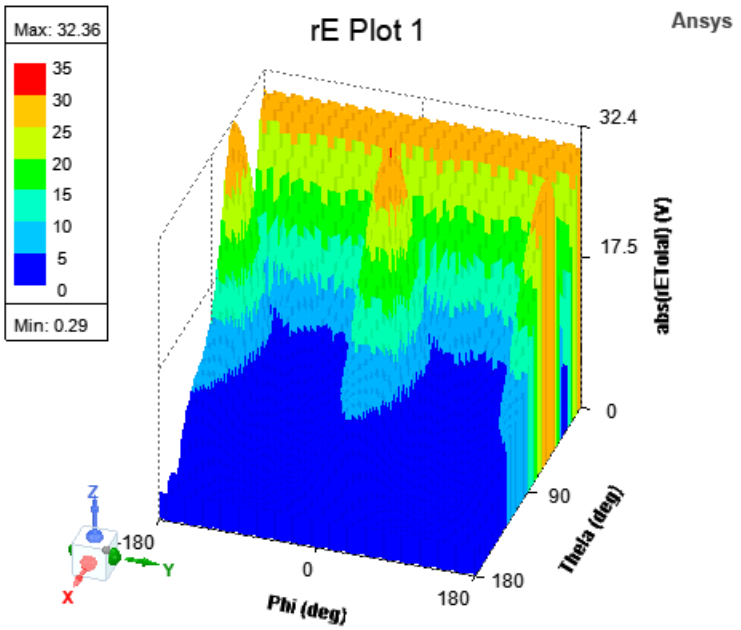
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Visual Detail menu has four options: Low, Medium (the default), High, and Custom. If you select any Visual Detail, the 3D plot is rendered according to the selected Visual Detail level and the properties reflect the values chosen for the selected visual detail level. From this predefined visual detail level, if you modify any properties, Visual Detail is automatically set to Custom (or to another predefined visual detail level if the edits happen to match the settings for that level).

You can also manually set Visual Detail to Custom. In such a case, Custom will inherit property values corresponding to the previous level. This ensures that you can customize settings starting from a baseline provided by the preconfigured Low, Medium or High Visual Detail levels.

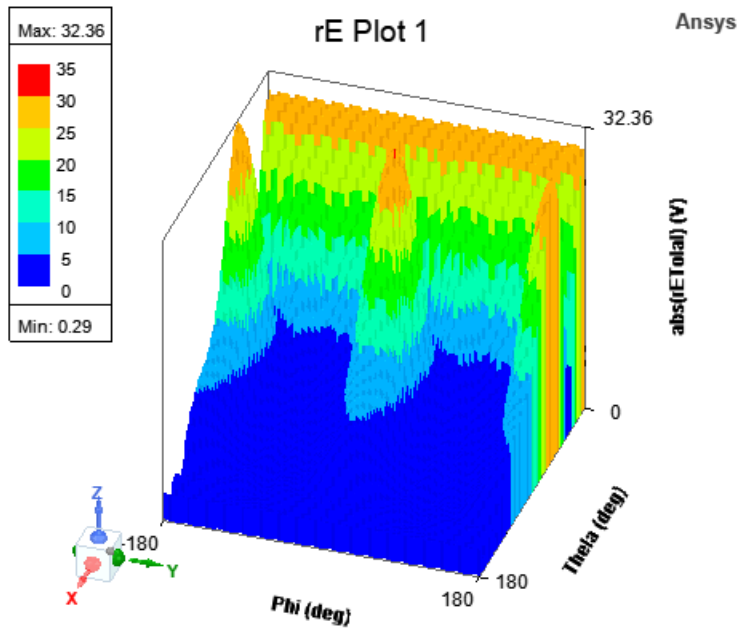
3D Rectangular Plot with Medium Visual Detail



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

On creation, a 3D Rectangular Plot has Visual Detail set to Medium and looks and feels as shown above. Specifically, under Medium Visual Detail level, a 3D Rectangular Plot has 3 ticks per axis (X, Y, Z axis) which will show min, max and middle value. This setting also shows axes labels.

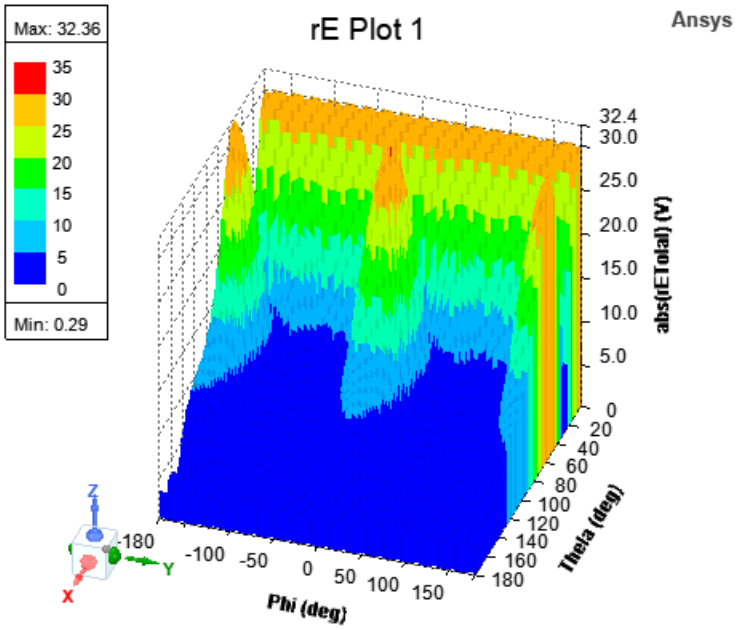
3D Rectangular Bar Plot with Low Visual Detail



With Visual Detail set to Low, a 3D Rectangular Plot shows axes with 2 ticks corresponding to min and max values. It also shows axes labels and grid borders.

3D Rectangular Bar Plot with High Visual Detail

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

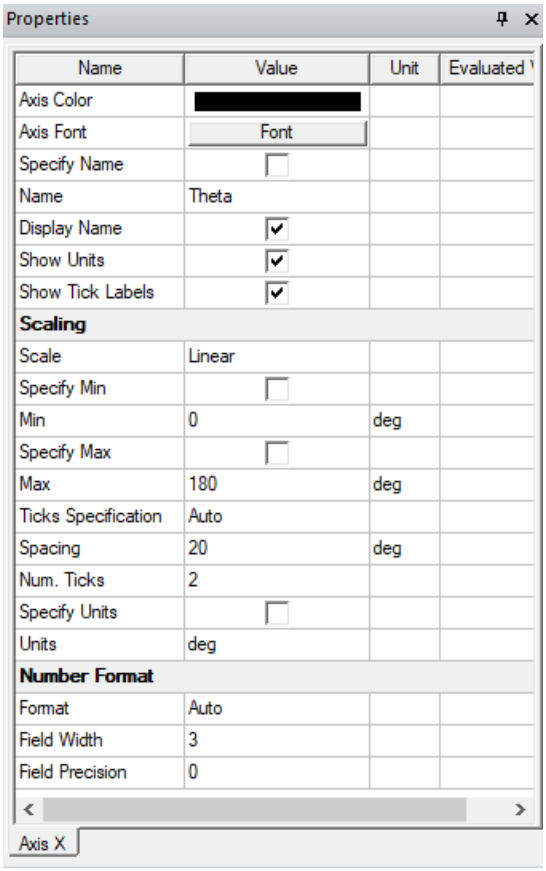


With Visual Detail set to High, a 3D Rectangular Plot shows all Cartesian axes and grids together with all ticks and axes labels.

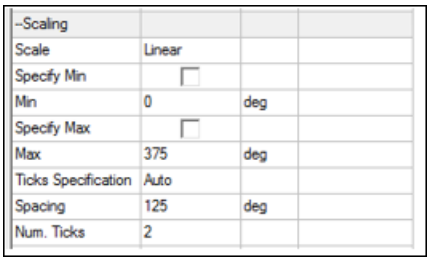
Axis Properties: Ticks Specification and Num. Ticks

Ticks Specification is available on Axis properties, as shown below:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ticks Specification is a menu with possible values as Auto, Spacing, and Num. Ticks, with Auto being the default value. If Ticks Specification is Auto, then a spacing value is automatically calculated and used to calculate and display the tick labels. **Spacing** shows the calculated value, and Num. Ticks shows the number of ticks based on this spacing value, as shown below:



You can edit the **Spacing** field when Ticks Specification is set to Spacing; otherwise, it is read only.

You can edit the **Num. Ticks** field when Ticks Specification is Num. Ticks; otherwise, it is read only.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Valid Num. Ticks are between 0 and 100, including 0 and 100. If you enter an invalid value, an error message is shown. If you enter a spacing value that results in number of ticks greater than 100, then an appropriate value is shown.

- If Num. Ticks is 0, then no ticks are shown on the axis.
- If Num. Ticks is 1, then only the max value tick is shown on the axis.
- If Num. Ticks is 2, then only the min and max value ticks are shown on the axis.
- If Num. Ticks is greater than 2, then evenly spaced ticks (including min and max) are shown on the axis.

Plotting a Quantity Versus Adaptive Pass

Any quantity that can be plotted in the reporter may be plotted versus the adaptive solution pass, primarily to evaluate the convergence of the quantity versus the mesh.

1. From the **Results** menu (or from the Maxwell menu or right-click on **Results** on the Project tree), click **Create <*type*> Report**, and select **Rectangular plot** from the report type menu.

   The **Report** dialog appears.

2. In the **Context** section, select the **AdaptivePass** context for the Setup of interest. The **Trace** tab **X** value drop down list will contain an option to plot against **Pass**.
3. Under the **Trace** tab, **Y** component section, specify the information to plot along the y-axis:
   a. In the **Category** list, click the type of information (for example, Expression Cache or Output Variable) to plot.
   b. In the **Quantity** list, click the value to plot.
   c. In the **Function** list, click the mathematical function of the quantity to plot.
   d. Value field displays the currently specified Quantity and Function. You can edit this field directly.

      > **Note:** Color indicates a valid expression.

   e. **Range Function** button – opens the **Set Range Function** dialog box. This applies currently specified Quantity and Function.
4. On the **Families** tab, confirm or modify the sweep variables that will be plotted.
5. Click **New Report.**

   This creates a new report in Project tree, displays the report with the defined trace, and enables the **Add Trace** button on the **Report** dialog.

   The function of the selected quantity will be plotted against the swept variable values or quantities you specified on an x-y graph. The plot is listed under **Results** in the project

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tree and the traces are listed under the plot. When you select the traces or plots, their properties are displayed in the Properties window. These properties can be edited directly to modify the plot.

6. Optionally, add another trace to the plot by following the procedure above, using **Add Trace rather than New Report**.

Creating Data Tables

A data table is a grid with rows and columns that displays, in numeric form, selected quantities against a swept variable or other quantities.

1. Click **Maxwell**> **Results**> **Create *<type>* Report** or right-click **Results** in the Project Manager and click **Create *<type>* Report**.
2. In the **display type** menu, click **Data Table**.

   The *Report* dialog box appears.

3. In the **Context** section make selections from the following field or fields, depending on the design and solution type.
   a. Solution field with a drop down selection list. This lists the available solutions, whether sweeps or adaptive passes.
   b. Domain field with a drop down selection list. Whether this field appears, and the domains listed depend on the Solution type and the report ***<type>*** selected. For modal and terminal solution data reports, the domain can be **Sweep** or **Time**.
   c. Geometry field with a drop down selection list. For field and radiated field reports, this applies the quantity to a geometry or radiated field setup.
4. Under the **Trace** tab **Y** component section, select the quantity you are interested in and its associated function:
   a. On the **Category** drop-down menu, click the type of information to plot. The category selected provides the default name for the plot.
   b. On the **Quantity** list, click the values to plot. Use Ctrl+click to make multiple selections.
   c. In the **Function** list, click the mathematical function to apply to the quantity for the plot.
   d. The **Value** field displays the currently specified Quantity and Function. You can edit this field directly.

      > **Note:**
      >
      > Color shows valid expression.

   e. **Range Function** button -- opens the *Set Range Function* dialog box. This applies currently specified Quantity and Function.

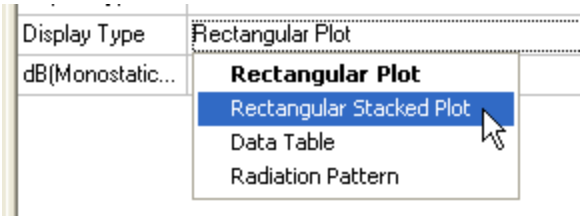
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. On the **Trace** tab **X (Primary sweep)** line, select the sweep variable from the drop-down menu and specify all values or select values.
6. Click **New Report**.

   This creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog box. The default name is based on the Report Category you selected, for example, S Parameter Plot *n* or Output Variables Plot *n*. You can edit the plot names in the project tree and the plot header text in the report synchronizes.

   The Y quantity will be listed at each variable value or additional quantity value you specified. The data table is listed under **Results** in the project tree. The plot is listed under **Results** in the project tree and the traces are listed under the plot. When you select the traces or plots, their properties are displayed in the Properties window. These properties can be edited directly to modify the plot.
7. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

You can also modify the display type of an existing plot from the Properties dialog for that plot. Select the Report icon in the Project tree to display the Properties dialog box. Selecting the Display Type field displays a menu with selections available for that plot.



Once you make a selection, the plot display updates for the current selection.

If you choose to print a data table:

- Selecting print "All" prints the whole table for current data page (if there is more than one data page).
- Selecting print "Pages" prints user-specified pages.
- If the table is bigger than the screen view (that is, it has scroll bar), printing first scrolls right, prints until no more scrolling and then scroll down.
- The Page number appears at the bottom of the page, aligned at center.
- The table layout of each page follows the screen, but with no scroll bar will be printed, and no data page bar as on screen.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating Rectangular Contour Plots

This is an x-y-z graph of results. Any data that you can currently plot in 3D (as 3D Cartesian or 3D polar) is a candidate for a contour plot.

1. On the **Results** menu (**Maxwell** menu or right-click **Results** in the Project Manager), click **Create *<type>* Report**, and select **Rectangular Contour plot** from the report type menu.

   The *Report* dialog box appears.

2. In the **Context** section make selections from the following field or fields, depending on the design and solution type:
   a. Solution field with a drop down selection list. This lists the available solutions, whether sweeps or adaptive passes.
   b. Geometry field with a drop-down selection list. For field and radiated field reports, this applies the quantity to a geometry or radiated field setup.
   c. Domain field with a drop-down selection list.
3. Under the **Trace** tab **Z** Component area, specify the information to plot as contours:
   a. In the **Category** list, click the type of information to plot. The selected Category provides the default plot name.
   b. In the **Quantity** list, click the value to plot.
   c. In the **Function** list, click the mathematical function of the quantity to plot.
   d. The **Value** field displays the currently specified Quantity and Function. You can edit this field directly.

      > **Note:**
      >
      > Color shows valid expression.

   e. **Range Function** button -- opens the *Set Range Function* dialog box. This applies currently specified Quantity and Function.
4. On the **Trace** tab **Y** (Secondary sweep) lines, specify the information to plot along the y-axis in one of the following ways:
   - Select the sweep variable to use from the drop-down list.
   - If sweeps are available, you can select the browse button to display a dialog that lets you select particular values. The quantity will be plotted against the primary sweep variable listed.
5. On the **Trace** tab **X** (Primary sweep) lines, specify the information to plot along the x-axis in one of the following ways:
   - Select the sweep variable to use from the drop-down list.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If sweeps are available, you can select the browse button to display a dialog that lets you select particular values. The quantity will be plotted against the primary sweep variable listed.

6. Click **New Report**.

   This creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog box. The default name is based on the Report Category you selected, for example, S Parameter Plot *n* or Output Variables Plot *n*). You can edit the plot names in the project tree and the plot header text in the report synchronizes.

   The function of the selected quantity or quantities will be plotted against the values you specified on an x-y-z graph. The plot is listed under **Results** in the Project Manager. When you select the traces or plots, their properties are displayed in the Properties window. These properties can be edited directly to modify the plot.

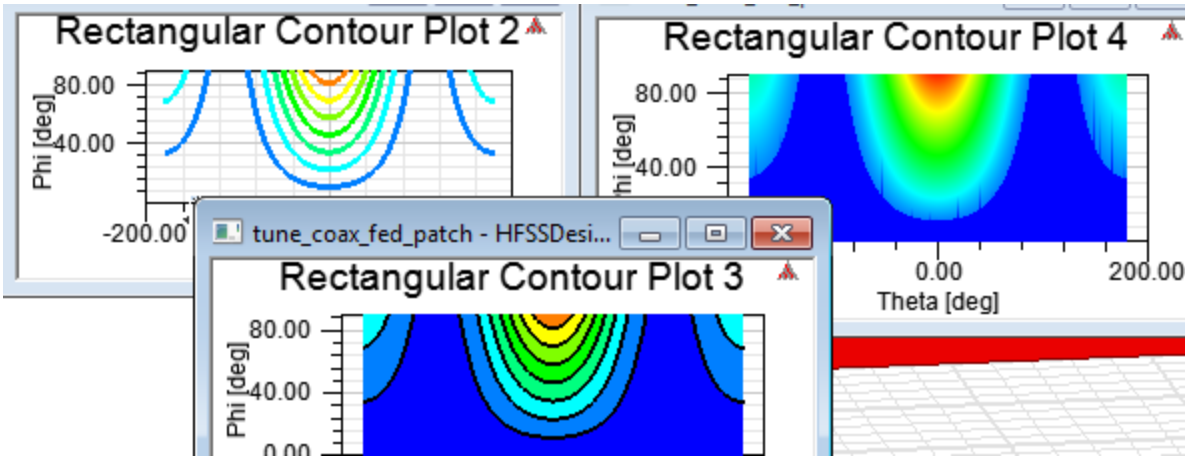
   The Trace Properties for Cartesian plots include tabs for:

   - Attributes tab, including Name, Line Style, Line Width, and Color.
   - Contour tab, including Colormap Type, Color, Spectrum, IsoValType, whether to Overlay Contour Lines (on Fringe or Tone plots), Scaling parameters, and Number of contours and spacing.

   Colormap Type can be Spectrum, Ramp, or Uniform.

   Spectrum can be Rainbow, Temperature, Magenta, or Gray

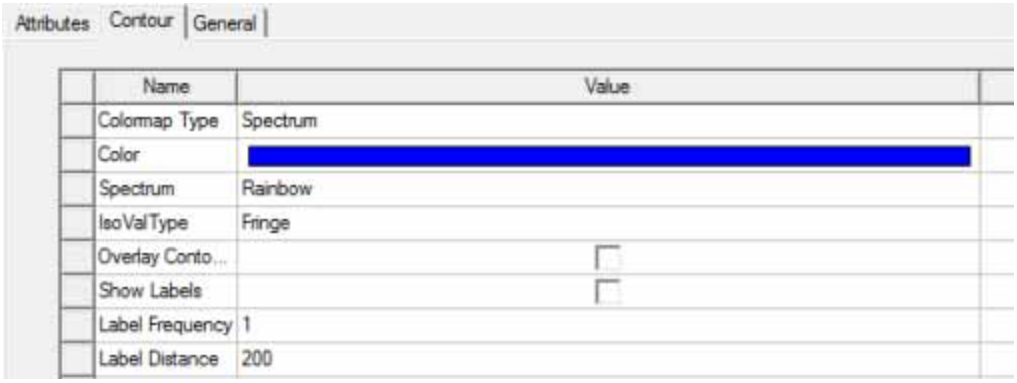
   IsoValType can be Line, Fringe, or Tone.



   General tab, including Back Color, Plot Area Color, Enable Y Axis Stripes, Field Width, Prevision, and whether to Use Scientific Notation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also access these properties by double-clicking on the Contour plot and viewing the **Contour** tab.



7. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

Delta Markers in 2D Reports

To view the difference between any two marker points in a report:

1. Set the first marker by clicking and holding the mouse button.
2. Move the mouse without releasing left button to another position, and then release the left button to create second marker.

In the marker text window, you see the difference between the two markers instead of the X, Y value of marker.

Working with Traces

A trace in a 2D or 3D report defines one or more curves on a graph. A trace in a data table defines part of the displayed matrix of text values.

The values used for a plot's axes (which may be X, Y, Z, phi, theta, or R depending on the display type) can be variables in the design, such as frequency, or functions and expressions based on the design's solutions. If you have solved one or more variables at several values, you can "sweep" over some or all of those values, resulting in a curve in 2D or 3D space.

A report can include any number of traces and, for rectangular graphs, up to 20 independent y-axes. Traces appear in the Project tree under their report. They can be selected, copied and pasted.

When you move a cursor over a trace in a report, the cursor changes to show that you can make a selection:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For PC systems, the cursor changes to the color of the selectable trace.
- For Linux systems, the cursor changes to a solid black arrow, rather than the default black outline.

In general, to add a trace to a report:

1. Select a report in the Project window and right-click and select **Modify Report**.
2. In the *Report* dialog box, specify the Y component information.
    a. Specify the Category of information you want to plot from the drop-down menu.

       The Category drop-down menu lists the available categories for the Solution type and the current design. Selecting a category changes the Quantity and Function lists to represent what is available for that category.

    b. Specify the Quantity you want to plot by selecting from the Quantity list.

       The selected quantity appears in the Value field, operated on any selected function.

    c. Select the Function to apply to the specified quantity.
    d. The Value field shows the trace being readied for plotting on the Y-axis. This field is editable when the text cursor is present. You can modify the information to be plotted by typing the name of the quantity or sweep variable to plot along an axis directly in the text boxes.

       > **Note:**
       >
       > Color shows valid expression.

    e. **Range Function** button -- opens the *Set Range Function* dialog box. This applies the currently specified Quantity and Function.
3. In the *Report* dialog box, specify the X axis information (for example Primary Sweep).
4. Click **Add Trace**.

   A trace is added to the traces list under its report icon in the Project tree. The trace represents the function of the quantity you selected and will be plotted against other quantities or swept variable values. Selecting a Trace in the Project tree displays the Properties window for that Trace. Selecting a trace in the report or legend displays the display Properties window for that trace.

   Trace icons can be selected, copied, and pasted for their definitions or their data. They can be selected and deleted from the Project Manager.

   By the default, the Trace name is the definition (the category, quantity and function). The trace will be visible in the report when you click **Add Trace**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Trace properties can be edited directly in the respective Properties windows or edited in the *Report* dialog box. To change the name or definition of a trace, see Editing Trace Properties. To edit other display properties of a trace, see Editing the Display Properties of Traces

Editing Trace Properties

To edit trace properties such as the name, Y Axis association, the component definition, the context, or the variables select the trace in the Project tree.

To edit a **trace name**:

1. Select the trace in the Project Manager.

   This displays a docked Properties window for the Trace.

2. Check the Specify Name box.

   This enables editing of either the Name field in the docked properties dialog box, or the Trace label text in the Project tree. Editing this name changes the display in the Legend and in the Project tree, but not the underlying Y-component definition.

> **Note:**
>
> To control the display of the Solution Name and Variation Key in the Legend, see Report 2D: Legend Tab.

To edit the Y Axis associated with the trace (2D Rectangular and Rectangular Contour plots):

1. Select the trace in the Project Manager.
2. In the docked properties window for the trace, select the Y Axis to be associated with the trace from the drop-down menu. Up to twenty independent Y axes can be added to a plot.

To edit a **trace component definition**:

1. Select the trace in the Project Manager.
2. In the docked Properties window for the trace, select the component field of interest, and select **Edit...** from the drop-down menu.

   This displays the edit Component field window from which you can edit the category, quantity and function.

3. Click **OK** to apply the changes and close the dialog box.

To edit a **trace Context**:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the trace in the Project Manager to display the docked properties window.
2. In properties window, click the **Solution** field or the **Domain** field. If other selections are possible, they can be selected from the drop-down menu.

To edit a **variable** for a trace:

1. Select the trace in the Project Manager to display the docked properties window.
2. Under the **Variables** category, on the Families line, click the **Edit** button to display the Edit families dialog box.

   From this dialog box, you can select the **Sweeps** or **Variations** radio buttons. If other nominal values are available you can click the ellipsis button to select from a list.

Editing the Display Properties of Traces

Editing the display properties of traces differs for 2D and 3D reports. To edit the display properties of a trace for a 2D report:

1. Select a trace in an open **Report** window.
2. Click once on the trace to view settings in the docked *Properties* window, or double-click to open *Properties* dialog box.

   The display properties window for a 2D trace includes a **General** tab and an **Attributes** tab.

   The *General* tab properties apply to the general appearance of the plot. They include the Background color, Contrast color, Field width, and Whether to use Scientific notation for marker and delta marker displays. (X and Y notation display is set separately, in the Axis property tabs.)

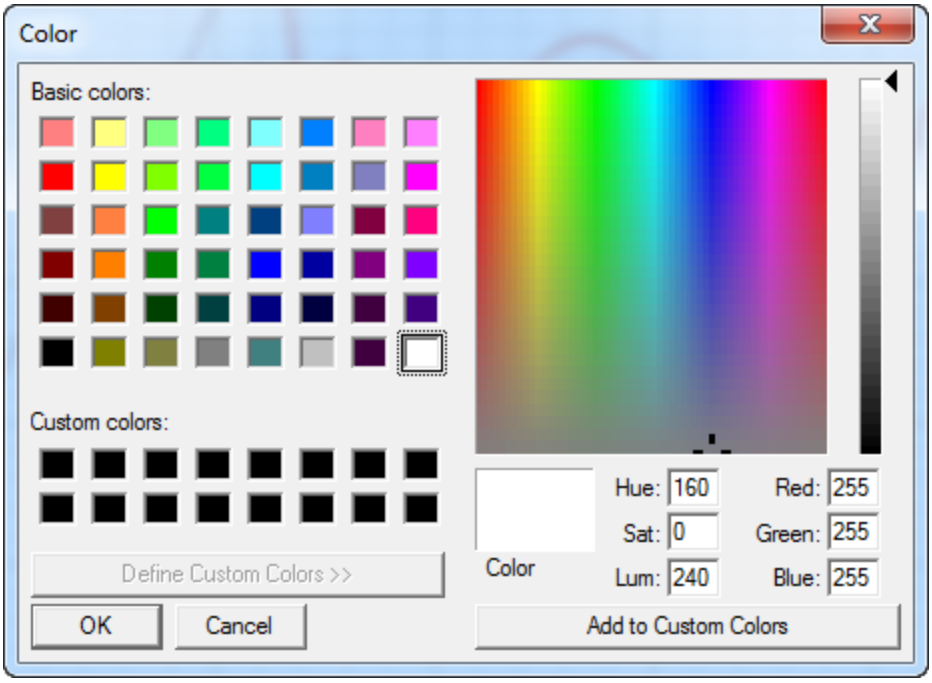
   The *Attributes* tab properties apply specifically to the trace. The defaults are set in the Report2D options. They include:

   - Name -- not editable by selecting the trace from the Report. It shows the characteristics of the trace as defined in the *Report* dialog box.

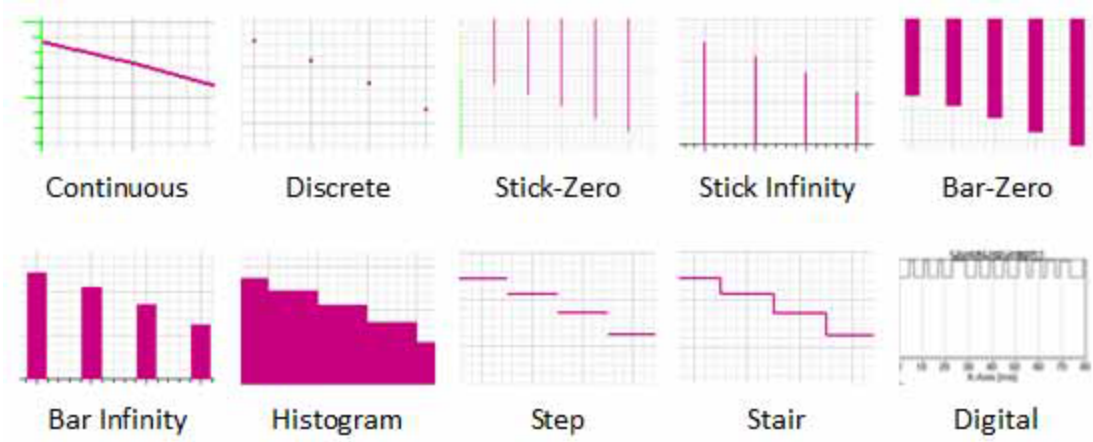
     To edit a trace name, see Editing Trace Properties.

   - Color -- shows the Trace color. Double-click to open a Color dialog box. You can select from Basic colors, or custom colors. You can define up to 16 custom colors by selecting or by editing the values for Hue, Saturation, Luminescence, and the Red, Green, and Blue.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



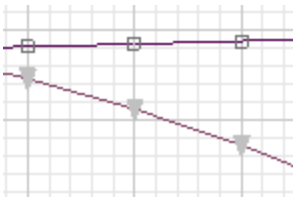
- Line style -- a drop-down menu lets you select Solid, Dot, Dash, or Dot-dash.
- Line width -- a text field lets you edit the numeric value.
- Trace type -- the drop-down menu contains entries for Continuous, Discrete, Bar-Zero, Bar Infinity, Stick Zero, Stick Infinity, Histogram, Step, and Stair.



Notice the difference between Stair and Digital is that each Stair centers on a data point with transitions halfway between points, and Digital transitions from each data point to the next value.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The next four properties work together to define whether to show a symbol on data points, the symbol frequency, the symbol style, and whether to display the symbol as solid or hollow.



- Show Symbol -- whether to show a symbol at the data points on the line.
- Symbol Frequency -- how often to show symbols on the trace, based on the number of data points per symbol used. For example, specify 1 for one symbol per data point. Specify 10 for one symbol for every 10 data points.
- Symbol Style -- use a drop-down menu to select from box, circle, vertical ellipse, horizontal ellipse, vertical up triangle, vertical down triangle, horizontal left triangle, horizontal right triangle.
- Fill Symbol -- use the check box to set the symbol display as a solid or as hollow.
- Symbol Arrows -- use the check box to use arrows on the curve ends.

> **Note:**
>
> So that curves with single points always appear, Box is the default symbol.

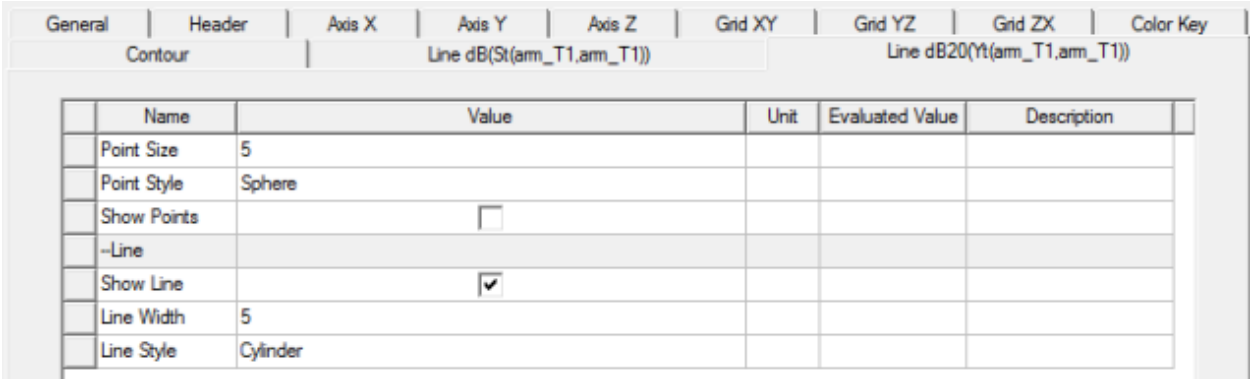
3. Edit the properties, if needed. Click OK to apply the changes and close the window.

To edit the display properties of a trace in a 3D report:

1. Click on the trace. This opens a Properties dialog for the plot with a tab named for the trace selected.
2. The editable properties include Point size, Point Style, whether to show points, whether to

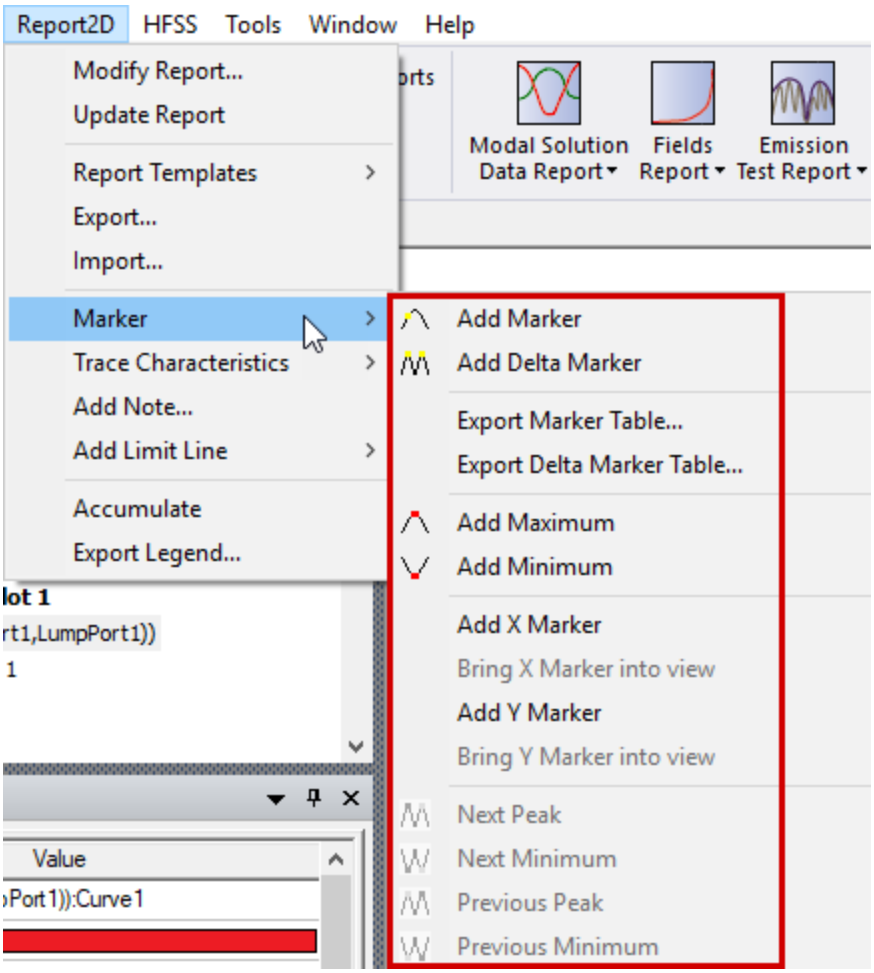
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

show line, line width, and line style.



Adding Data Markers to Traces

The Reporter includes **Report 2D** > **Marker** > menu commands and icons:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also access these commands from the shortcut menu that appears when you right-click inside a report plot window.

These command let you add markers to traces. A marker appears as "m*N*" at the marked point, where *N* increments from 1 as you place additional markers. Each marker can be selected and has editable properties including name, font, background and color. As you place markers, one or more marker legends may be displayed, depending on the **View** > **Active View Visibility** settings for the legends. The main marker legend appears in the upper left of the plot, and lists the marker names and their X and Y values in a table. You can control the number format for the table values via the properties window, general tab. Under *Marker/Other Number Format*, you can specify field width, precision, and whether to use scientific notation. This value is independent of the Axis tab number properties. A separate marker legend appears for Delta Markers, as described for the **Delta Marker** command.

When you enter Marker mode, the cursor arrow is accompanied by an "m" while a circle on the selected trace shows the current position for a potential marker.

To end Marker mode, right-click to display the shortcut menu, and select **End Marker Mode**.

The available Marker mode commands and associated icons are the following:

- **Marker** – this command lets you place a marker at an arbitrary point on a selected trace.
- **X Marker** – this command adds up to 10 movable markers at the origin of the plot with a vertical line rising from the X axis. Each added marker has its own color and editable properties. To move an X marker, click on the X label and drag it to the desired location. The label at the bottom of the line gives the X coordinate, and flag on the vertical line identifies the Y coordinate on the trace. A trace property lets you lock the drag feature to leave the marker in place. The X markers are cleared by the **Clear All** command.
- **Bring X Marker into view** – this command is enabled if an X Marker is not visible in the plot. It allows you to select from a list of existing X Markers to bring into view.
- **Y Marker** – this commands adds up to 10 Y Markers with a horizontal line extending from the Y axis. For more detail on Y Markers and their use, see Y Markers in stacked XY plots.
- **Bring Y Marker into view** – this command is enabled if a Y Marker is not visible in the plot. It allows you to select from a list of existing Y Markers to bring into view.
- **Maximum** – places a marker at the Maximum value on the selected trace.
- **Minimum** – places a marker at the Minimum value on the selected trace.
- **Delta Marker** – enters delta marker mode, placing a circle on the selected trace. Clicking on the trace sets an initial point and subsequent clicks on arbitrary points on the trace place additional markers until you leave marker mode. These markers have their own legend, which includes the following information for each pair of markers specified:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Delta(X)	Delta(Y)	Slope(Y)	InvSlope(Y)
d(m2,m3)	0.4700	1.8319	3.8976	0.2566

- **Next Peak** – moves a selected marker on the next peak on a trace. You must exit marker mode and select a marker to enable this command.
- **Next Minimum** – moves a selected marker to the next minimum on a selected trace. You must exit marker mode and select a marker to enable this command.
- **Previous Peak** – moves a selected marker on the previous peak on a selected trace. You must exit marker mode and select a marker to enable this command.
- **Previous Minimum** – places a marker on the previous minimum on a selected trace. You must exit marker mode and select a marker to enable this command.
- **Go to Start** (Right arrow) – moves a selected trace marker to the first data point. Enabled by leaving marker mode and selecting a marker.
- **Go to Previous** (Left arrow) – moves a selected trace marker to the previous data point.
- **Go to Next** – moves a selected trace marker to the next data point.
- **Go to End** – moves a selected trace marker to the last data point.
- **Next Curve** – selects the next curve in the report, based on the order in the trace legend.
- **Previous Curve** – selects the previous curve in the report, based on the order in the trace legend.
- **Clear All** – clears all markers on a report.

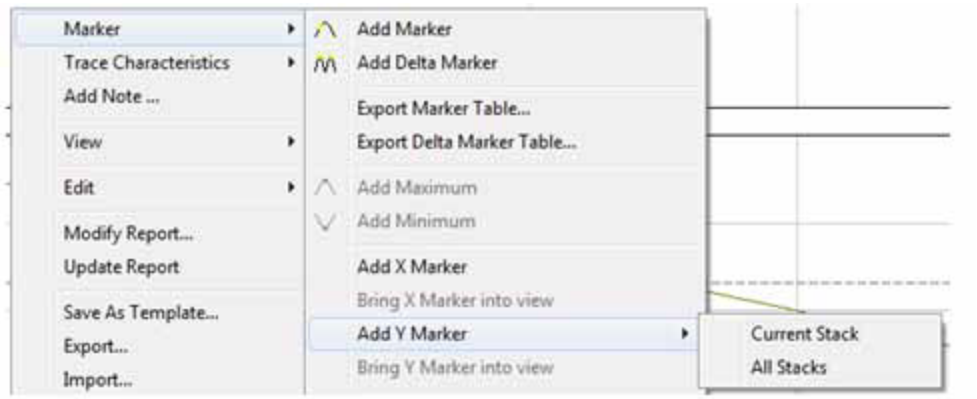
Y Markers in Stacked XY Plots

Y Markers allows for easy analysis and comparison of curves at a particular y-coordinate. Y Markers can be used to compare stacked curves.

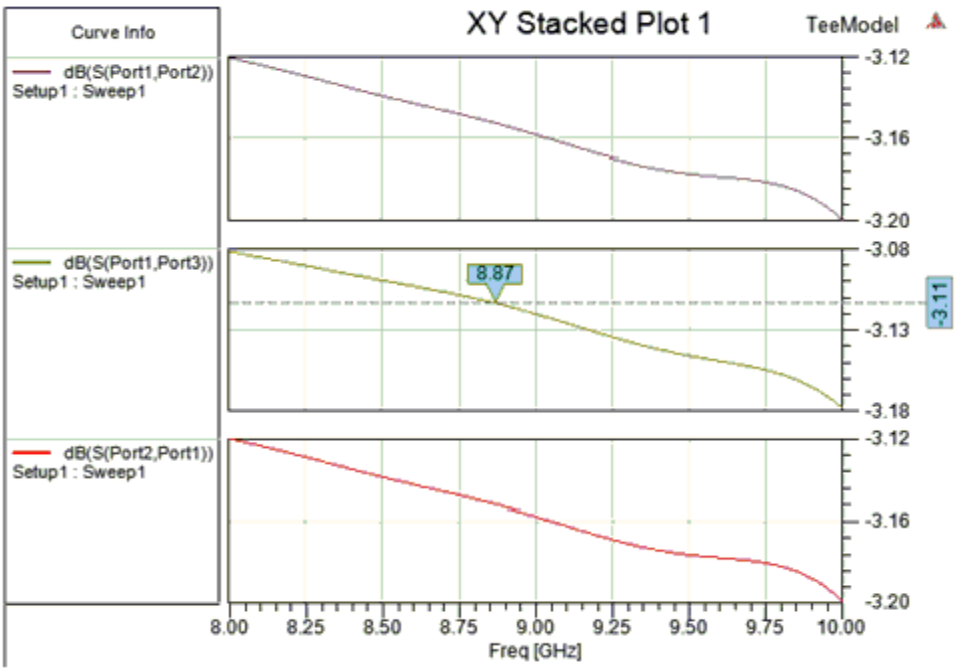
Creating Y Markers in Stacked Plots

There are two ways to create Y Markers in Stacked Plots. You can create a Y Marker for a particular stack or for all stacks. Right-clicking on any stack shows the following shortcut menu:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

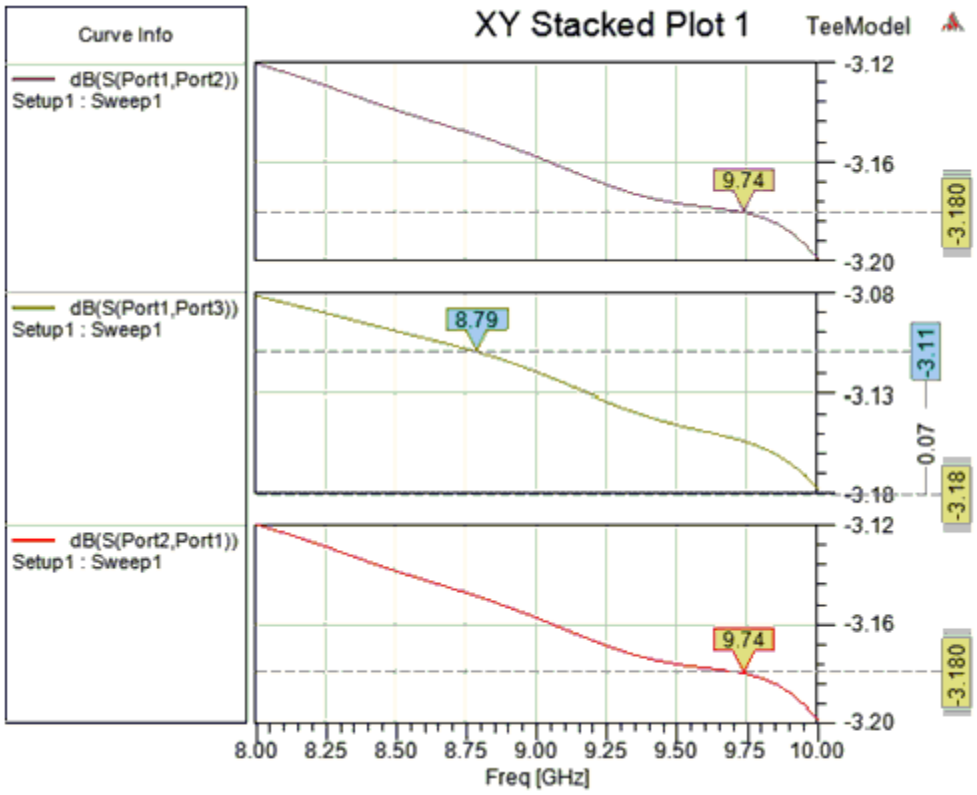


Add Y Marker > **Current Stack** creates a Y marker for the stack on which user performed right mouse button click. The following figure shows that a Y Marker was added to second stack only:



Add Y Marker > **All Stacks** creates one Y marker in each stack with same value. Initially this value is the minimum Y value of the Y ranges in all the stacks. This is shown is figure below:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

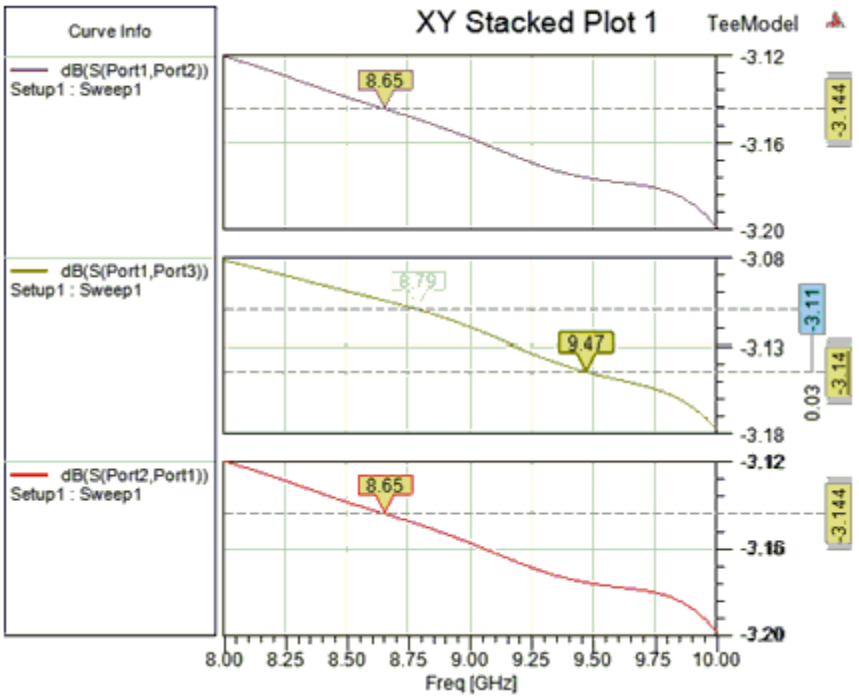


Notice that the Y Marker for All Stacks has a different appearance than the Y Marker for a particular stack, that is, it has double parallel lines above and below the Y Marker textbox.

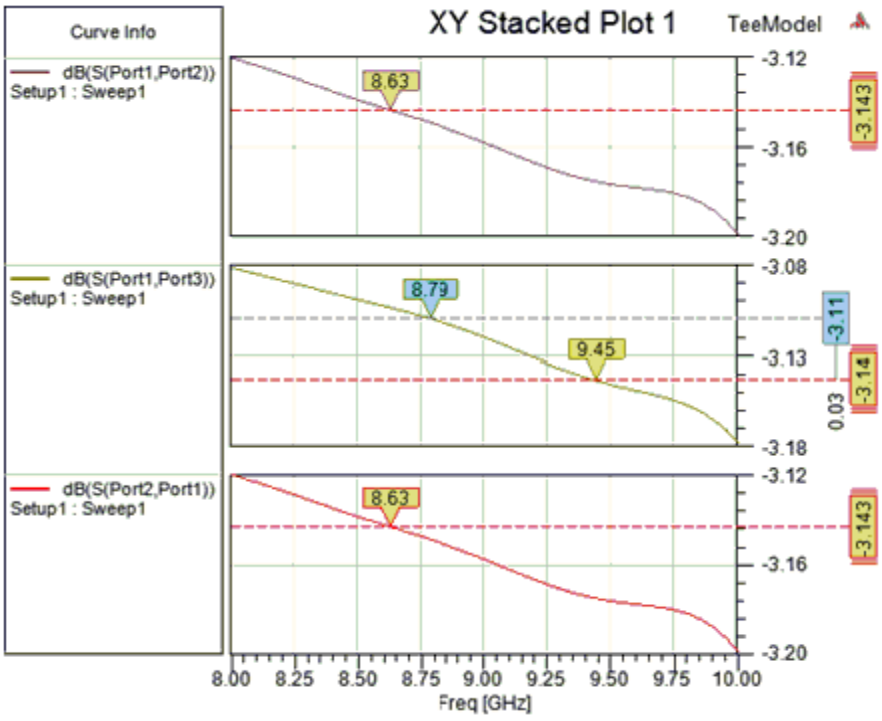
Synchronized Y Markers

All the "same" Y markers for all stacks are synchronized, that is to say that if one Y marker is dragged or it's value is changed, all the "same" Y markers in all the stacks will change their position too. The following figure shows that when Y marker in bottom stack was dragged, Y marker in top stacks moved as well:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



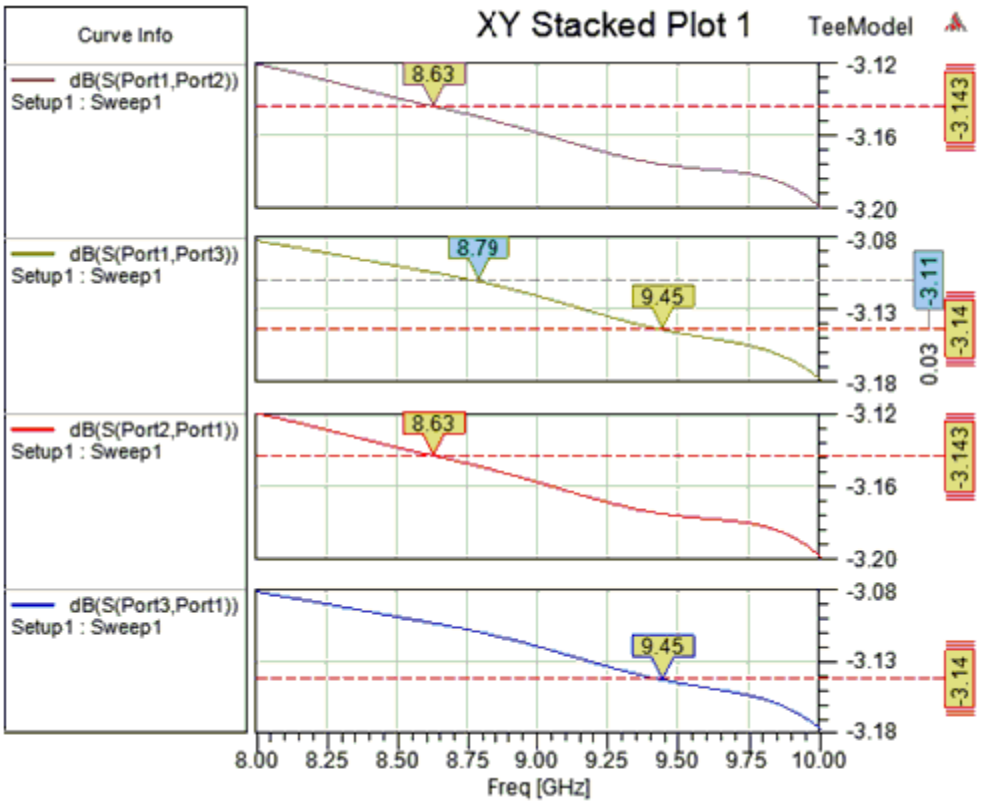
Also if a property of any one Y Marker is changed, all the "same" Y Markers show the change in property as well. For example the following figure shows that when the line color of a Y Marker in the top stack was changed to red color, a Y Marker in bottom stack show the same line color as well:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Automatic Y Markers for the New Stack

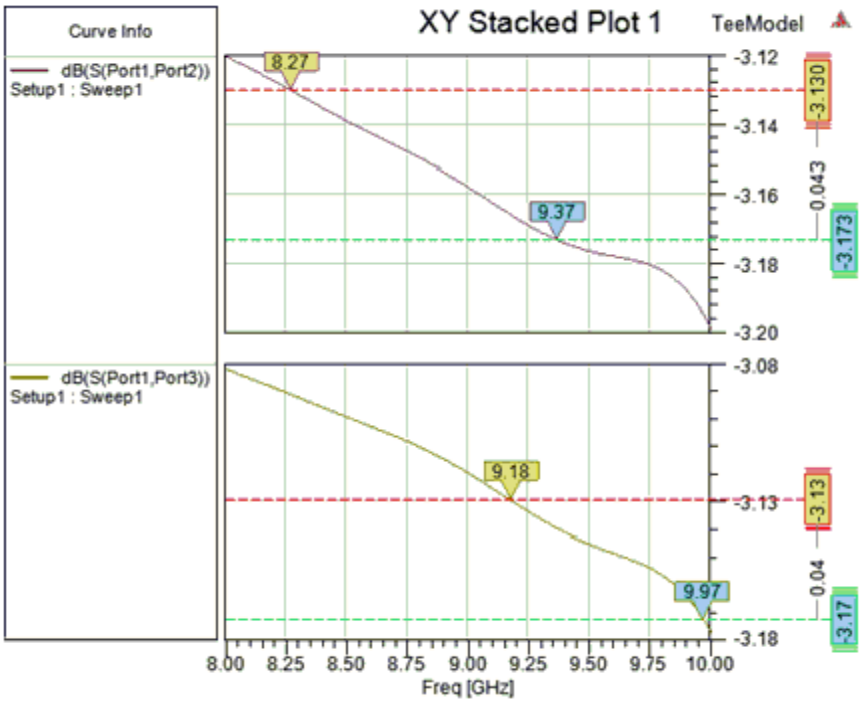
When a new curve is added to the plot, it gets all the Y Markers for all stacks in other stacks, excluding the Y Marker for particular stacks. The following figure shows that when the new curve "dB(S(Port3, Port1)" was added, a Y Marker was added to it with value -3.14 and it has all the same properties as other "same" Y Markers in other stacks:



Y Marker Delta Annotations

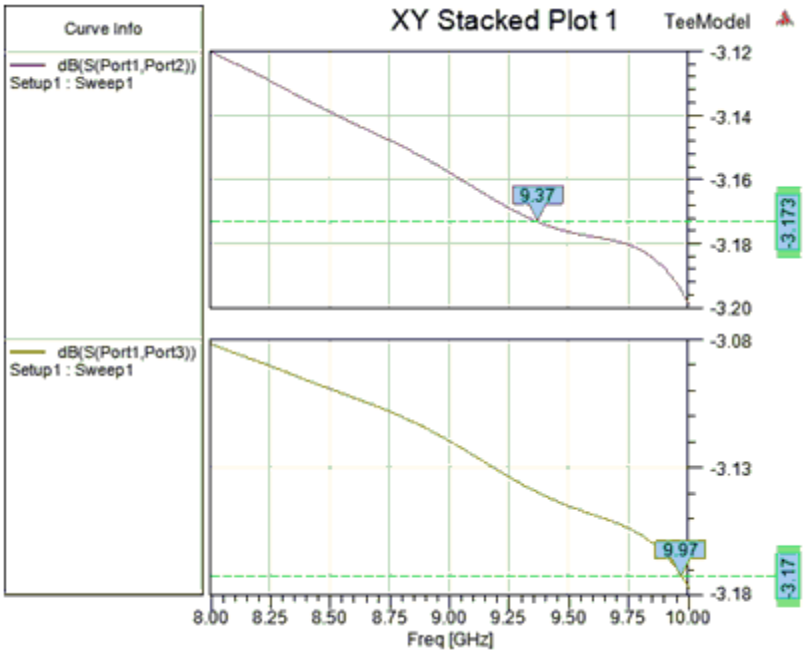
When two more Y Markers are present in a Stacked Eye Diagram, then delta annotations are shown between a pair of adjacent Y Markers in all the stacks, as shown in figure below:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Deleting a Y Marker

To delete a Y Marker, select a Y Marker in any stack and press the Delete key. This action will also delete all the corresponding Y Markers in all the stacks. For example, when the Y Marker with value -3.13 (red Y Marker) was deleted from the bottom most stack, all of the corresponding Y Markers were also deleted:

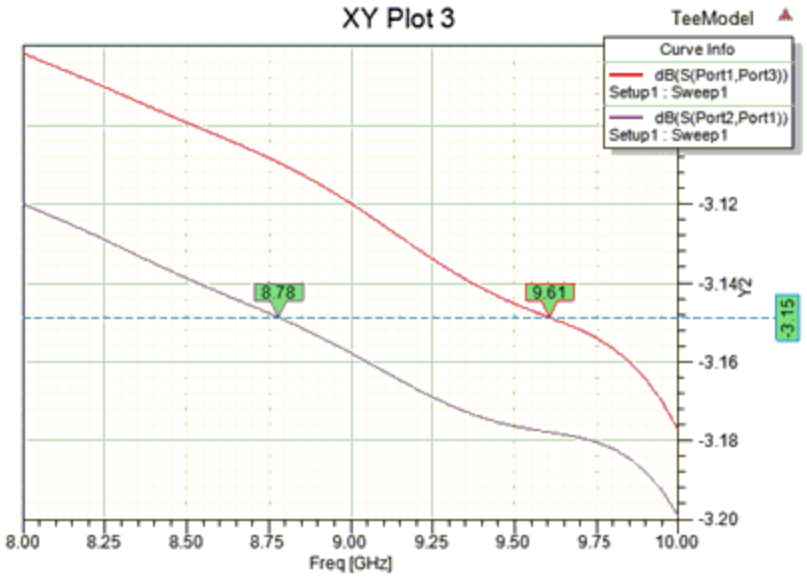


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

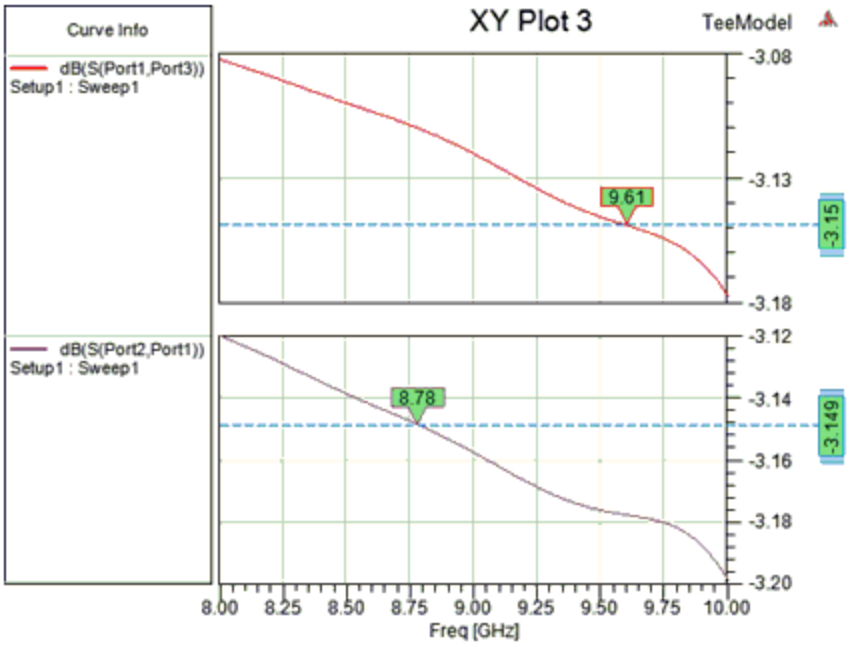
Note that on deleting a stack, Y Markers in other stacks are not affected.

Converting Rectangular XY Plot to Rectangular Stacked XY Plot

The following figure shows a Rectangular XY Plot with two curves and a Y Marker with value -3.15 (blue Y Marker):



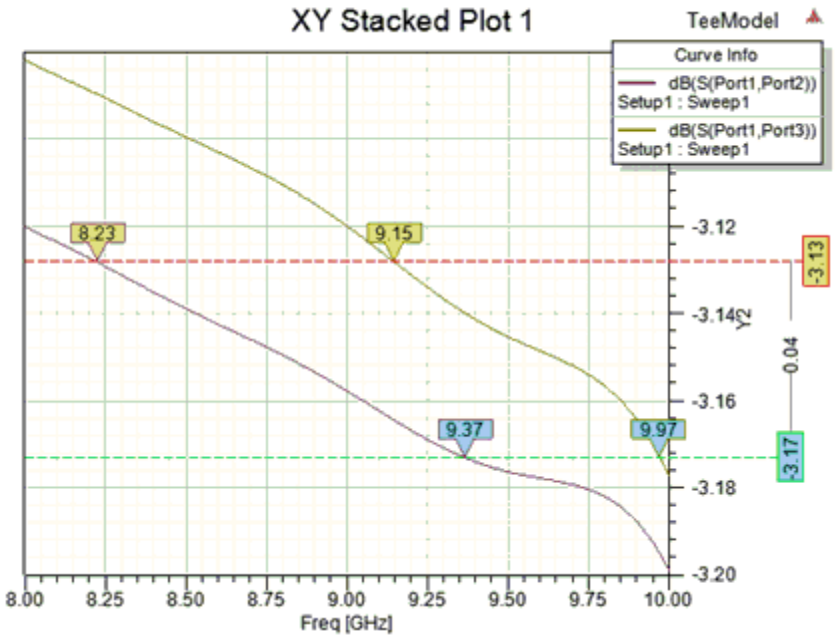
If you change the Display Type property of this plot to Rectangular Stacked Plot then a Rectangular Stacked XY Plot is created with each curve in its own stack and a Y Marker is shown in each stack with value -3.15 (blue Y Marker):



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Similarly when you change a Rectangular Stacked XY Plot to a Rectangular XY Plot then all the "same" Y Markers in all the stacks are shown as a single Y Marker in Rectangular XY Plot as shown in following figures:

The Rectangular Stacked XY Plot in the previous figure, when converted to Rectangular XY Plot, looks like the following figure:



Discarding Report Values Below a Specified Threshold

To prevent real small numbers from skewing a plot, you can discard small values (below a specifiable threshold).

1. Double-click the X or Y axis of interest on an open plot display.

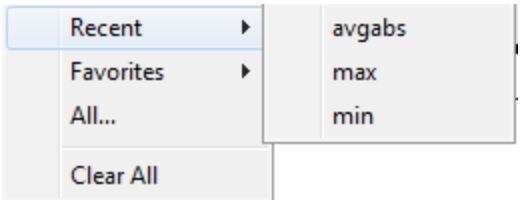
   This opens the **Properties** window for the Axis.

2. Under the **Axis** tab, use the scroll bar to find the **Specify Discard Values** property.
3. Click the check box to enable the property.
4. Enter a value in the **Discard Below** field. Units specified elsewhere in the Axis property are applied to this value. The Discard Below text box is inactive if the Specify Discard Values check box is not enabled.
5. Click **OK** to apply the Discard Values to the report.

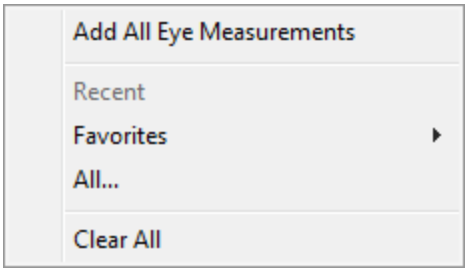
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adding Characteristics to a Trace

There are several options for adding characteristics to a trace. When you click **Report2D** > **Trace Characteristics**, or right-click a selected trace, the shortcut menu is displayed. The following example shows the menu with expanded **Recent** selections.



The following shows the shortcut menu for Eye Measurements, which includes the **Add All Eye Measurements** option. You can use this option to add all eye measurements at once.



Adding a Recently Used Trace Characteristic

If you recently used a characteristic, you can add it to a selected trace by selecting from a list of recently used characteristics. A maximum of 10 is displayed in the menu, and they are sorted alphabetically.

To add a recently used characteristic to a selected trace:

1. Select a trace in a report plot or legend.
2. Click **Report2D** > **Trace Characteristics**, or right-click the selected trace to display the shortcut menu.
3. Select **Recent**, and then select the function you want. The specified characteristic is added to the trace.

Adding a Trace Characteristic from Favorites

You can add a trace characteristic to a selected trace by selecting from a list of favorites. A maximum of 10 is displayed in the menu, and they are sorted alphabetically.

To add a favorite characteristic to a selected trace:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select a trace in a report plot or legend.
2. Click **Report2D** > **Trace Characteristics**, or right-click the selected trace to display the shortcut menu.
3. Select **Favorites** and then select the function you want. The specified characteristic is added to the trace.

Adding Trace Characteristics to your Favorites

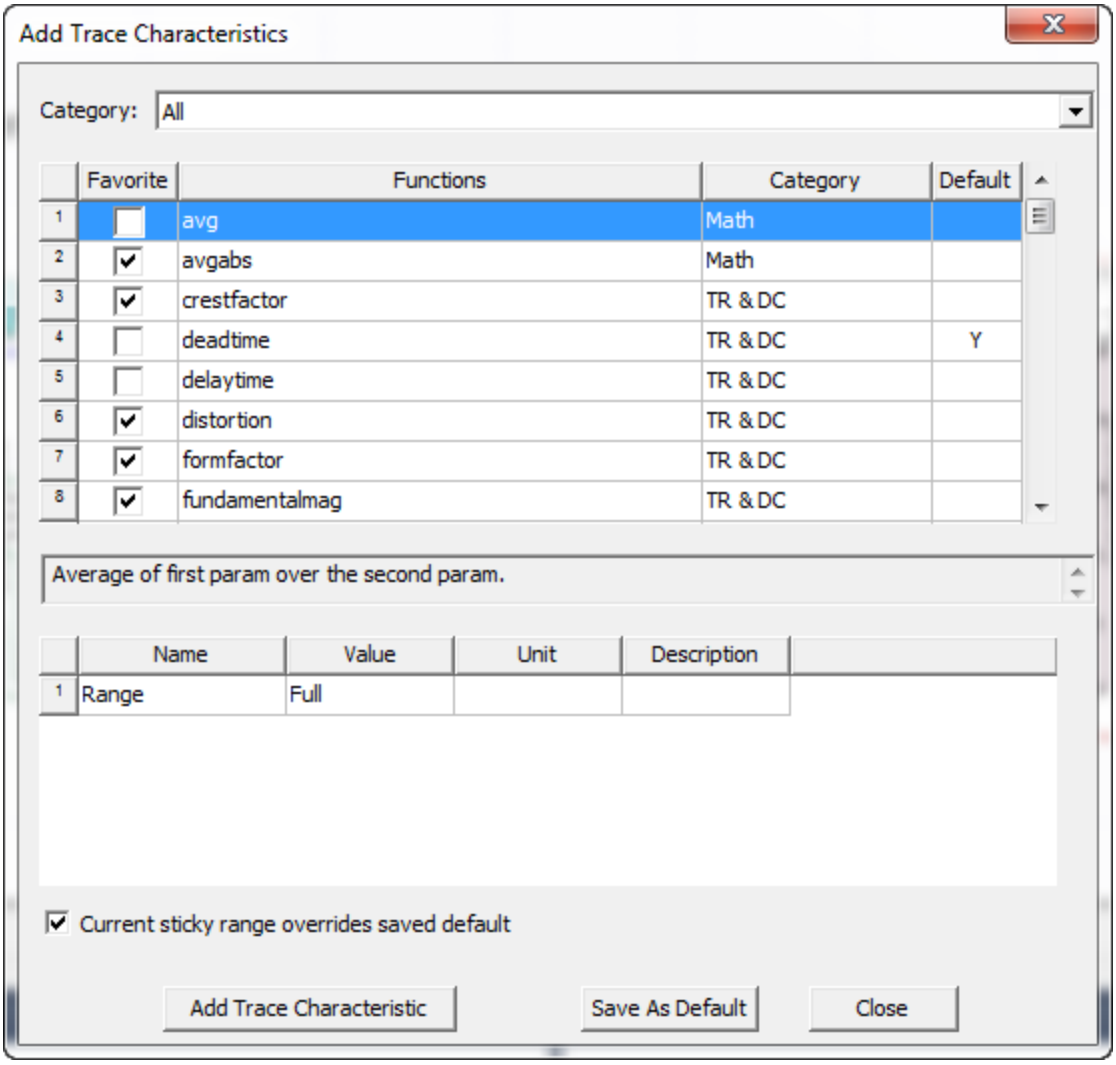
You can add trace characteristics to your list of favorites.

To add characteristics to your list of favorites:

1. Select a trace in a report plot or legend.
2. Click **Report2D** > **Trace Characteristics** or right-click the selected trace to display the shortcut menu.
3. Select **All**.

   The *Add Trace Characteristics* dialog box displays.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Click the **Favorite** check box in front of any function you want to add to your Favorites. You can define as many favorites as you need, but no more than ten are displayed in the menu, and they are displayed in alphabetical order.
5. Click **Close**. You can view the current favorites by selecting Favorites in the Category drop-down menu.

> **Note:**
>
> You can remove favorites by clearing the **Favorite** check box for one or more functions, and clicking **Close**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

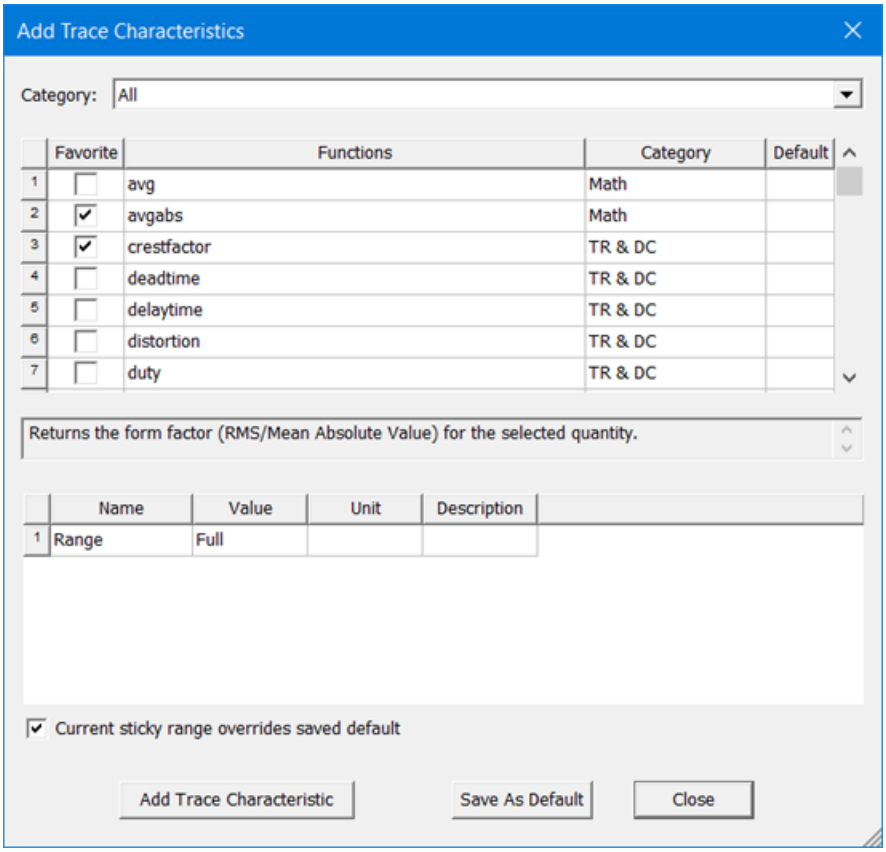
Adding Characteristics Using the Add Trace Characteristics Window

You can add characteristics to a selected trace by selecting from the **Add Trace Characteristics** window.

To add additional characteristics to a selected trace:

1. Select a trace in a report plot or legend.
2. Click **Report2D** > **Trace Characteristics**, or right-click the selected trace to display the short cut menu.
3. Select **All**.

   The **Add Trace Characteristics** window displays:



4. Select the desired **Category**.

   The available categories depend on the plot, and the selecting of a category displays its associated functions.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note: Categories** and **Functions** only appear if they are available for that plot.

Category	Functions
Recent	Displays the most recent functions used, sorted by the time they were added.
Favorites	Displays all favorites. The defaults are avg, max, min, and pk2pk.
All	Displays all available functions.
Math	avg, avgabs, integ, integabs, max, mean, min, pk2pk, pkavg, ripple, rms, rmsAC, stddev, sum, variance, XatYMax, XatYMin, XatYVal, XWidthAtYVal, YatXMax, YatXMin, YatXVal
PulseWidth	pulsefall9010, pulsefront1090, pulsefront3090, pulsemax, pulsemaxtime, pulsemin, pulsemintime, pulsetail50, pulsewidth5050, pw_minus, pw_minus_avg, pw_minus_max, pw_minus_min, pw_minus_rms, pw_plus, pw_plus_avg, pw_plus_max, pw_plus_min, pw_plus_rms
Overshoot/ Undershoot	overshoot, overshootAbs, undershoot, undershootAbs
TR & DC	crestfactor, deadtime, delaytime, distortion, formfactor, fundamentalmag, risetime, settlingtime
Error	iae, ise, itae, itse
Period	per, pmax, pmin, prms
AC	gainmargin, phasemargin, gaincrossover, phasecrossover, lowercutoff, uppercutoff, bandwidth, peakgain, peakgainfreq
Radiation	lSidelobeY, rSidelobeY, lSidelobeX, rSidelobeX, xdb10Beamwidth, xdb20Beamwidth
Eye Measurements	EyeLevelZero, EyeLevelOne, EyeAmplitude, EyeHeight, EyeSignalToNoise, EyeOpeningFactor, EyeWidth, EyeJitterP2P, EyeJitterRMS, EyeRiseTime, EyeFallTime, MinEyeWidth, MinEyeHeight
TDR	Shunt_C_in_pF, Series_L_in_nH

For a selected function, the *Add Trace Characteristics* dialog box displays the function's purpose in a text field.

5. Some categories and functions call for you to specify one or two additional values in a table. You can save these values using the **Save as Default** button. The Default column shows a Y if there is a saved default value for the function.
6. Select the **Current sticky range overrides saved default** check box if you do not want the range value in the table to be changed when the function selection is changed: the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

current range value becomes the “sticky range.” If the check box is not checked, the range value is updated from the saved default values and becomes a new sticky range.

7. Click the **AddTrace Characteristic** button to add the specified characteristics to the trace.
8. Click **Close**.

Removing All Trace Characteristics

1. Select a trace in a report plot or legend.
2. Click **Report2D** > **Trace Characteristics** or right-click the selected trace to display the shortcut menu.
3. Select **Trace Characteristics** > **Clear All**.

Trace characteristics are cleared from the selected trace.

Removing Traces

You can remove traces from the traces list in the following ways:

To *remove one trace* from the report:

- Select the trace you want to remove from the Project tree, and then click **Delete**.

To *remove all traces* from the report:

- Select all the traces and click **Delete**.

Copy and Paste of Report and Trace Definitions

You can copy and paste report and individual trace definitions within a single design or across designs. The report or trace definition will be evaluated within the context of the target design or report.

> **Note:**
>
> - If the report or trace definition contains properties that do not exist in the target design (for example, a port name) an error will be posted that indicates a solution does not exist for this trace
> - You must copy and paste trace definitions between the same report types. For example, you cannot copy a trace from a Modal Solution Data report and paste it in a Far Fields report.

To copy a Report Definition:

Right-click the report name in the Project Manager and select **Copy Definition** from the shortcut menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To paste the Report Definition:

Right-click Results in the Project Manager of the target design and select **Paste**.

A new report is created and it contains the copied definitions.

To copy an individual Trace Definition(s):

Right-click the trace or traces under a report name in the project tree and select **Copy Definition**.

To paste the Trace Definition(s):

Right-click the report in the target design to which you would like to copy the trace or traces and select **Paste**.

A new trace(s) is added to the report and it contains the copied trace definition(s).

> **Note:**
>
> If you copy and paste a report or trace definition to a design that contains a definition with the same name, then an incremented number is appended to the pasted report or trace name.

Copy and Paste of Report and Trace Data

You can copy and paste report and individual trace data within a single design or across designs. The report and trace definitions and all underlying data within the report or trace are copied and pasted to the target design or report.

To copy all data from a report:

Right-click the report name in the project tree and select **Copy Data**, or use the menu bar **Edit** > **Copy Data**, or right-click within a plot to display a shortcut menu with **Copy Data**.

To paste copied report data:

Right-click Results in the project tree of the target design and select **Paste**.

To copy data from an individual trace(s) in a report:

Right-click the trace or traces under a report name in the project tree and select **Copy Data**.

To paste copied trace data:

Right-click the report in the target design to which you would like to copy the trace data and select **Paste**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

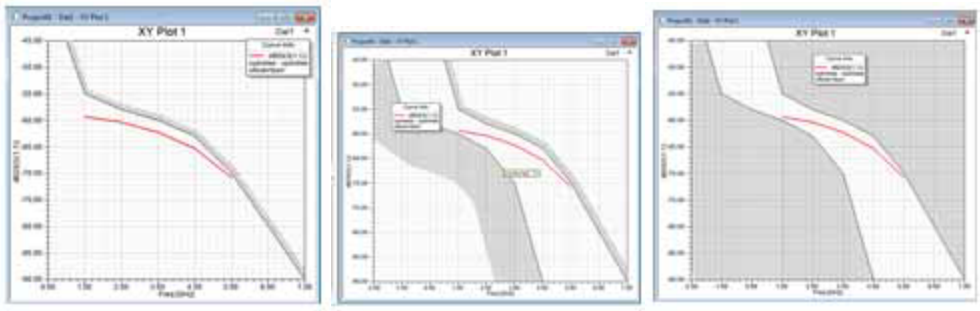
Note:

If you copy and paste report or trace data which contains the same name definition as a report or trace in the target design then an incremented number will be appended to the pasted name.

Limit Lines in Cartesian Plots

Limit lines are simple graphical representation of constraints on XY plots. These are modeled as a sequence of XY point pairs, or as offsets from a selected curve. You can designate a single limit line to delineate an upper limit, or two lines to delineate upper and lower limits, or upper and lower offset lines simultaneously.

You can control the display properties of the line including color and hatch width in pixels. These lines are available only on XY plots (and not on the XY-like plots: bode, stacked etc)

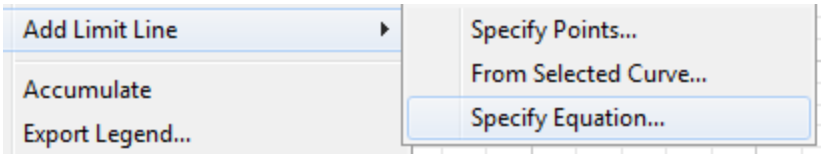


Note:

Limit lines are available only on Rectangular (XY) plots, not on XY-like plots such as Bode or Rectangular Stacked. On 2D Plots, the axes extents are based on the extents of the curves.

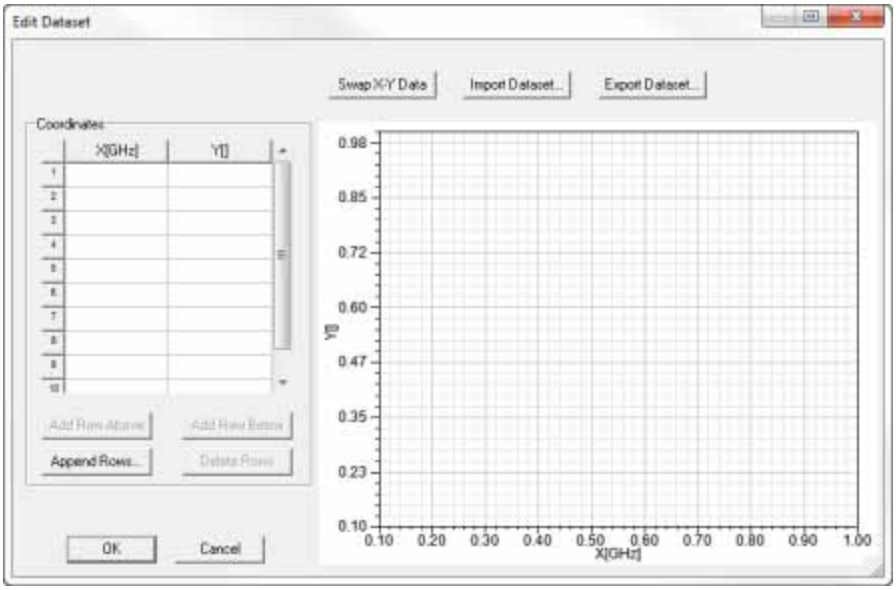
To create a limit line:

1. Click **Report2D** > **Add Limit Line** or right-click an XY plot and select **Add Limit Line...** from the Context menu. You then select whether to **Specify Points**, or **From a Selected Curve**, or **Specify Equation**.

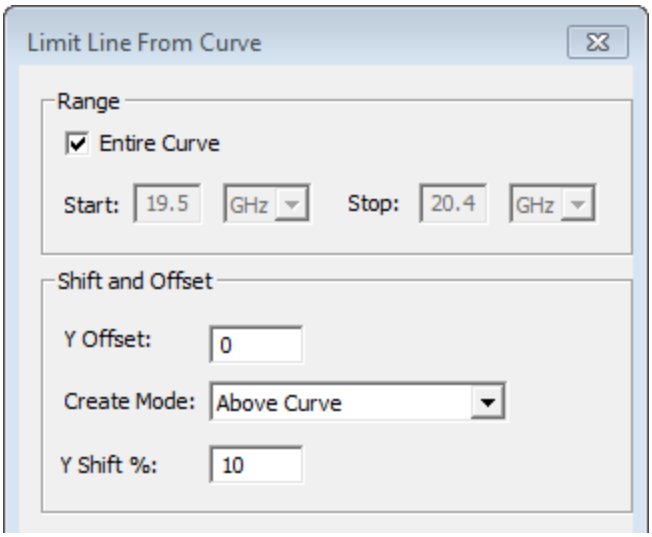


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select **Specify Points** to open an *Edit Dataset* dialog box so that you can specify points.

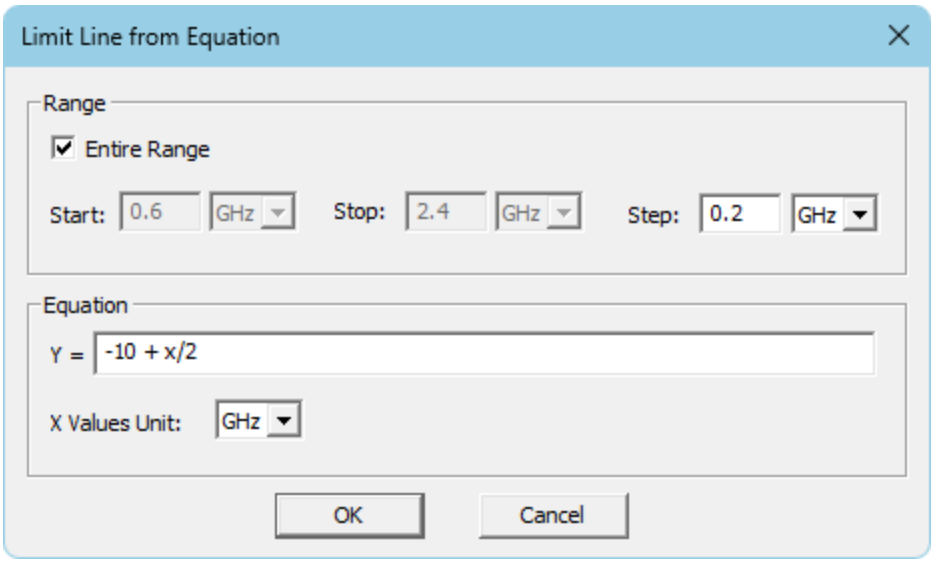


Select **From Selected Curve** to open the *Limit Line From Curve* dialog box.



Select **Specify Equation** to open the *Limit Line from Equation* dialog box. Specify the desired **Equation** (y as a function of x). Choose the **X Values Unit** from the drop-down menu. The available units depend on the type of sweep quantity (frequency, distance, and so on). The default choice is the current X axis unit for the plot.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. You can use the *Edit Dataset* dialog box to do the following:
   - enter the EY values directly.
   - import XY values from a .tab file.
   - export Dataset to a file.

   If you require additional data points, you can use the buttons to **Append Rows** to the Coordinates table. If you select a row in the Coordinates table, you can then use the buttons to **Add Row Above**, **Add Row Below** the selected rows, or **Delete Rows**.

   You can use Shift+click to select multiple adjacent rows, or Ctrl+click to select any rows for deletion.

   > **Note:**
   >
   > Each limit line is associated with a particular Y axis (because it has to be scaled the same way as all the curves associated with the axis, follow its log/linear scale and so on). This Y axis association defaults to the first available Y axis when the limit line is created. However, if the plot contains multiple Y axes, it can be associated with a different Y axis later via its properties tab.

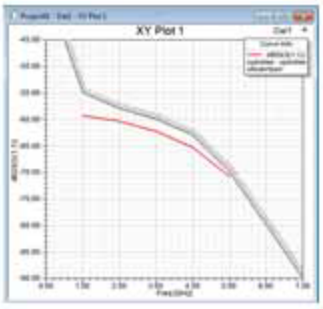
3. You can use the *Limit Line From Curve* dialog box to create a limit line:
   - Define a range using the Entire Curve, or a specified Start and Stop.

   If you uncheck “Entire Curve,” the Start and Stop fields are enabled and initialized based on the zoom level.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

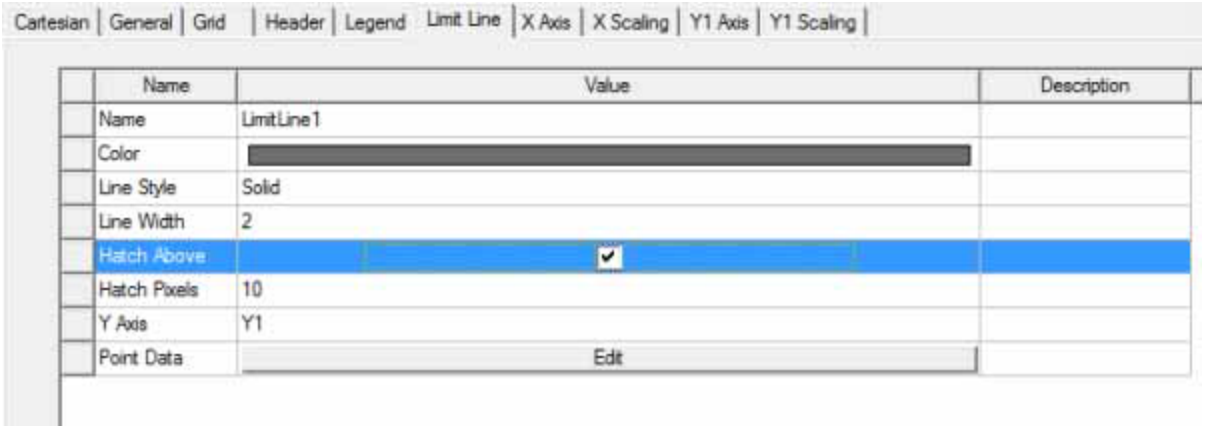
- Shift and Offset relative to the Y, either as a Y Offset value or as a Y Shift %.
- Create Mode as Above Curve, Below Curve, or Above and Below Curve.

4. You can use the *Limit Line from Equation* dialog box to create a limit line based on an equation in the form Y = f(x).
   - Only x variable is allowed
   - x value is linearly sampled and f(x) is evaluated to get y. By default, the entire range for plot is covered. You can uncheck **Entire Range** to specify Start and Stop. You can always edit Step and unit values for x.
   - If y results in NaN, an error is indicated and limit line is shown.
   - If y results in Inf value, then it maps to the MapInfValue specified on its associated y axis.
   - Selecting the limit line added in such a way should show: Start, Stop, Step, and Equation in limit line property window. You can edit these values in the Properties window.
5. Once you click **OK**, the limit line, or lines, you define are added to the plot. Each limit line divides the plot into two regions within the context of its length. By default, the upper region is hatched to designate constraint violation.



6. You can select the limit line in the plot to edit its properties via the **Limit Line** tab of the plot properties.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

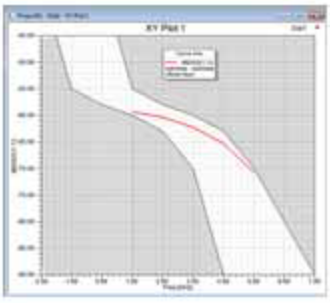


- Line properties: Color, Style and Width
- Y axis association
- Hatch properties
- Hatch width in pixels
- Hatch direction (hatch above or below the Limit line)
- The point definition of the limit line itself.

For a limit line specified as an Equation, the Properties also include:

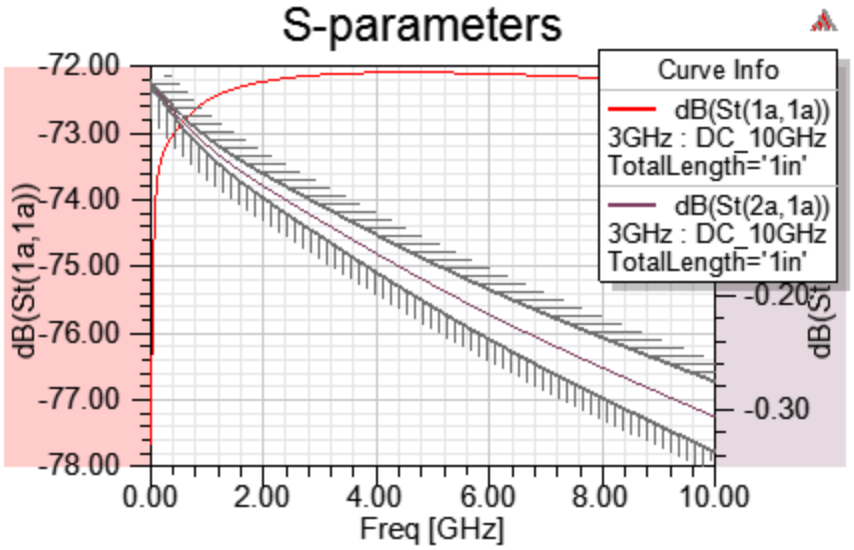
- Start, Stop, Step, and Equation values.

7. If you add a second limit line, you can designate it as hatch below by unchecking **Hatch Above** to produce a tunnel marking upper and lower constraints.



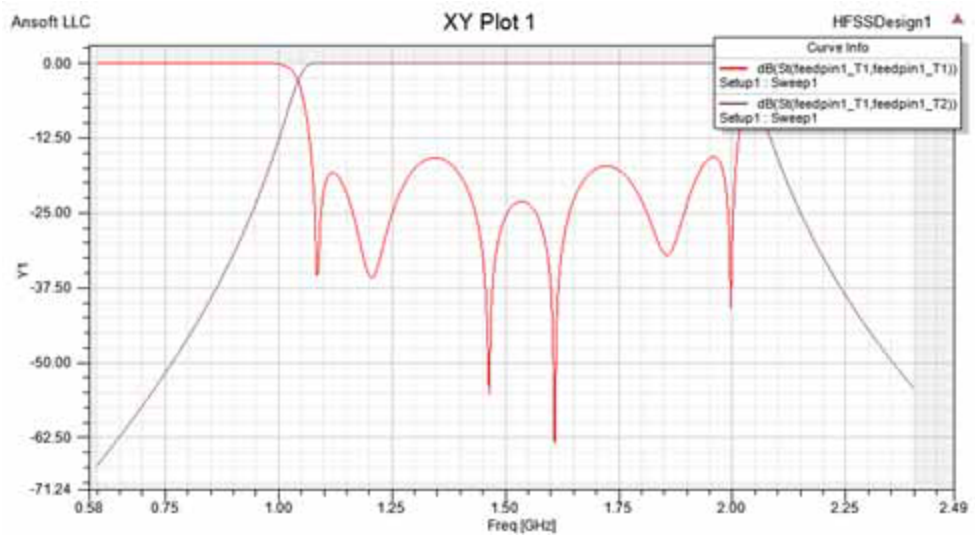
The following example shows the a limit line from curve plot, where the Hatch Above property for the lower limit line has been unchecked.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



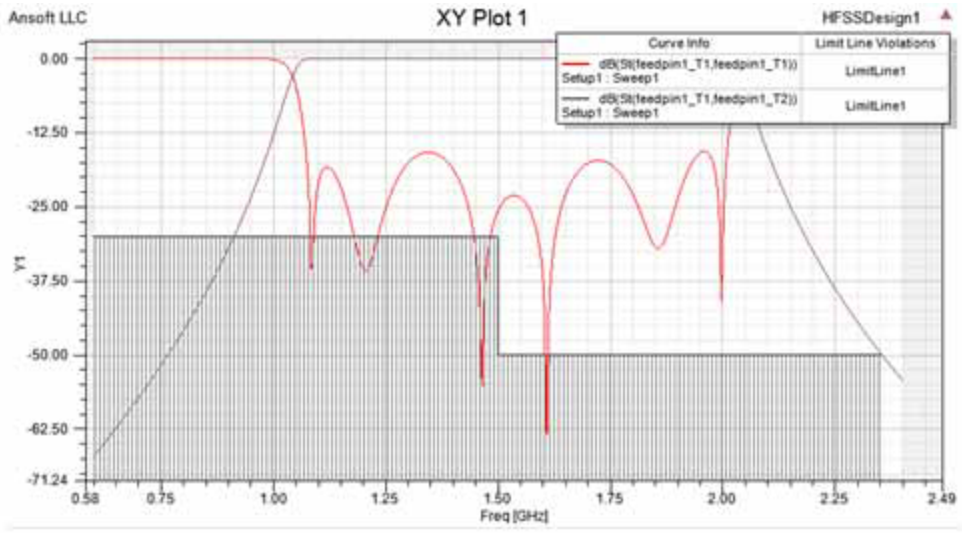
Limit Line Violations

You can use a plotting feature to help you discern whether a curve violates a limit line or not. Consider following plot which shows two curves:

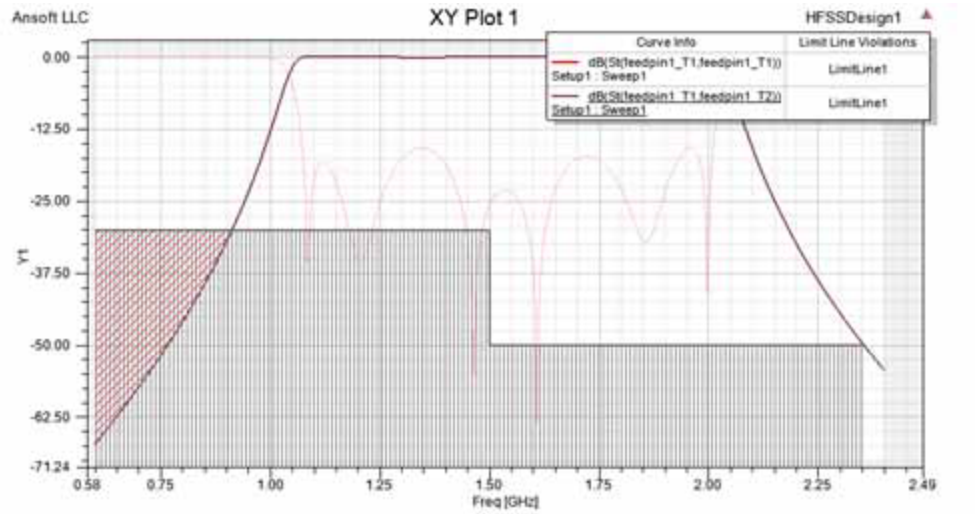


Suppose that the response cannot be below -30 dB until 1.5 GHz, and cannot be below -50 dB at 1.5 GHz or higher. You add a limit line for this requirement in the plot using 'Add a Limit Line' functionality. The plot automatically calculates whether a curve violates this requirement, that is, the limit line and show it in the legends window, as shown in following figure:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



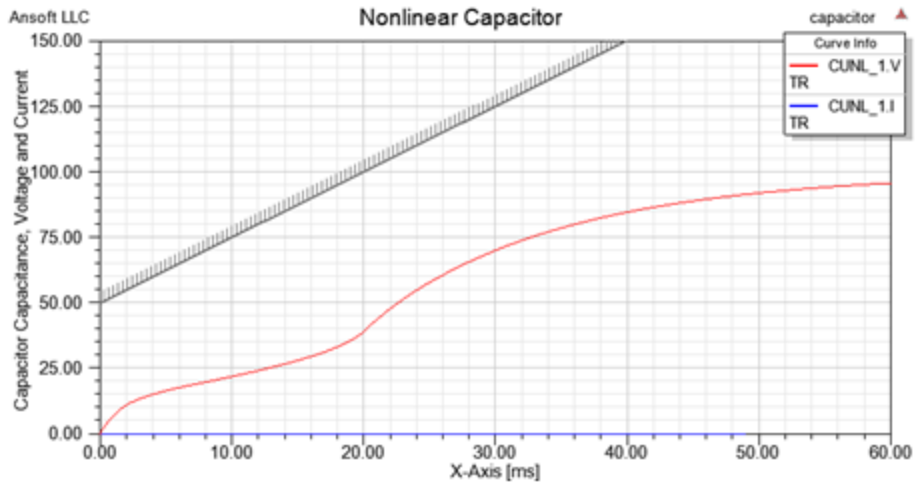
If a curve is selected, then the plot shows the region of the curve that violates the limit line (shaded with slanted red lines), as shown in following figures:



Note:

As shown in the following figure, if no curve violates a limit line, then the limit line hatching is restrained to 10 pixels in length. The minimal hatching keeps the focus on the curve traces rather than the limit lines.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Error Handling

If an error is encountered while calculating Limit Line Violations for a particular plot trace, the *Curve Info* window shows "NaN (<*Limit Line name*>)" under the *Limit Line Violations* column for that curve. For example, if a value is not available or goes to infinity at a particular point, the program is unable to evaluate that point. One of the coordinates is "Not-a-Number" (NaN).

Sweeping a Variable in a Report

In Maxwell, a swept variable is a variable that typically has more than one value. You can plot any calculated or derived quantity against one or more of the swept variable's values.

For large projects or projects with many variables, you may obtain faster post processing before generating a solution by selecting which variables function as Sweep variables. Only the variables with Sweep enabled are indexed for post processing. See Adding a Design Variable and Adding a Project variable.

To specify the swept variable values to plot a selected quantity against:

1. In the **Report** dialog box, select the variable from the X (Primary Sweep) drop-down menu.
2. To modify the values that will be plotted for a variable:
    a. Click the ellipsis **[...]** button on the **X (Primary Sweep)** line of the **Report** dialog to displays a pop-up list of the possible values.
    b. Select **Use all values** or click the Edited button to display a dialog that lets you specify the sweeps to use.

    All of the selected variable's values will be plotted.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** When specifying a Time sweep for a solution that is defined with varying save field, it is recommended that you select **Use all values**.

Sweeping Values Across a Distance

1. If you are plotting a field quantity along a line, define a polyline object in the problem region.
2. In the *Report* dialog box, click the line geometry of interest in the **Geometry** list.
3. Specify the quantities you want to plot along the axes.
4. For the **X (Primary Sweep)**, select the **Distance** variable.

   The values at which the selected quantity or quantities will be plotted are listed to the right. By default, a post-processing polyline object is divided into 100 equally spaced points.

5. For Near field, to plot the selected quantity or quantities at every point on the line, select **All Values**.

   For Near field, to plot the selected quantity, or quantities, at specific points on the line, clear the **All Values** option, and then select the point values on which you want to plot.

> **Note:**
>
> All maximum near-field data calculated by Maxwell is at their maximum over the selected line object; if you plot the parameter over a sweep of values, the parameter will have the same value at each point on the plot.

Selecting a Function for a Plot

The value of a quantity being plotted depends upon its mathematical function, which you select from the **Trace** tab **Function** list in the *Report* dialog box. The available, valid functions depend on the type of quantity (real or complex) that is being plotted. The function is applied to the quantity which is implicitly defined by all the swept and current variables. For example, "S(11)" is the value of the S-parameter for every swept combination of variables (e.g., "height", "frequency"). (A smaller set of functions appears for the Function list in the Output Variables dialog.)

These functions can also be applied to previously specified Quantities and Functions as **Range Functions** when using the Set Range Function dialog box.

Some of these functions can operate along an entire curve. These are: min, max, integ, avg, rms, pk2pk, cang_deg and cang_rad.

You can select from the following functions in the **Trace** tab **Function** list or type them directly into the Yor X field, if necessary.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

abs	Absolute value of the simulation quantity which results in a number that is always positive.
acos	Arc cosine (the inverse function of a cosine).
acosh	Inverse hyperbolic arc cosine.
ang	Magnitude of an angle.
ang_deg	Angle (phase) of a complex number, cut at +/-180. Returns angular values in degree units, and not suitable for use in Optimetrics, which works in SI values and evaluates ang_deg expressions in radians. See ang_deg_val.
ang_deg_val	Angle (phase) of a complex number in unitless degree values, and suitable for use in Optimetrics, which works in SI values. Returns simple numbers.
ang_rad	Angle in radians.
arg	Argument of a complex number. It is the angle the complex number makes with the positive x axis. Same as **ang_deg**.
asin	Arc sine (the inverse function of sine).
asinh	Inverse hyperbolic sine.
atan	Arc tangent (the inverse function of a tan).
atanh	Inverse hyperbolic tan.
atan2	Two argument function. For non-0 x,y, the function returns the angle between the + x-axis and the given x,y coordinates.
avg	Returns the average of the values of the selected quantity. avg = (Area between the curve and the X-axis) / (X length of the curve)
avgabs	Returns the mean of the absolute value of the selected quantity.
bandwidth	Returns the 3dB bandwidth of the selected simulation quantity. For bandwidth, the calculation is based on 3dB below the maximum peak.
cang_deg	Cumulative angle (phase) of the first parameter (a complex number) in degrees, along the second parameter (typically sweep variable). Returns a double precision value cut at +/-180. Returns angular values in degree units, and not suitable for use in Optimetrics, which works in SI values and evaluates cang_deg expressions in radians. See cang_deg_val.
cang_deg_val	Cumulative angle (phase) of the first parameter of the selected simulation quantity in unitless degree values and suitable for use in Optimetrics, which works in SI values. . Returns simple numbers.
cang_rad	Cumulative angle of the first parameter in radians along a second parameter (typically a sweep variable). Returns a double precision value.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

cmplx(*re*, *im*)	A complex number, where *re* is the real part and *im* is the imaginary part.
conjg	Conjugate of the complex number.
cos	Cosine.
cosh	Hyperbolic cosine.
crestfactor	Returns the crest factor (peak/RMS) for the selected quantity.
cum_integ	The cumulative integral function returns a set of values that have the same length as the original set of points (the first element will always be zero). Element *I* of the set returned by cum_integ is the integral of elements 1 through *I* of the original data set.
cum_sum	The cumulative sum function returns a data set that has the same length as the original set of points. Element *I* of the set returned by cum_sum is the sum of elements 1 through *I* of the original data set.
dB(x)	20*log10(|x|) to base 10.
dBc	Decibels relative to the carrier. It is the power ratio of the signal to a carrier signal. Gives the relative signal strength.
dBm(x)	10*log10(|x|) +30.
dBm	(for electric field quantities) is computed as: 20.0*log10(x) + 60.0
dBu	(for electric field quantities) is computed as: 20.0*log10(x) + 120.0
dBW(x)	10*log10(|x|).
dB10	10*log(|x|) to base 10.
dB10normalize	10*log [normalize(mag(x))].
dB20	20*log(x) to base 10.
dB20normalize	20*log [normalize(mag(x))].
deadtime	Obtains the latest time when the qtyl is within a tolerance of zero.
delaytime	Obtains the time from zero to 50% of the target point.
degel	Conversion from degrees electrical to seconds with respect to Hz.
deriv	Derivative of a given parameter.
distortion	Returns the total distortion for the selected simulation quantity and an additional argument frequency, which is the frequency in Hz at which to calculate the fundamental RMS of the simulation quantity.
even	Returns 1 if integer part of the number is even; returns 0 otherwise.
exp	Exponential function (the natural anti-logarithm) of the simulation quantity.
fmod	nReturns the double precision remainder of x/y.
formfactor	Returns the form factor (RMS/Mean Absolute Value) for the selected quantity.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

fundamentalmag	Returns the RMS value of the fundamental frequency for the selected quantity, and an additional argument, Frequency, which specifies the fundamental frequency.
gaincrossover	Returns the gain crossover frequency (where the gain is 0 dB) of the selected simulation quantity in Hz.
gainmargin	Returns the gain margin in dB at the phase crossover frequency of the selected simulation quantity. It also requires a reference simulation quantity to which the measured quantity is compared and the AC magnitude and phase angle of the reference quantity. These are entered as the arguments Reference Channel, Base Source Magnitude, and Base Source Angle.
iae	Returns the integral of the absolute deviation of the selected quantity from a target value that is entered via the additional argument.
if	if(cond_exp,true_exp, false_exp).
im	Imaginary part of the complex number.
int	Truncated integer function.
integ	Integral of the selected quantity. Uses trapezoidal area.
integabs	Absolute value of integral.
ise	Returns the integral of the squared deviation of the selected quantity from a target value that is entered via an additional argument.
itae	Returns the time-weighted squared deviation of the selected quantity from a target value that is entered via an additional argument.
itse	Returns the time-weighted squared deviation of the selected quantity from a target value that is entered via an additional argument. To use this function, you need to open the Add Trace Characteristics dialog and select the Error category.
j0	Bessel function of the first kind (0^{th} order).
j1	Bessel function of the first kind (1^{st} order).
jn	Bessel function of the first kind (nth order).
ln	Natural logarithm.
log	Natural logarithm (same as ln).
log10	Logarithm base 10.
lowercutoff	Returns the lower 3dB frequency of the selected simulation channel in Hertz.
lsidelobeX	The 'x' value for the left side lobe: the next highest value to the left of the max value.
lsidelobeY	The 'y' value for the left side lobe: the next highest value to the left of the max value.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

mag	Magnitude of the complex number.
max	Returns maximum value of the simulation quantity.
max_swp	Returns maximum value of a sweep.
max2	Maximum value of the two simulation quantities. For example, **max2 (a,b)** will plot maximum of **a** and **b** for a particular instance.
mean	Returns the average in the set of quantities selected. mean = sum(all y-value) / (number of y-values)
min	Returns the minimum value of the simulation quantity.
min_swp	Returns the minimum value of a sweep.
min2	Minimum value of the two simulation quantities. For example, **min2 (a,b)** will plot minimum of **a** and **b** for a particular instance.
mod	Returns the double precision modulus (x - floor(x/y)*y). Not supported in solvers.
nint	Nearest integer.
none	Returns null value.
normalize	Divides each value within a trace by the maximum value of the trace. ex. normalize(mag(x)).
odd	Returns 1 if integer part of the number is odd; returns 0 otherwise.
overshoot	Calculates peak overshoot given a threshold value and number of evenly spaced points over entire time range.
peakgain	Returns the peak value of gain of the selected simulation quantity in dB.
peakgainfreq	Returns the frequency in Hz at which the peak gain of the selected simulation quantity occurs.
polar	Coverts the complex number in rectangular co-ordinates to polar co-ordinates.
per	Returns the period of a simulation quantity.
phasecrossover	Returns the phase crossover frequency, at which the phase is -180 degrees, in Hz for the selected simulation quantity.
phasemargin	Returns the phase angle in degrees at the gain crossover frequency of the selected simulation quantity.
pk2pk	Peak to peak. Difference between max and min of the first parameter over the second parameter. Returns the peak-to-peak value for the selected simulation quantity.
pkavg	Returns the ratio of the peak to peak-to-average for the selected quantity.
pmax	Maximum period of the selected simulation quantity.
pmin	Minimum period of the selected simulation quantity.
pow	Raises x to the power of y; pow(x,y).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

prms	Period Root Mean Square.
pulsefall9010	Returns the pulse fall time of the selected quantity according to the 90%-10% estimate.
pulsefront1090	Returns the pulse front time of the selected quantity according to the 10%-90% estimate.
pulsefront3090	Returns the pulse front time of the selected quantity according to the 30%-90% estimate.
pulsemax	Returns the pulse maximum from the front and tail estimates for the selected quantity.
pulsemaxtime	Returns the time at which the maximum pulse value of the selected quantity is reached.
pulsemin	Returns the pulse minimum from the front and tail estimates for the selected quantity.
pulsemintime	Returns the time at which the minimum pulse value of the selected quantity is reached.
pulsetail50	Returns the pulse tail time of the selected quantity from the virtual peak to 50%.
pulsewidth5050	Returns the pulse width of the selected quantity as measured from the 50% points on the pulse front and pulse tail.
pwl	Piecewise Linear.
pwl_periodic	Piecewise Linear for periodic extrapolation on x.
pwlx	Piecewise Linear x with linear extrapolation on x.
pw_minus	Pulse width of the first negative pulse.
pw_minus_avg	Returns the average of the negative pulse width input stream.
pw_minus_max	Returns the maximum pulse width of the negative pulse of input stream.
pw_minus_min	Returns the minimum pulse width of the negative pulse of input stream.
pw_minus_rms	RMS of the negative pulse width input stream.
pw_plus	Pulse width of the first positive pulse.
pw_plus_avg	Average of the positive pulse width input stream.
pw_plus_max	Max. Pulse width of the positive pulse of input stream.
pw_plus_min	Min. Pulse width of the positive pulse of input stream.
pw_plus_rms	RMS of the positive pulse width input stream.
re	Real part of the complex number.
rect	Converts the complex number in polar to rectangular co-ordinates.
rem	Returns the fractional part of a decimal number such that rem(x) = x-int (x) Syntax: rem(x)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ripple	Returns the ripple factor (AC RMS/Mean) for the selected quantity.
risetime	Obtains the time taken to go from 10% to 90% of target point.
rms	Returns the root mean square value of the selected quantity.
rmsAC	Returns the AC RMS for the selected quantity.
root	nth root function.
rSidelobeX	Returns the X value of right side-lobe occurrence.
rSidelobeY	Returns the Y value of right side-lobe occurrence.
settlingtime	Returns the latest time at which the value of the selected simulation quantity fell outside its tolerance band. The target value of the quantity and the +/- bandwidth of the tolerance band are the additional args.
sgn	Sign extraction.
sin	Sine.
sinh	Hyperbolic sine.
slidingmean	Returns the moving average value of the selected simulation quantity (specified by the first argument). The average is calculated over a period (specified by the second argument).
slidingrms	Returns the moving RMS value of the selected simulation quantity (specified by the first argument). The RMS value is calculated over a period (specified by the second argument).
sqr	Square of the selected simulation quantity.
sqrt	Square root of the selected simulation quantity.
stddev	Returns the standard deviation of given values.
sum	Returns the sum of the given values.
tan	Tangent.
tanh	Hyperbolic tangent.
undershoot	Calculates peak undershoot given a threshold value and number of evenly spaced points over entire time range.
uppercutoff	Returns the upper 3dB frequency of the selected simulation channel in Hz.
variance	Calculates the variance of the given values.
XAtYMax	Threshold crossing time: report first time (x value) at which an output quantity crosses YMax.
XAtYMin	Threshold crossing time: report first time (x value) at which an output quantity crosses a user definable threshold.
XAtYVal	Returns the X value at the first occurrence of Y value.
XWidthAtYVal	Returns the X width between the first 2 occurrence of Y value.
xdb10beamdwidth	Width between left and right occurrences of values 'x' db10 from max. Takes 'x' as argument (3.0 default). To use this function, you need to

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	open the Add Trace Characteristics dialog and select the Radiation category.
xdb20beamwidth	Width between left and right occurrences of values 'x' db20 from max. Takes 'x' as argument (3.0 default). To use this function, you need to open the Add Trace Characteristics dialog and select the Radiation category.
YAtXMax	Returns the X value at maximum value of Y.
YAtXMin	Returns the Y value at minimum value of X.
YAtXVal	Returns the Y value at the first occurrence of X value.
y0	Bessel function of the second kind (0^{th} order).
y1	Bessel function of the second kind (1^{st} order).
yn	Bessel function of the second kind (nth order).

Selecting Solution Quantities to Plot

Each trace in a report includes a quantity that is plotted along an axis. The quantity being plotted can be a value that was calculated by Maxwell, such as L_{11}, or a value from a calculated expression.

To select a parameter to plot:

1. In the **Report** dialog box, select one of the solution type dependent categories. Some of the available categories and descriptions of their associated quantities are listed below. The actual list you see will depend upon the solution type and solution setup for your design. The selected category provides the default name for the plot, such as Force Plot *n*. You can edit the plot names in the project tree and the plot header in the report properties.

Variables	Intrinsic variables, such as frequency or theta, or user-defined project variables, such as the length of a quarter-wave transformer.
Output Variables	Derived quantities from the original field solution.
L, R, or Z	Inductance, Resistance, and Impedance matrix entries when the matrix parameter is defined.
Force or Torque	Force or Torque values when the parameter is defined.
Loss	▪ **Loss quantities for 2D and 3D transient solutions include:** CoreLoss, EddyCurrentLoss, ExcessLoss, HysteresisLoss, SolidLoss, StrandedLoss, StrandedLossR, and StrandedLossAC. [1] The availability of solution data for CoreLoss, EddyCurrentLoss,

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ExcessLoss, and HysteresisLoss depends on the material definitions in the design:

- If the design is hysteresis model-based, the solver provides solution data to plot only for HysteresisLoss, EddyCurrentLoss, and TotalLoss.
- If the design contains a non-hysteresis based model, core loss model type: Electrical Steel, the solver provides solution data to plot for all four quantities (CoreLoss, EddyCurrentLoss, ExcessLoss, and HysteresisLoss).
- For all other cases (for example: linear, power ferrite), the solver provides solution data to plot only for CoreLoss.

> **Note:** For 2D transient solutions only, object-based quantities of CoreLoss, EddyCurrentLoss, ExcessLoss, and HysteresisLoss are also calculated, and can be selected for plotting, for objects whose **Core Loss Setting** has been checked in the **Set Core Loss** dialog box.

- **Loss quantities for 2D and 3D AC magnetic solutions include:** CoreLoss, SolidLoss, StrandedLoss, StrandedLossAC, and StrandedLossR.

End Connection

- If a 2D transient End Connection excitation has been assigned, quantities include: Current, InducedVoltage, RingCurrent (the current in the first segment of the ring), and SolidLoss.
- If a 2D AC magnetic End Connection excitation has been assigned, quantities include: Current and InducedVoltage.

Expression Cache The values of expressions listed in the Expression cache of the Solution setup can be plotted, for example, as a function of adaptive pass to monitor their convergence.

Expression Converge This can be used to plot convergence, as a function of adaptive pass, of expressions defined in the Expression cache of the Solution setup. When defining the report, for "Context", select Solution: *SetupN:Adaptive Pass*.

[1]*NOTE: StrandedLoss includes DC losses only; while StrandedLossAC includes DC losses and additional AC losses due to skin effect and proximity effect.*

2. Select a quantity to plot from the **Quantity** list. The available quantities depend on the selected category and the setup of the design.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting a Field Quantity to Plot

When plotting field quantities, the quantity can be a value that was automatically calculated by Maxwell such as the magnitude of S_{11}, a value from a calculated expression, or an intrinsic (inherent) variable value such as frequency or phase.

To select a field quantity to plot:

1. When you create the report, specify the Report Type as "Fields" and the plot type (for example, radiation pattern.)
2. In the *Report* dialog box, select Geometry for the Context, unless you are plotting scalar (for example, integration). For example, to plot near-field values across a sphere, you select the sphere object from the **Geometry** list in the *Traces* dialog box when you create a report.
3. In the *Report* dialog box, select one of the following categories. The selected category provides the default plot name. You can edit the plot names in the project tree and the plot header text in the report synchronizes.

Variables	Intrinsic variables, such as frequency or phase, or user-defined project variables, such as the length of a quarter-wave transformer.
Output Variables	User defined expressions applied to derive quantities from the original field solution.
Calculator Expressions	Includes scalar and vector field quantities automatically calculated by Maxwell, as well as derived field quantities that are defined by calculated expressions you set up in the Fields Calculator.

4. Select a quantity to plot from the **Quantity** list. The available quantities will depend upon the selected category and the setup of the design. See Field Quantities list for definitions.

Plotting Imported Solution Data

1. In the **Solution** drop-down menu of the *Report* dialog box, click the imported data you want to plot.
2. Follow the procedure for creating a report.

Postprocessed Quantities

Both preprocessing and postprocessing quantities are available when creating reports.

For example, when you create a Magnetostatic Report for the **Magnetostatic Solver** the following are four of the quantities that appear in the **Category** list in the **Report** dialog box:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **L** – The postprocessed value of inductance (for example, the inductance of a 1-turn coil).
- **Lnom** – The inductance quantity that is solved with no post processing.
- **MagFlux** – The postprocessed value of the total magnetic flux as seen by the coil (current source) including all sources (current, magnets, etc.).
- **MagFluxNom** – The total magnetic flux quantity that is solved with no postprocessing as seen by the coil (current source) including all sources (current, magnets, etc.)

Similarly, for an AC Magnetic Report, the following are three of the quantities that appear in the **Category** list:

- **Winding** – If the design includes windings, Flux Linkage, Induced Voltage, Input Current, Input Voltage, and Current quantities are available (depending on the winding type). See also Viewing Winding Results (AC Magnetic).
- **Loss** – Hysteresis Loss (Maxwell 2D AC Magnetic solutions only), Core Loss, Solid Loss, Stranded Loss Stranded Loss R, and Stranded Loss AC. See also Viewing Loss Results (AC Magnetic).
- **End Connection** (Maxwell 2D AC Magnetic solutions only) – Induced Voltage, and Current. See also Viewing End Connection Results (2D AC Magnetic).

Per-Winding Loss Outputs for Maxwell 2D/3D Transient and AC Magnetic Designs

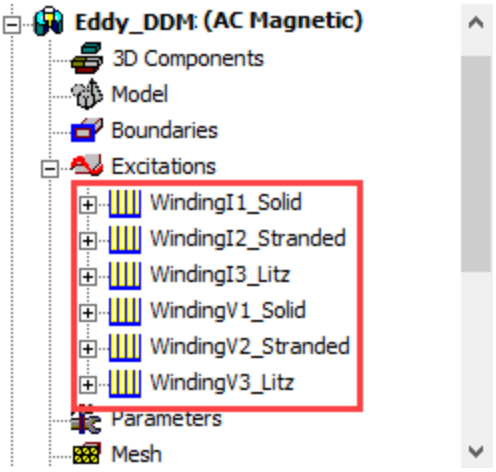
Maxwell 2D/3D transient (T-omega only) and AC Magnetic designs support loss quantity outputs for each winding when creating reports.

For the various winding types, the supported per-winding loss quantities are

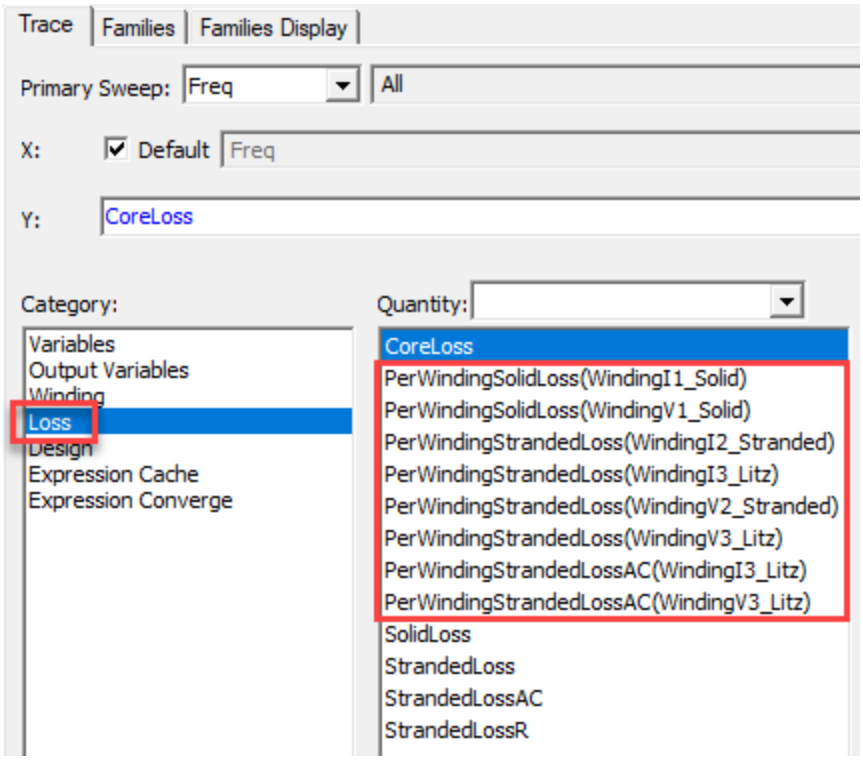
- Solid winding
    - **SolidLoss**
- Stranded winding without Litz wire
    - **StrandedLoss**
- Stranded winding with Litz wire
    - **StrandedLoss**
    - **StrandedLossAC**

For example, the winding assignments in the AC Magnetic design shown below,

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



result in the following per-winding loss quantities available in the reporter.

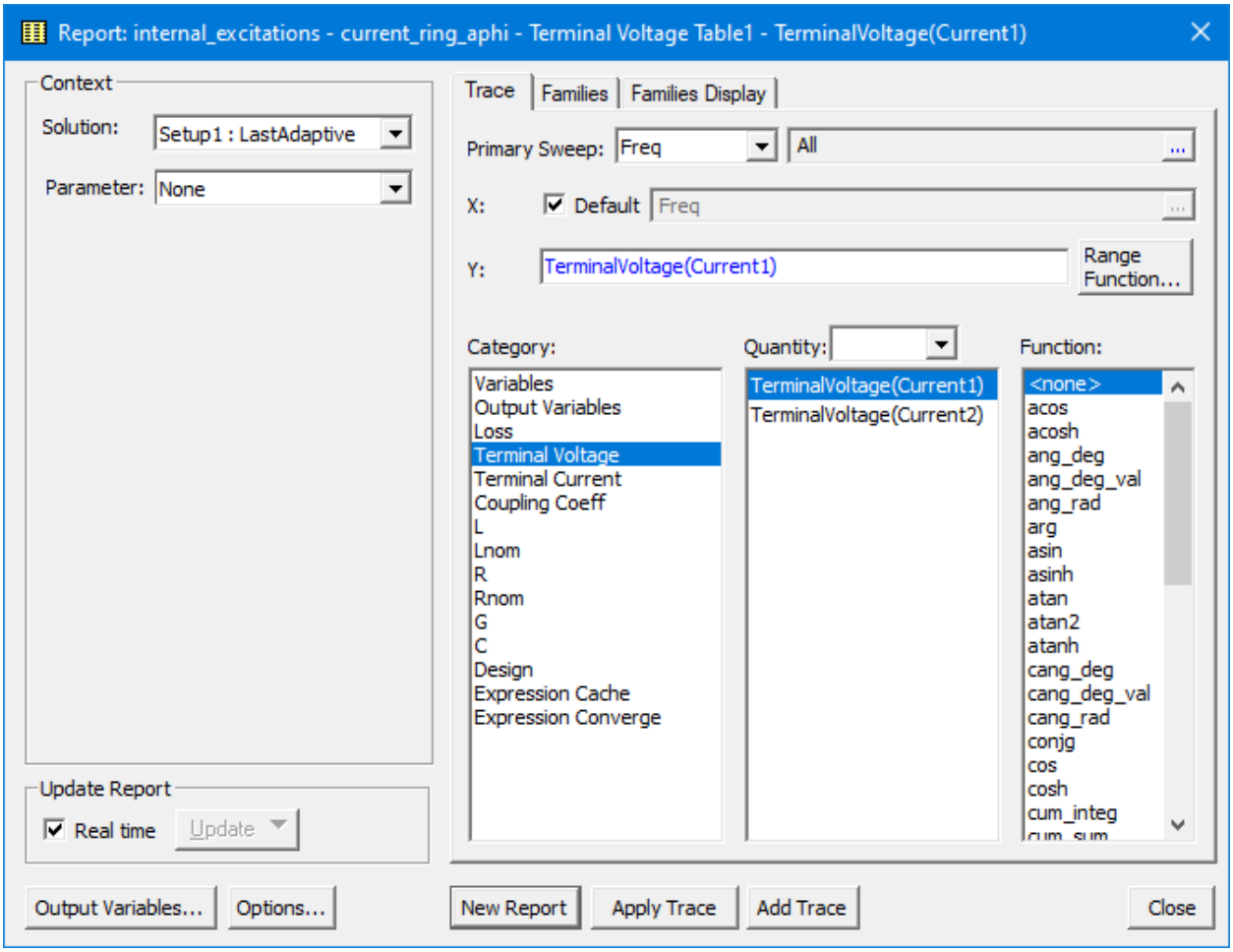


Note: Per-winding loss quantities will not affect legacy project reports.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Terminal Current/Voltage Output

For Eddy and Transient A-Phi solvers, it is possible to compute the current and voltages on terminals.



The terminal current is computed by

$$I = \int J \cdot n ds$$

where ***n*** is the surface normal pointing inwards to the domain in the case of single potential excitation models, and it is the excitation direction for double potential excitation models.

The terminal voltage is the electric potential on the surface in the case of single potential excitation models and the voltage drop in the case of double potential excitation models.

Setting a Range Function

To apply a range function to the Y, Z, or Mag component of a trace:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

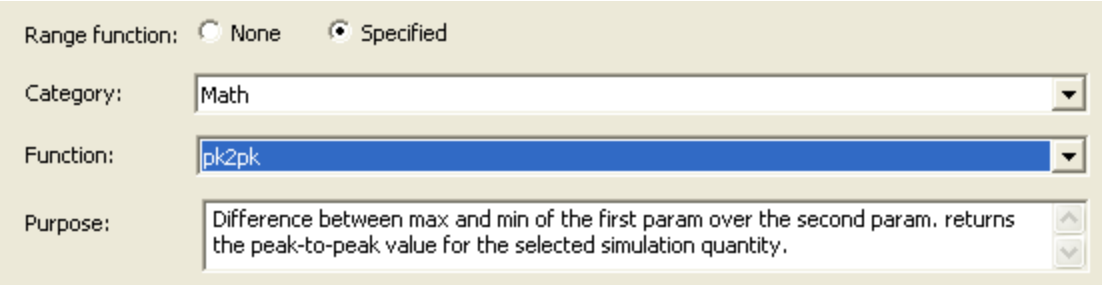
1. Click **Range Function** in the *Report* dialog box.

   This opens the *Set Range Function* dialog box. The functions available are the same as described in the Selecting a Function section, with the exception of those for the Eye Measurements category.

2. To enable Range function selection, click the Specified radio button.

   Selecting the None radio button disables the Range Function fields.

3. Select the **Category**, and then an associated Function to apply. The available categories depend on the plot, and Category enables the display of associated functions.



   Given a selected Function, and Category, the **Set Range Function** dialog displays a text field that explains the Purpose of the function. See figure above.

   Selecting a function causes the display of a description in the **Purpose** field. If the function requires a value (such as the XatYVal Math function or the pw_minus_max Pulse Width function), the following table the function field displays the name, editable value field, unit, and description.

4. Use the **Over Sweep** drop-down menu to select from available sweeps.
5. To select from available Sweeps, or to edit them, use the ellipsis [**...**] button and uncheck **Use All Sweeps**.

   This enables a list of the sweeps. The sweep(s) you select is displayed on the Over Sweep line. You can use the buttons to **Clear All Selections** or **Select All** sweeps.

6. Select the Sweeps **Default** or **Edited** radio buttons to specify whether to accept the default or edited sweeps.
7. To edit the sweeps further, select the ellipsis button to display an *Edit Sweep* dialog box.

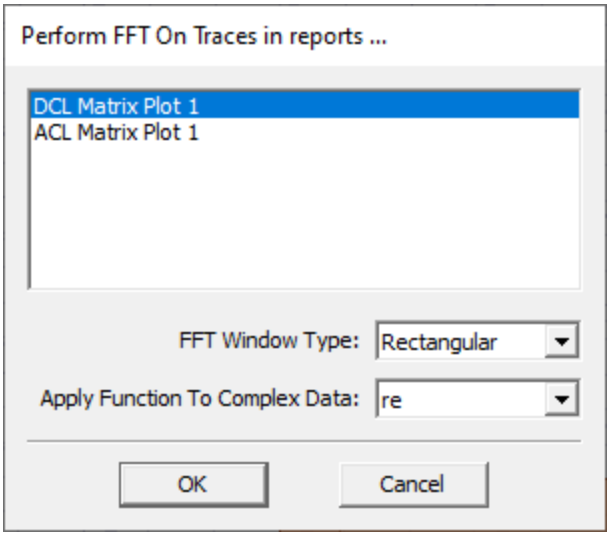
   For frequency variables, this lets you specify a single value, linear step, linear count, decade count, octave count, or exponential count. You can **Add** legal values to the list of sweep values, **Update** the list for changes, or **Delete** selected entries.

8. Click **OK** to apply the range function.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Perform FFT on a Report

You can perform FFT on an existing 2D plot by using the **Results** > **Perform FFT** command. You can perform TDR on an existing 3D plot by using **Maxwell** > **Results** > **Perform FFT**. This opens the **Perform FFT on Traces in Reports** dialog box.



1. Select the report you want from the list.
2. Select the FFT window type to apply. Winding functions cause the FFT of the signal to have non-zero values away from ω. Each window function trades off the ability to resolve comparable signals and frequencies versus the ability to resolve signals of different strengths and frequencies.
3. Select the function to apply to complex data.

   The new report appears under **Results** in the **Project Manager**. The new report name prefixes FFT to the name of the original report. Trace names are also prefixed with FFT.

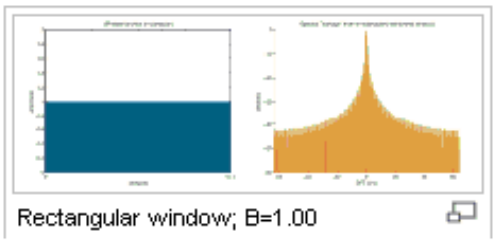
FFT Window Functions

The window type list for **Perform FFT on Report** includes:

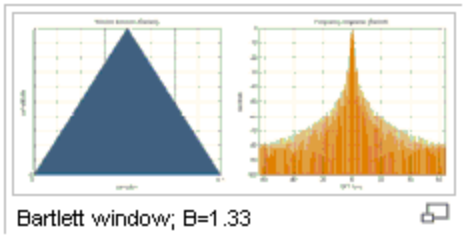
Window Function	Preferred Use
Rectangular	A low dynamic range function offering good resolution for signals of comparable strength. Poor when signals have very different amplitudes. $w(n)=1$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Window Function	Preferred Use

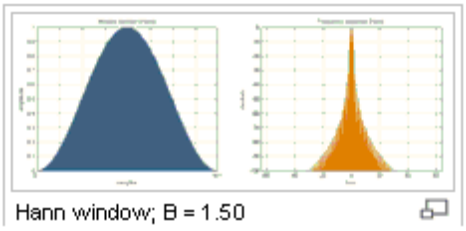


Tri — A Bartlett window with the endpoints valued at zero.



Van Hann — A moderate dynamic range function, designed for narrow band applications.

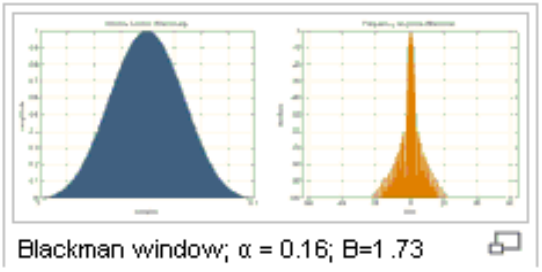
$$w(n) = 0.5(1 - cos\frac{2\pi n}{N-1})$$



Blackman — A high dynamic range function, with lower resolution, designed for wide band applications.

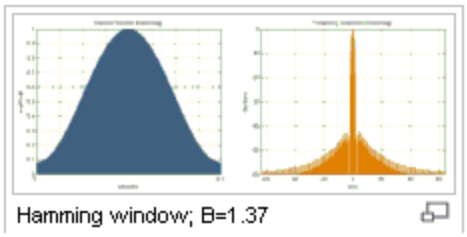
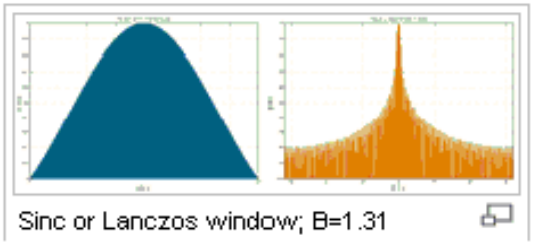
$$w(n) = \alpha_0 - \alpha_1 cos(\frac{2\pi n}{N-1}) + \alpha_2 cos(\frac{4\pi n}{N-1})$$

where $a_0 = (1-\alpha)/2$; $\alpha_1 = 1/2$; $\alpha_2 = \alpha/2$



Hamming — A moderate dynamic range function, designed for narrow band applications. It

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Window Function	Preferred Use
	minimizes the maximum sidelobe. $w(n) = 0.54 - 0.46cos\left(\frac{2\pi n}{N-1}\right)$ 
Lanczos	The Lanczos window offers a windowed form of the infinite sinc filter, providing the central lobe of a horizontally stretched sinc, sinc(x/a) for -a £ x £ a. 
Weber	
Welch	This approach applies a parabola-shaped window to the frequency domain data. It is based on the Bartlett method but splits the signal into overlapping segments, which are then windowed. The intent is to balance the influence of data in the center of the function.

Apply FFT to Report Functions

The choices in the Perform FFT on Traces in Reports dialog include:

ang_deg	**Angle (phase) of a complex number, cut at +/- 180**
ang_deg_val	**Angle (phase of a complex number in unitless degree values. Returns simple numbers.**
ang_rad	**Angle in radians**
arg	

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

cang_deg	**Cumulative angle (phase) of the first parameter (a complex number) in degrees, along the second parameter (typically sweep variable). Returns a double precision value cut at +/-180.**
cang_deg_val	**Cumulative angle (phase) of the first parameter of the selected simulation quantity in unitless degree values. Returns simple numbers.**
cang_rad	**Cumulative angle of the first parameter in radians along a second parameter (typically a sweep variable) Returns a double precision value.**
dB(x)	**20*log10(|x|)**
dB 10normalize	**10*log [normalize(mag(x))]**
dB 20normalize	**20*log [normalize(mag(x))]**
dBc	
GetGroupDelay	
im	**Imaginary part of the complex number**
mag	**Magnitude of the complex number**
normalize	**Divides each value within a trace by the maximum value of the trace. ex. normalize (mag(x))**
re	**Real part of the complex number**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The evaluated value of an expression is always interpreted in SI units. However, when an angle quantity is plotted in a report, you have the option to plot values in units other than SI. If you want to plot the polar angle of a complex simulation result, S_{11} say, you can choose between ang_deg(S_{11}) and ang_rad(S_{11}). Both of these return the exact same angle quantity but in degree and radian units respectively.

Note that when used in expressions, some surprising outcomes might result. For example, the expression "1+ang_deg(S_{11})" represents an 'angle' and the number "1" is treated as "1 rad". The angle SI unit is attached to any unitless number that is added/subtracted from an angle value. If you want to treat "1" as degrees, make it explicit and use "1deg + ang_deg(S_{11})" instead.

If you are interested in unitless degree values, two additional functions exist: ang_deg_val(S_{11}) and cang_deg_val(S_{11}). These return simple numbers and are treated as such by any expression. If the complex S_{11} lies on the positive Y axis say, ang_deg_val (S_{11}) would be 90 and "1 + ang_deg_val(S_{11})" will be 91.

Perform TDR on Report

You can perform TDR on an existing 2D plot by using the **Results** > **Perform TDR on Report** command. You can perform TDR on an existing 3D plot by using the **HFSS 3D Layout** > **Results** > **Perform TDR on Report** command. This opens a **Perform TDR on Traces in reports** dialog box.

1. Select the report you want from the list in the dialog box.
2. Specify the input signal as **Step** or **Impulse**.
3. Set the **Rise Time**.
4. In the **Type** drop-down menu, select which window type to apply.

   Windowing functions cause the FFT of the signal to have non-zero values away from ω. Each window function trades off the ability to resolve comparable signals and frequencies versus the ability to resolve signals of different strengths and frequencies.

5. To specify a window width, enter a percentage into the **Width (%)** field.
6. If you select the Kaiser function, specify a number in the **Kaiser Parameter**.

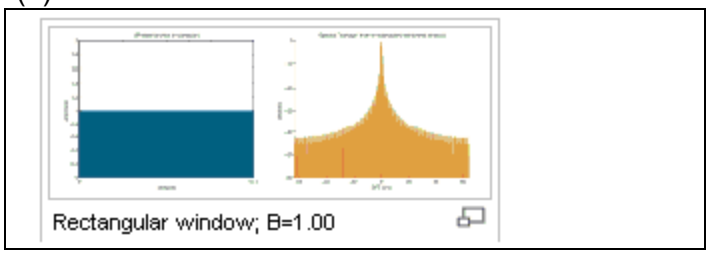
The new report displays and appears in the Project tree. The new report name prefixes TDR to the name of the original report. Trace names are also prefixed with TDR.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TDR Windowing Functions

Windowing functions cause the FFT of the signal to have non-zero values away from ω. Each window function trades off the ability to resolve comparable signals and frequencies versus the ability to resolve signals of different strengths and frequencies. The window type list includes the following:

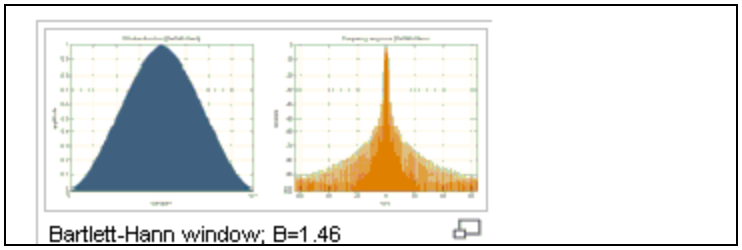
Window Function	Preferred Use
Rectangular	A low dynamic range function offering good resolution for signals of comparable strength. Poor when signals have very different amplitudes. $w(n)=1$.



Bartlett — A high dynamic range function, with lower resolution, designed for wide band applications.

$$w(n) = a_0 - a_1 \left| \frac{n}{N-1} - \frac{1}{2} \right| - a_2 \cos\left(\frac{2\pi n}{N-1}\right)$$

where a_0=0.62; a_1=0.48; a_2=0.38

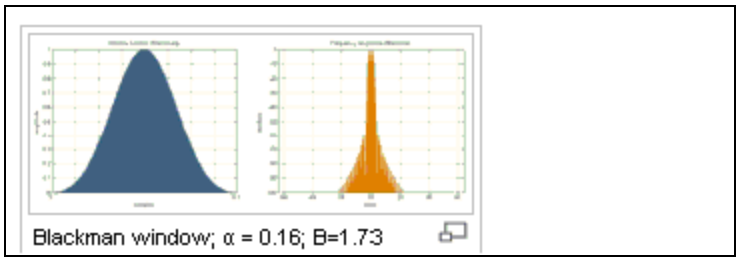
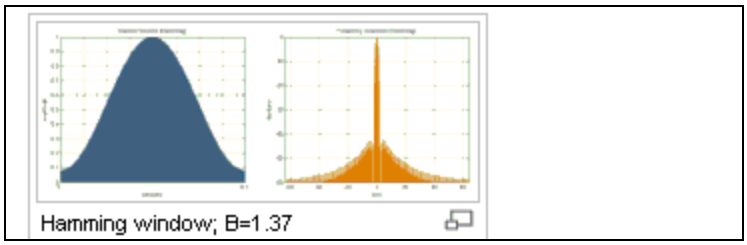
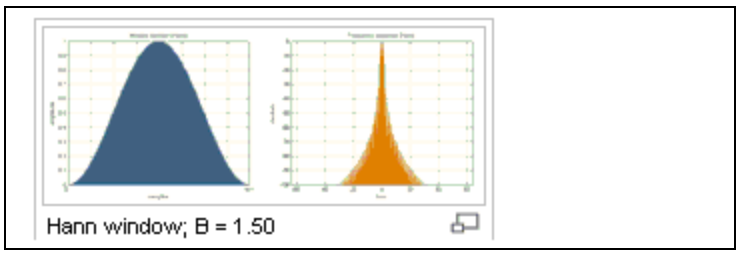
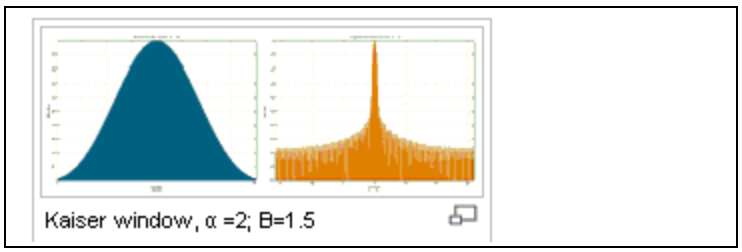


Blackman — A high dynamic range function, with lower resolution, designed for wide band applications.

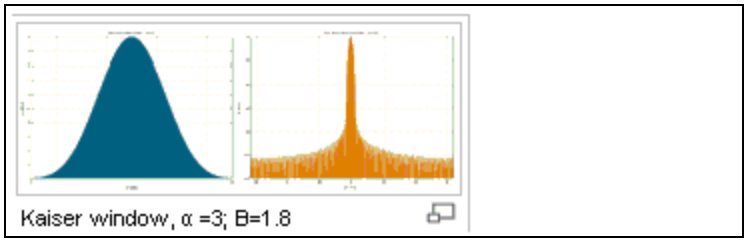
$$w(n) = a_0 - a_1 \cos\left(\frac{2\pi n}{N-1}\right) + a_2 \cos\left(\frac{4\pi n}{N-1}\right)$$

where a_0 =(1-α)/2; α_1=1/2; α_2=α/2

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Window Function	Preferred Use
	
Hamming	A moderate dynamic range function, designed for narrow band applications. It minimizes the maximum sidelobe. $w(n) = 0.54 - 0.46\cos\left(\frac{2\pi n}{N-1}\right)$ 
Hanning (default)	A moderate dynamic range function, designed for narrow band applications. $w(n) = 0.5\left(1 - \cos\frac{2\pi n}{N-1}\right)$ 
Kaiser	Selecting the Kaiser plot also enables a field to specify an associated Kaiser parameter. The larger the Kaiser parameter, the wider the window. The parameter controls the trade off between width of the central lobe and the area of the side lobes. 

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Window Function	Preferred Use
	
Welch	This approach applies a parabola-shaped window to the frequency domain data. It is based on the Bartlett method but splits the signal into overlapping segments, which are then windowed. The intent is to balance the influence of data in the center of the function.

You can use the **Save as Default** to set the current values as a default.

Spectral Domain Plots in Transient (FFT)

When plotting in the spectral domain, you must specify the parameters for the Fast Fourier Transform (FFT). The FFT function used by Maxwell uses windowing; therefore, the user must enter a start and stop time within which the data is periodic.

To enter the FFT parameters:

1. In the **Report** window, be sure the **Domain** is set to **Spectral**.
2. In the **Context** section of the reporter dialog box, click **FFT Options**.

   The **FFT Options** dialog box appears.
3. Enter the **Start Time** for the window using the drop-down menu.
4. Enter the **End Time** using the drop-down menu.
5. Enter the **Window Type** using the drop-down menu.

The samples and number of points are calculated based on the chosen start and end values. The signal must be periodic during the time interval chosen or an error will result.

Time Domain Plots in AC Magnetic (IFFT)

When creating time domain plots from frequency domain data, you must specify the parameters for the Inverse Fast Fourier Transform (IFFT):

1. In the **Report** window, be sure the **Domain** is set to **Time**.
2. In the **Context** section of the reporter dialog box, click **IFFT Options**.

   The **IFFT Options** dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In the **Input Signal** section of the dialog select either **Impulse** or **Step** signal to use in performing the IFFT.
4. For **Step** signal, input the rise time for the pulse. The rise time should chosen so that the frequency content is within the bandwidth of the available frequency sweep data.
5. Select the **Window Type** from the drop-down menu.
6. Enter the window **Width** in percent.

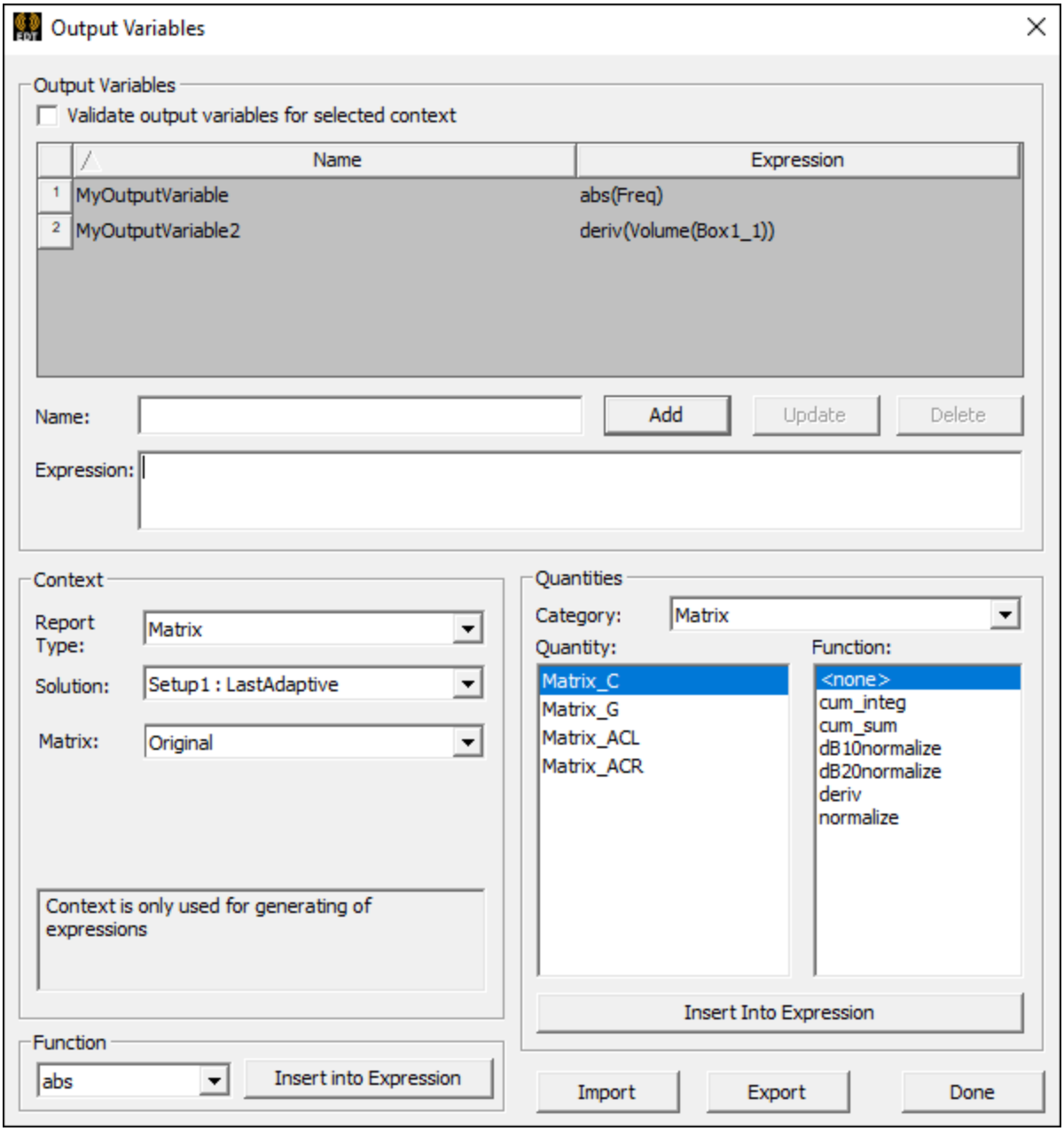
The **Maximum Plot Time** and the **Delta Time** are calculated based on the solution data available.

Specifying Output Variables

Access the **Output Variables** window one of several ways:

- Click **Maxwell** > **Results** > **Output Variables**.
- In the Project Manager, right-click **Results** and select **Output Variables** from the shortcut menu.
- In the **Solution Setup** dialog box, select the **Expression Cache** tab, click **Add...** to display the **Add to expression cache** dialog box, and click the **Output Variables** button.
- Click the **Output Variables** button in the **Report** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Output Variables** window contains four sections:

- **Output Variables** – contains a list of existing output variables, and allows you to specify the name and expression for a new output variable. The **Validate output variable...** check box allows you to validate an expression before adding it. Valid expressions appear in blue and invalid expressions appear in red.
- **Context** – specifies the Report Type and Solution. Depending on the type of report selected, additional fields may appear. All selections affect the functions and quantities

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

listed.

> **Note:**
>
> The context is used only for generating expressions.

- **Quantities** – allows you to insert quantities into the Expression area of the **Output Variables** section.
- **Function** – allows you to insert completed expressions into the Expression area of the **Output Variables** section.

To add an output variable:

1. Enter a **Name** for the variable.
2. Specify the **Context**. This impacts the available quantities and functions.
3. Specify the **Quantities** (required) and **Functions** (optional) using the **Insert Into Expression** buttons.
4. Review the **Expression**. When it is ready, click **Add**.

   The new output variable appears in the **Output Variables** list.

To delete an output variable:

1. Remove all references to the output variable from the project.
2. Save the project to erase the command history.
3. Open the **Output Variables** window using one of the methods above.
4. Select the variable from the **Output Variables** list and click **Delete**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The evaluated value of an expression is always interpreted in SI units. However, when an angle quantity is plotted in a report, you have the option to plot values in units other than SI. For example, if you want to plot the polar angle of a complex simulation result S_{11}, you can choose between ang_deg(S_{11}) and ang_rad(S_{11}). Both of these return the exact same angle quantity, but in degree and radian units respectively.

When using non-SI units in expressions, surprising outcomes might result. For example, the expression 1+ang_deg(S_{11}) represents an angle and the number 1 is treated as 1 rad. The angle SI unit is attached to any unitless number that is added/subtracted from an angle value. If you want to treat 1 as degrees, make it explicit and use 1deg + ang_deg(S_{11}) instead.

If you are interested in unitless degree values, two additional functions exist: ang_deg_val(S_{11}) and cang_deg_val(S_{11}). These return simple numbers and are treated as such by any expression. For example, if the complex S_{11} lies on the positive Y axis, ang_deg_val(S_{11}) would be 90 and 1 + ang_deg_val(S_{11}) would be 91.

Building an Expression Using Existing Quantities

When you are entering an expression for a new output variable, you can insert part or all of the expression using the options in the **Calculation** and **Function** sections of the **Output Variables** window.

To add an input variable by inserting part or all of the expression:

1. Click **Maxwell 3D, Maxwell 2D,** or **RMxprt** and then select **Results** > **Output Variables** or, in the Project Tree, right-click on **Results** and select **Output Variables** from the short-cut menu, or on the **Modify Report** dialog box, click the Output Variables button.

   The **Output Variables** window appears.

2. In the **Output Variables** section, enter a name for the new variable in the **Name** box.
3. To insert a quantity:
   a. From the **Report Type** drop-down menu, select the type of report from which you want to select the quantity.
   b. From the **Solution** drop-down menu, select the solution from which you want to select the quantity.
   c. From the **Category** list, select the type of quantity you want to enter.
   d. From the **Quantity** list, select the quantity or the geometry.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

e. From the **Function** list, select a ready-made function (this option is the same as inserting the function from the **Function** section).
f. If applicable, from the **Domain** list, select the solution domain.
g. Click **Insert Into Expression**.

The selected quantity is entered into the **Expression** area of the **Output Variables** section.

4. To insert a function:
   a. In the **Function** section, select a ready-made function from the drop-down menu.
   b. Click **Insert Function into Expression**.

   The function appears in the **Expression** area of the **Output Variables** section.
5. When you are finished defining the variable in the **Expression** area, click **Add** to add the new variable to the list.
6. Repeat steps 2 through 5 to add additional variables.
7. When you are finished adding output variables, click **Done** to close the **Output Variables** window.

> **Note:** Remember the evaluated value of an expression is always interpreted as in SI units. However, when a quantity is plotted in a report, you have the option to plot values in units other than SI.

Function List for Output Variables

The **Output Variables** window includes a second function list containing functions to enter directly into the Expression field. These functions can also be applied to previously specified Quantities and Functions.

Some of these functions can operate along an entire curve. These are: min, max, integ, avg, rms, pk2pk, cang_deg and cang_rad.

You can select from the functions in the **Output Variables** dialog box's **Function** list or type them directly into the Expression field, if necessary.

Defining Settings for Output Variables for Transient Solutions

Any output quantity or expression within either the **Transient Report** or **Field Report** type can be defined as an output variable in the **Output Variables** dialog box, accessed via **Results** in the Project Manager tree.

To add an output variable to output with time:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click **Add** to select the output variables to be added to the list for output with time. The newly defined variable is added to the list.

> **Note:** The **Add** button will not be enabled if there are no variables defined in the project.

To delete an output variable:

- Select the variable from the list, and click **Delete**.

Do one of the following in the **Evaluation Timestep** section on the **Output Variables** tab of the **Solve Setup** window:

- Select **Using Constant**, type a value, and select the units. Select this option if you want to use a constant value for when you calculate output variable values.
- Select **Evaluate every Nth timestep**, and type a value. Select this option if you want to evaluate the output variable after a certain number of solution timesteps, that is, instead of calculating the output variable every fixed time interval, you want to calculate it every certain number of timesteps. For example, if you want to evaluate the output variable every third timestep, type **3** in this text box.

Report Data

The follow sections describe how you can view, update, export, import, override, and delete various values used for display in plots and reports.

The topics for this section include

- Updating Reports (Post-Processing Data)
- Deleting Reports
- Exporting Plot Data
- Importing 2D Plot Data
- Report File Formats
- Exporting Graphics Files from a Plot
- Plotting Imported Solution Data
- Setting a Range Function

Opening Reports

- To open a single report, right-click the report in the **Results** section of the **Project Manager** window for the design, and click **Open Report** on the menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To open all reports, either select **Maxwell 2D** (or **Maxwell 3D**) > **Results** > **Open All Reports**, or right-click the **Results** icon in the **Project Manager** window for the design, and click **Open All Reports** on the pop-up menu.

Updating Reports (Post-Processing Data)

To manually update the report that is active in the design area using the latest simulation results, right-click in the report and select **Update Report** from the menu. Alternatively, you can right-click on the report under **Results** in the Project Manager and select **Update Report** from the context menu.

If you wish to update all reports in the design, select **Maxwell 2D** (or **Maxwell 3D**) > **Results** > **Update All Reports**. Alternatively, right-click on the **Results** icon in the Project Manager and select **Update All Reports** from the menu.

Deleting Reports

To use **Delete All Reports**:

1. Click **Maxwell** > **Results** > **Delete All Reports.** You can also right-click on the Results folder in the Project Manager, to display the shortcut menu, and click **Delete All Reports.**

   All items under the Results folder in the Project Manager are removed.

To use **Delete** for a selected report:

1. Right-click a report icon in the Project Manager to display the shortcut menu.
2. Click **Delete** to remove this report.

> **Warning:**
>
> Solution data that has been deleted cannot be recovered!

Exporting Reports

You can export reports in a variety of text and graphic formats. You must have an existing plot open to see the corresponding Report2D or Report3D menu.

1. Click **Report2D** or **Report3D** on the top menu, and then select **Export**. Alternatively, right-click the plot, and then click **Export ...** on the shortcut menu.

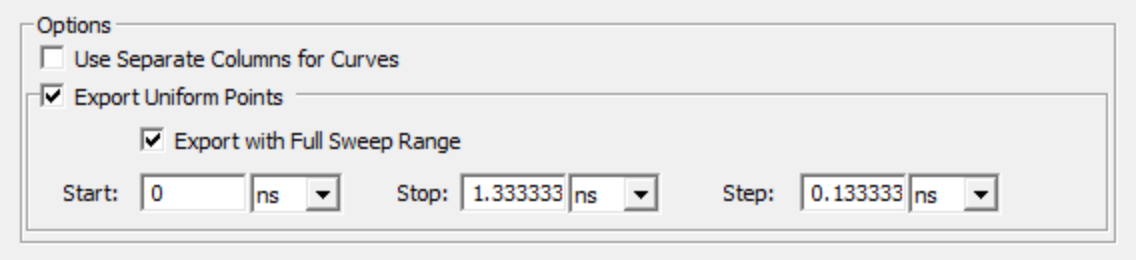
   The **Export Report** dialog box opens.

2. Use the file browser to find the directory where you want to save the file.
3. Type the name of the file in the **File name** box.
4. Select one of the following file formats from the **Save as type** drop-down menu. The options are slightly different for 2D reports and 3D reports, as noted:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- ***.csv** – Comma delimited data files.
- ***.tab** – Tab delimited data files.
- ***.dat** – Ansoft PlotData files (2D reports only).
- ***.txt** – Post processor format files (2D reports only).
- ***.exy** – Ansys EnSight XY date files. (2D reports only).
- ***.case** – Ansys case files. (3D reports only).
- ***.rdat** – Ansoft Report Data format files.
- ***.emf** – Microsoft EMF files.
- ***.gif** – Graphics Interchange Format files.
- ***.bmp** – Bitmap files.
- ***.wrl** – Virtual Reality Modeling Language (VRML) files.
- ***.tif, *.tiff** – Tagged Image File Format files.
- ***.jpg, *.jpeg** – Joint Photographics Experts Group files.

5. Optionally, for .csv and .tab files, you can check, Use Separate Columns for curves.
6. Optionally, for 2D reports, you can choose to Export Uniform Points, which enables fields in which you can specify the start, stop, and step values and units to output uniformly spaced point data.



7. Click **Save** to export the report to the specified location in the selected data or graphical format.

Importing 2D Plot Data

To import 2D plot data into a report:

1. Click **Report2D** on the top menu, and select **Import Data**.

   The **Open** dialog box appears.

2. Browse to the directory where the data file is located.
3. Select the file format from the **Files of type** drop-down:
   - ***.csv** – Comma delimited data files.
   - ***.tab** – Tab delimited data files.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- ***.dat** – Ansoft PlotData files.
- ***.txt** – Post processor format files.
- ***.rdat** – Ansoft Report Data files.

For more information, see Report File Formats.

1. Select the file.
2. Click **Open** to import and plot the data, or **Cancel** to close the dialog box without importing any data.

> **Note:**
>
> For a report trace where the primary sweep is NOT the same as the x-component, an export report then import report may not produce the same curve. In this circumstance, only *.rdat import the same curve/trace. For other formats, the import produces two separated traces.

Report File Formats

The following table provides information about the file formats that can be imported into plot data reports:

File Extension	Information
csv	• Uses a comma (,) as the separation character. • The x-axis value is in the first column. Each curve's Y-values make up one column, and the curves are in the same order as in the plot legend. • The first row is the X-axis name [unit] and the information for each curve, which is the same as the plot curve legend. • Other rows are the X-axis value and the Y-value for each curve.
tab	• Uses the tab character as the separation character. • The x-axis value is in the first column. Each curve's Y-values make up one column, and the curves are in the same order as in the plot legend. • The first row is the X-axis name [unit] and the information for each curve, which is the same as the plot curve legend. • Other rows are the X-axis value and the Y-value for each curve.
txt	The txt file must begin with a header that contains the following: • A first line made of 92 equal signs (=) • A second line that begins with the company name and ends with the date in MM/DD/YY format

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File Extension	Information
	• A third line that begins with a plot name and ends with the time in hh:mm:ss format • A fourth line, which is empty • A fifth line made of 92 hyphens (-) **Note:** Each header line must be 92 characters in length, except the empty line. • Each column has a fixed width, separated by white space. • The x-axis value is in the first column. Each curve's Y-values make up one column, and the curves are in the same order as in the plot legend. • The first row is the X-axis name [unit] and the trace name [unit] for each curve, which is the same as the plot curve legend. • The second row is an empty column and the variable values for each curve, which is the same as the plot curve legend. • Other rows are the X-axis value and the Y-value for each curve.
dat	DAT is an Ansys-specific format. Ansys recommends that you do not generate files using this format. Files imported with these formats should be exported only from Ansys products.
rdat	RDAT is an Ansys-specific format. Ansys recommends that you do not generate files using this format. Files imported with these formats should be exported only from Ansys products.

Creating Custom Report Templates and Defaults

You can edit properties from any report type and save it as a template or as the default. This can save repeated editing of properties (for example, the company name, or color schemes) when you create other reports. Once you create templates, you can access them from the **Results** > **Report Templates** > menu and the **2D Reports** > **Report Templates** > **Apply Settings** menu.

See Modifying the Background Properties of a Report for a discussion of format changes you can make to any report.

To save an edited report as a template:

1. In the Project Manager tree, right-click on the report name of interest to display the shortcut menu and click **Report Templates** > **Save**. You can also click **Report2D** >

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Report templates > **Save**.

This displays the **Report Save As Report Template** file browser. By default, the directory is your \\AnsysEM\\<*productName*>\userlib\ReportTemplates directory. You can also save to the SysLib directory.

2. Typically, you accept the directory.
3. You must provide a file name, which will be given an .rpt extension.

   It is good practice to give the template a descriptive name, showing both the kind of format you begin with (such as XY Plot or 3D Plot) and apt description of the distinguishing edits (such as for company name, or color scheme). Once saved, this name will appear on the **PersonalLib** menu.

4. The **Save As Type** field currently supports the Ansoft Report Format (.rpt) format.
5. Click **Save** to save the template to the PersonalLib menu.

All .rpt templates in the directory appear on the **Results** > **Report Templates** > **PersonalLib** menu. Selecting a report from the PersonalLib menu opens a report that you can then Modify to add traces or perform other edits. Templates in the SysLib directory appear on the **Report Templates** menu.

To save an edited report as a default:

1. In the Project Manager tree, right-click on the report name of interest to display the shortcut menu and click **Report Templates** > **Save Settings as default**. You can also click **Report2D** > **Report templates** > **Save Settings as default** or **Report3D** > **Save Settings as default**.

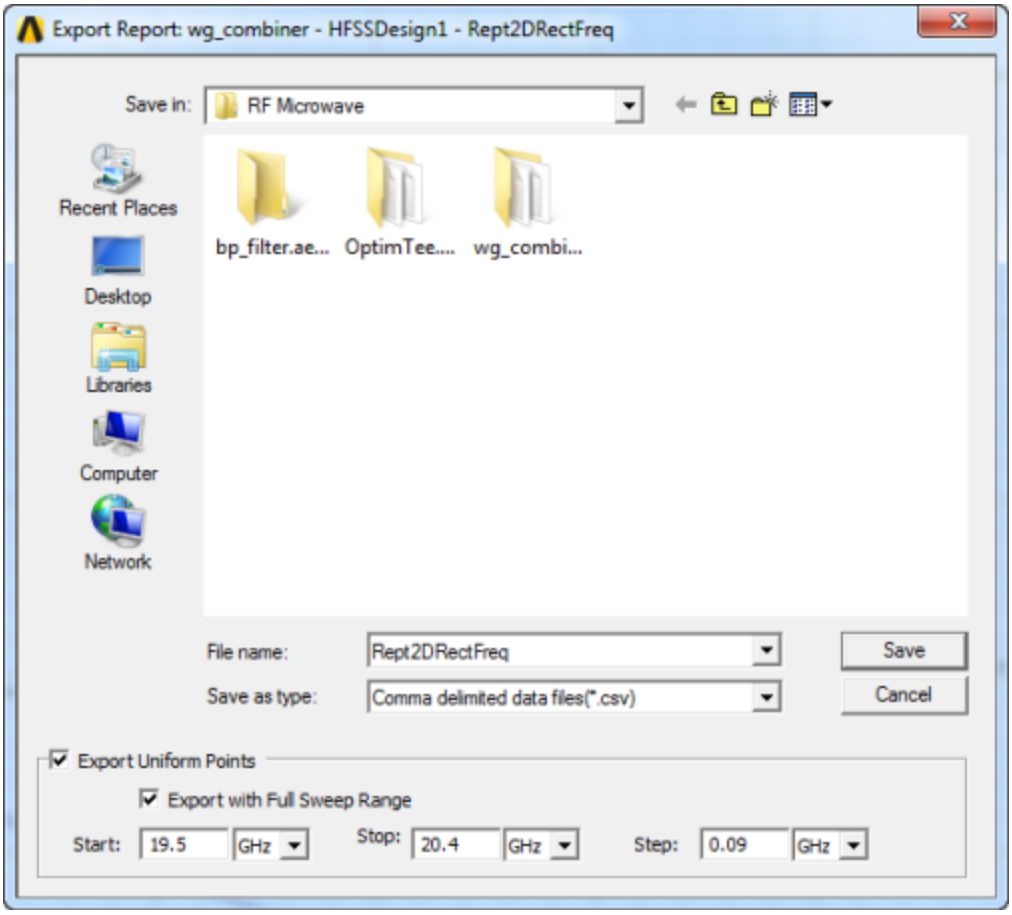
Exporting Reports as Graphics

You can export reports as figures in several formats. You must have an existing plot open to see the **Report2D** or **Report 3D** menu.

1. With a report open, click **Report2D** > **Export...** or **Report3D** > **Export...**

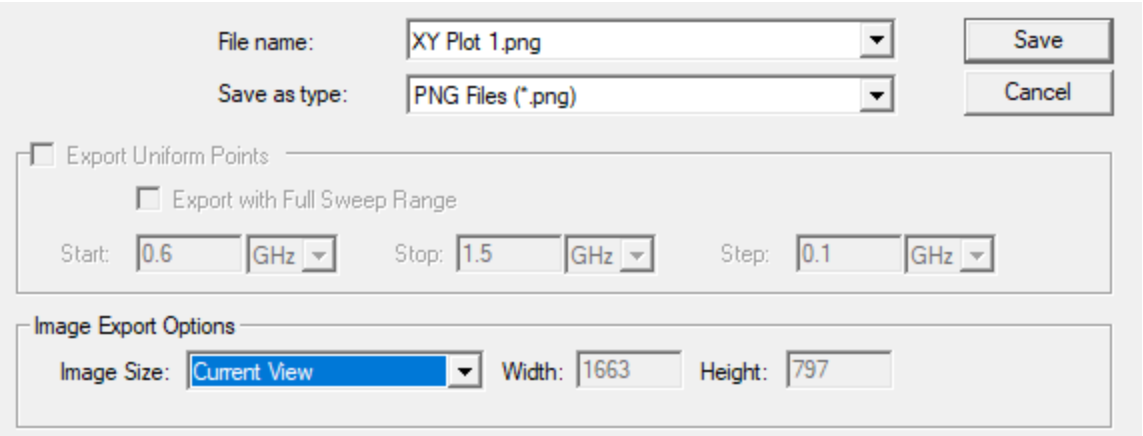
   The **Export Report** dialog box appears:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

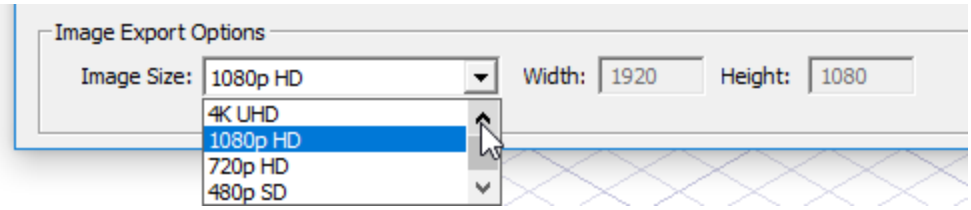


2. Click **Browse...** to open the **Export Report** browser window.
3. Specify the file location and name, and select a graphics format from the drop-down menu:
    - .bmp
    - .gif
    - .jpeg
    - .png
    - .tiff
    - .wrl
4. When you select an image format, the **Image Export Options** appear at the bottom of the dialog box:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Specify the **Image Export Options** by selecting from the drop-down menu. You may need to scroll or click the up or down arrows to see all selections. These include: Current View, User Defined (which enables the Width and Height fields), Full Screen, 8K UHD, 4K UHD, 1080p HD, 720p HD, and 480p SD. The resolution for the current selection is displayed in the Width and Height fields.



6. Click **Save** to close the browser window, and then **OK** to close the **Export** window.

To export an image file of a report to specified high resolution, use the scripting commands. The image can be created at a specified resolution. Fonts and line thickness is not scaled. Only the image. You will have to iteratively increase font sizes until you find a suitable output.

Copying Reports to the Clipboard as Images

You can copy images of reports to the clipboard, so that they can be pasted into other applications as follows:

1. Make the report the active window. Alternatively, for a plot-on-schematic, click on the plot to select it.
2. Click **Edit** > **Copy Image**, or right-click the report window or plot-on-schematic and select **Copy Image** on the context menu to copy the report to the clipboard as an image.
3. Open the application into which you want to paste the image, and paste the image.

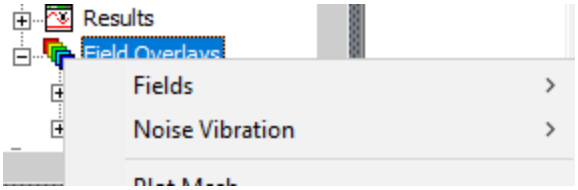
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

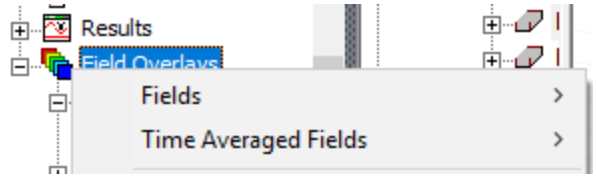
Plotting Field Overlays

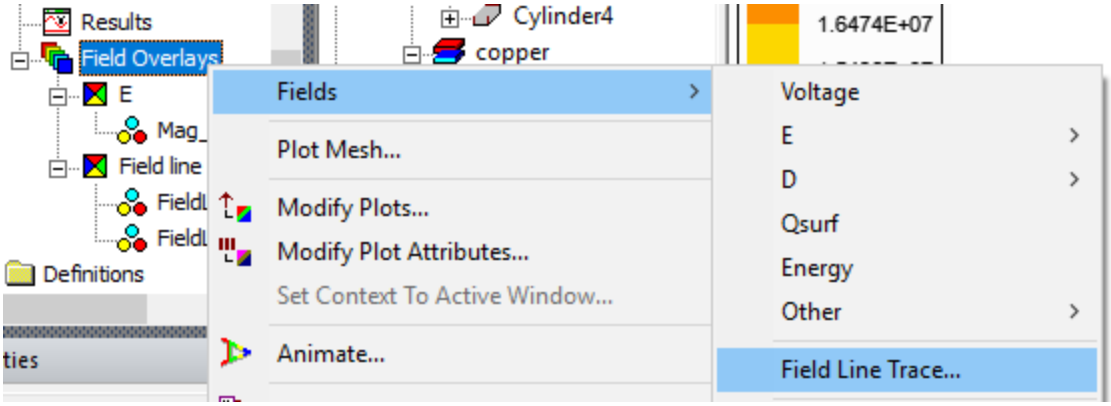
Field overlays are representations of basic or derived field quantities on surfaces or objects for the current design variation. You must select a geometry to create a plot. You can set the design variation via the **Set Design Variation** dialog box. This dialog box is accessible from the **Solution Data** window via by clicking the ellipsis button on the right of the Design Variation field, and via the **Results** > **Apply Solved Variation** command.

You can control the visibility of Field Overlays by means of the **View** > **Visibility** > **Active View Visibility** dialog box, **Fields Reporter** tab, shown above or by right-clicking on the Field plot in the Project tree and checking or unchecking **Plot Visibility**.

There are several types of Field overlays that you can plot:







- **Fields**
- **Time Averaged Fields** (*Transient designs only*)
- **Noise Vibration** *(Transient designs if element-based harmonic forces are set up)*
- **Field Line Trace** (*Electrostatic designs only*)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plotting Standard Fields Overlays

Field overlays are representations of basic or derived field quantities on surfaces or objects for the current design variation. You can set the design variation via the **Set Design Variation** dialog box. This dialog box is accessible from the **Solution Data** window via by clicking the ellipsis button on the right of the Design Variation field, and via the **Results** > **Apply Solved Variation** command.

To plot a standard field quantity:

1. Select a point, line, surface, or object to create the plot on or within. You may also select a plane or object selection in the history tree. If it does not exist, create it.

   > **Note:** For 2D Designs, a plane selection must be consistent with the drawing plane or an error will result.

2. On the main menu bar, click **Maxwell 3D** or **Maxwell 2D** > **Fields** > **Fields**.
3. On the **Fields** submenu, click the field quantity you want to plot. Available selections depend on the solved solution. (You can also right-click on **Field Overlays** in the Project Manager and select **Fields** > *field_quantity*).

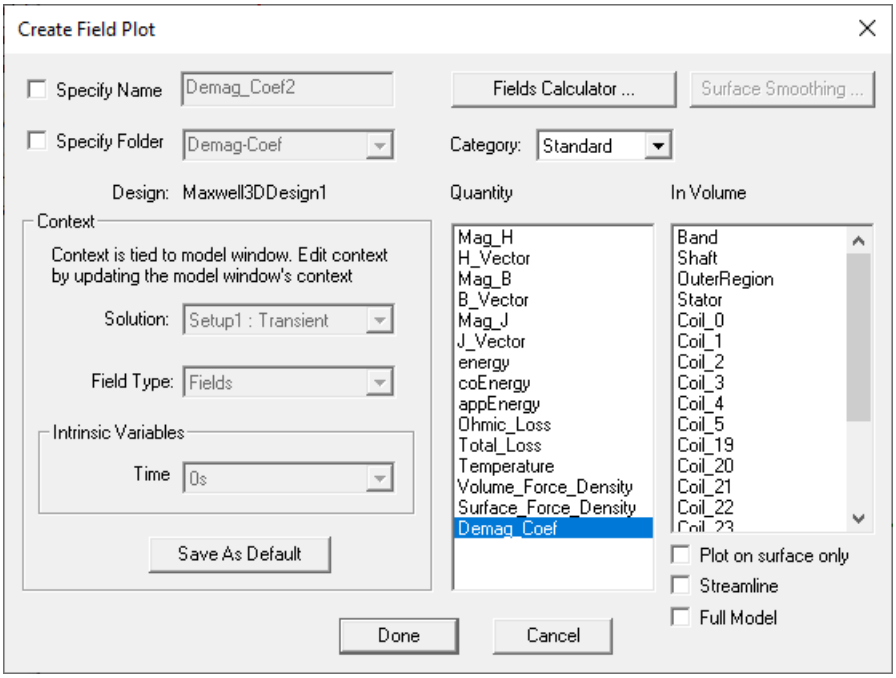
   If you select a scalar field quantity, a scalar surface or volume plot is created. If you select a vector field quantity, a vector surface or volume plot is created. If you select a vector quantity, you will be able to specify a Streamline plot. If the quantity you want to plot is not listed, see *Calculating a Derived Field Quantity*.

   For projects with Temperature dependent materials, the **Maxwell** > **Fields** > **Fields** > **Other...** menus selections include **Temperature**.

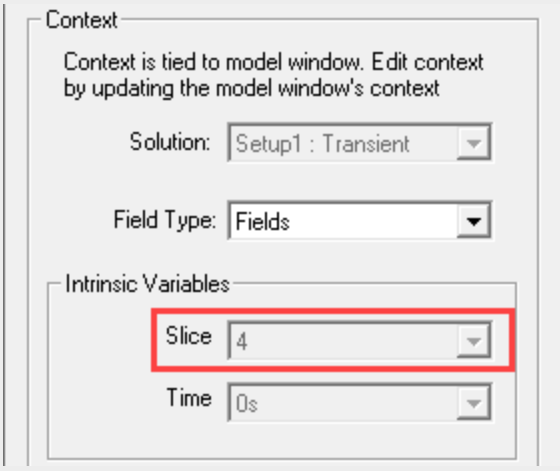
   After you select the field quantity to plot, the **Create Field Plot** dialog box appears.

   The **Specify Name** field shows a name based on the field quantity you selected, and the **Quantity** list shows the field quantity selected.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note: For Maxwell 2D transient solutions in which **Use Skew Model** is enabled on the **Model Settings** tab, an additional **Slice** field displays in the **Intrinsic Variables** panel.



4. To specify a name for the plot other than the default, select **SpecifyName**, and then type a new name in the **Name** box.
5. To specify a folder other than the default in which to store the plot, select **Specify Folder**, and then click a folder in the drop-down menu. Plot folders are listed under **Field Overlays** in the project tree. Plot folders let you group plots with the same quantity

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

together. All field plots under the same folder share the same color key.

> **Note:** All plots (field overlays) in the same folder have the same scale settings. To plot the same field with a different scale, you can create or move the new plot to a separate folder. By default, current density plots are stored in a folder called **J**, but you can specify a different or new folder. Plots in different folders have a different plot keys.

6. Select the solution to plot from the **Solution** drop-down menu.
7. Select the **Fields** type to plot in the **Field Type** drop-down menu.
8. Under **Intrinsic Variables**, select the variable value(s) at which the field quantity is evaluated. Variables available vary depending on the solution type.

   > **Note:** For Maxwell 2D transient solutions in which **Use Skew Model** is enabled on the **Model Settings** tab, a **Slice** field displays in the **Intrinsic Variables** panel showing the slice currently selected for viewing in the modeler window.

9. If desired, you can choose a different field quantity to plot from the **Quantity** list. For scalar quantities plotted on the surface of a geometry, the **Surface Smoothing** button is enabled.

   > **Note:** If you selected an *object*, you must check **Plot on surface only** to enable the **Surface Smoothing** button.

   For vector quantities, the **Surface Smoothing** button is disabled. For details on using the **Surface Smoothing** button, see Modifying Field Plots.

10. Select the volume or surface (region) in which to plot the field from the **In Volume** list.

    This selection enables you to limit plots to the intersection of a volume with the selected object or objects. You can select and deselect any items in the **In Volume** list. You can mix model objects with non-model boxes. For example, you might want to see a plot from part of two model objects by restricting the region to a non-model box overlapping those parts.

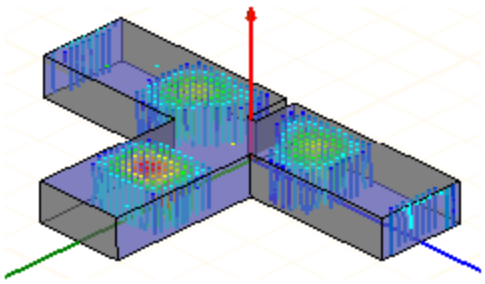
    > **Note:** Multiple selection should be used when there is a discontinuous field on a surface. If not, the field on both sides of the surface is plotted and each interferes with the other.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** Non-convex 3D non-model solids are not supported for Field Overlay Plots and Fields Calculator computations.

> **Note:** Object selections containing non-model object(s) are supported for Field Overlay Plots, but are not supported for Fields Calculator computations.

11. If you selected a vector quantity, you can use the check box to select **Streamline** plot. Streamlines are often used to indicate magnetic flux lines, *etc.* in plots.



> **Note:**
>
> - Before creating the plot, select the starting edge (in 2D), starting surface (in 3D), or the starting points for both 2D and 3D.
> - In the **Create Field Plot** dialog box, select **In Volume:** region, which is the region in which the streamlines will appear. Streamlines are outside of objects that are excited (sources).
> - To show more streamlines after the plot is created, on the **Modify attributes...** dialog **Plots** tab, reduce the **Seeds density** value. If no streamlines appear, reduce this by a factor of 10 (or 100) because the default seeding was too large. Refer to Setting Field Plot Attributes for adjusting the Streamline display parameters, and Setting Fields Reporter Options for setting Streamline defaults.
> - For exporting Streamline plots, refer to Exporting Field Plots.

12. *Optionally*, you can select the **Plot on surface only** check box to obtain a plot around the surfaces of selected objects.
13. *Optionally*, for model objects in partial model designs, click the **Full Model** check box to calculate the fields on the full model based on the partial model solution.
14. *Optionally*, click **Fields Calculator** to open the fields calculator in which you can create named expressions, which can then be selected in the Create Field Plot dialog by

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

choosing **Calculator** as the Category. The named expression(s) appear in the **Quantity** list.

15. Click **Done**.

The field quantity is plotted on the surfaces or within the objects you selected. The plot uses the attributes specified in the **Plot Attributes** dialog box.

The new plot appears in the view window. It is listed in the specified plot folder in the project tree. If you have created a field plot on a simulation in progress, the field plot is updated after the last adaptive solution.

If you want to update the field overlay before then, to view progress in the solution, select the Field icon in the Project tree that contains the field plot of interest, right-click to display the short cut menu, and select **Update Plots**.

To turn off the display of the plot, right-click on the plot and select **Plot Visibility** from the short-cut menu. Unchecking **Plot Visibility** turns off the plot display.

Plotting Standard Field Quantities

Maxwell makes numerous quantities available for field plotting derived directly from the field solution. The quantities available depend upon the solution type (magnetostatic, AC magnetic, *etc.*), on the design type (2D or 3D) and other parameters such as the use of lossy materials. The chart below lists possible solution quantities available when using **Maxwell 2D** > **Fields** > **Fields** or **Maxwell 3D** > **Fields** > **Fields**.

Description	Quantity	2D	3D
Vector Potential	**Flux_Lines, A_Vector**	Magnetostatic, AC Magnetic, Transient	
Magnetic Field Intensity	**Mag_H, ComplexMag_H, H_ Vector**	Magnetostatic, AC Magnetic, Transient	Magnetostatic, AC Magnetic, Transient
Magnetic Flux Density	**Mag_B, ComplexMag_B,B_ Vector**	Magnetostatic, AC Magnetic, Transient	Magnetostatic, AC Magnetic, Transient
Current Density	**JatPhase, Mag_J, ComplexMag_J, J_ Vector, Jphi, Jz**	Magnetostatic, AC Magnetic, Transient, ACConduction, DCConduction	Magnetostatic, AC Magnetic, Transient, DCConduction, Electric Transient, ACConduction
Direct Current Density	**Mag_Jc, ComplexMag_Jc, Jc_ Vector**		ACConduction
Electric Flux	**DatPhase, Mag_D,**	AC Magnetic,	AC Magnetic,

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Quantity	2D	3D
Density	**ComplexMag_D, D_ Vector**	Electrostatic, ACConduction	Electrostatic, DCConduction, Electric Transient, ACConduction
Electric Field Intensity	**Mag_E, ComplexMag_E, E_ Vector**	Electrostatic, ACConduction, DCConduction	AC Magnetic, Electrostatic, DCConduction, Electric Transient
Surface Charge Density	**QSurf**		Electrostatic, ACConduction, DCConduction, Electric Transient
Surface Current Density	**Jsurf**		AC Magnetic
(on an impedance boundary)			
Voltage	**Voltage**	Electrostatic, ACConduction, DCConduction	Electrostatic, DCConduction, Electric Transient, ACConduction, Transient A-Phi
Energy in the Field **Electric** **Magnetic**	**energy**	Magnetostatic, AC Magnetic, Transient, Electrostatic	Magnetostatic, AC Magnetic, Transient, Electrostatic, ACConduction, DCConduction, Electric Transient
Magnetic CoEnergy	**coEnergy**	Magnetostatic, AC Magnetic, Transient	Magnetostatic, Transient
	appEnergy	Magnetostatic, AC Magnetic, Transient	Magnetostatic, Transient
Ohmic Loss	**Ohmic_Loss**	Magnetostatic, AC Magnetic, Transient, Electrostatic, ACConduction, DCConduction	Magnetostatic, AC Magnetic, Transient, DCConduction, Electric Transient, ACConduction
Total Loss	**Total_Loss**	Transient (Magnetic)	
Hysteresis Loss	**Hysteresis_Loss**	AC Magnetic	AC Magnetic
Dielectric Loss	**Dielectric_Loss**	AC Magnetic	AC Magnetic ACConduction

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Quantity	2D	3D
Core Loss	Core_Loss	AC Magnetic	AC Magnetic
EM Loss	EM_Loss	AC Magnetic	AC Magnetic
Surface Loss Density	**SurfaceLossDensity**		AC Magnetic
Edge Force Density	**edgeForceDensity**	Magnetostatic, AC Magnetic (DC component only), Transient, Electrostatic	
Temperature	**Temp**	Magnetostatic, AC Magnetic, Electrostatic	Magnetostatic, AC Magnetic, Electrostatic
Surface Force Density	**surfaceForceDensity**	Magnetostatic, AC Magnetic (DC component only), Transient, Electrostatic	Magnetostatic, AC Magnetic (DC component only), Transient, Electrostatic, Electric Transient
Volume Force Density	**volumeForceDensity**		Magnetostatic, AC Magnetic, Transient, Electrostatic, Electric Transient
Displacement	**Displacement_ Vector, Mag_ Displacement**	Magnetostatic, AC Magnetic, Electrostatic	Magnetostatic, AC Magnetic, Electrostatic
Sum of Ohmic, Hysteresis, and Dielectric Loss	**emloss**	AC Magnetic	
Loss on Impedance boundary applied to a conductor	**edgeLossDensity**	AC Magnetic	
Demagnetization coefficient	**Demag_coef**	Magnetostatic, Transient	Magnetostatic, Transient
Loss Density	**LossDensity**	Transient	Transient

Force Density Quantities

Edge Force	The edge density is available only in Maxwell 2D. It is available only on object outlines adjacent to air or vacuum space. It is defined as

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$f = \overline{S} \cdot n$$

$$\overline{S} = \frac{1}{\mu_0}\begin{bmatrix} B_x^2 - B^2/2 & B_x B_y \\ B_x B_y & B_y^2 - B^2/2 \end{bmatrix} + \varepsilon_0 \begin{bmatrix} E_x^2 - E^2/2 & E_x E_y \\ E_x E_y & E_y^2 - E^2/2 \end{bmatrix}$$

Density

where *B* and *E* are the magnetic flux density and electric field intensity at the midpoint of the edge, respectively, and *n* is the outward normal direction of the edge. μ and ε are the permeability and permittivity in the ith neighboring element. For the electrostatic, AC, and DC solvers, only the electric field is included; while for the magnetostatic, AC Magnetic, and magnetic transient solvers, only the magnetic field is considered. For the AC and AC Magnetic solvers, edge force density is the averaged (DC) force density.

In Maxwell 2D, surface force density is defined as

$$f = \frac{\sum_{i=0}^{i=N} \overline{S}_i \cdot n_i l_i r_i}{AreaOfElement * r_c}$$

$$\overline{S}_i = \frac{1}{\mu}\begin{bmatrix} B_x^2 - B^2/2 & B_x B_y \\ B_x B_y & B_y^2 - B^2/2 \end{bmatrix} + \varepsilon \begin{bmatrix} E_x^2 - E^2/2 & E_x E_y \\ E_x E_y & E_y^2 - E^2/2 \end{bmatrix}$$

where *N* is the number of sides of the element, l_i is the length of side *i* of the element. For the RZ case, r_i and r_c are the radius of side *i* and the radius of the element center, respectively. μ and ε are the permeability and permittivity in the ith neighboring element. In the XY plane, r_i and r_c are set to 1.

Surface Force Density

In Maxwell 3D, surface force density is only available on object faces adjacent to air or vacuum space. It is defined as

$$f = \overline{S} \cdot n$$

$$\overline{S} = \frac{1}{\mu_0}\begin{bmatrix} B_x^2 - B^2/2 & B_x B_y & B_x B_z \\ B_x B_y & B_y^2 - B^2/2 & B_y B_z \\ B_x B_z & B_y B_z & B_z^2 - B^2/2 \end{bmatrix} + \varepsilon_0 \begin{bmatrix} E_x^2 - E^2/2 & E_x E_y & E_x E_z \\ E_x E_y & E_y^2 - E^2/2 & E_y E_z \\ E_x E & E_y E_z & E_z^2 - E^2/2 \end{bmatrix}$$

where *B* and *E* are the magnetic flux density and electric field intensity at the center of the surface face, respectively, and *n* is the outward normal direction of the surface face. For the electrostatic, AC, and DC solvers, only the electric field is included; while for the magnetostatic, AC Magnetic, and magnetic transient solvers, only the magnetic field is considered. For the AC and AC Magnetic solvers, surface force density is the averaged (DC) force density.

Volume Force Density

Volume force density is only available in Maxwell 3D. It is calculated as

$$f = \frac{\sum_{i=0}^{i=N} \overline{S}_i \cdot n_i A_i}{VolumeOfElement}$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\overline{S}_i = \frac{1}{\mu}\begin{bmatrix} B_x^2 - B^2/2 & B_xB_y & B_xB_z \\ B_xB_y & B_y^2 - B^2/2 & B_yB_z \\ B_xB_z & B_yB_z & B_z^2 - B^2/2 \end{bmatrix} + \varepsilon\begin{bmatrix} E_x^2 - E^2/2 & E_xE_y & E_xE_z \\ E_xE_y & E_y^2 - E^2/2 & E_yE_z \\ E_xE & E_yE_z & E_z^2 - E^2/2 \end{bmatrix}$$

μ and ε are the permeability and permittivity in the ith neighboring element. A_i is area of side *i* of element. For the electrostatic, AC, and DC solvers, only the electric field is included; while for the magnetostatic, AC Magnetic, and magnetic transient solvers, only the magnetic field is considered. For the AC and AC Magnetic solvers, volume force density is the averaged (DC) force density.

Lorentz force density is calculated instead of volume force density in pure conductors containing source current in magnetostatic, AC Magnetic, and magnetic transient solvers: *f = J x B.*

Loss Quantities

Ohmic loss	Ohmic loss is always associated with conduction current distribution in conductors that are not perfect. Thus, the resistivity of conductors is responsible for the ohmic power loss when current flows in such conductors. It is also called the Joule-Lenz effect. There is always a heating effect due to the ohmic loss, often called Joule heating.
Total loss	Total loss is associated with loss density fields in 2D and 3D magnetic transient solutions only. Total loss uses the total integrated loss output from the solver and averages it, using each solver timestep between the last two consecutive save-field times. It is useful for passing loss density information from Maxwell to the Ansys thermal solvers with fewer save-field times. For Litz Wire only, total loss also includes StrandedLossAC. Unlike most other loss quantities, which are instantaneous values, total loss is a running average over time, *i.e.* at time T1 it is calculated with fields from T1 to T2.
Hysteresis loss	Hysteresis loss is associated with loss density fields in 2D and 3D AC Magnetic solutions only. Hysteresis loss is short for magnetic hysteresis loss and represents power loss in some magnetic materials (electric steels or ferrites) in alternating (sinusoidal) magnetic fields. This loss is due to a phenomenon called "magnetic viscosity" which causes the B and H fields to have a phase shift between them. In the B-H plane, for linear materials, the relationship between these two fields describes an ellipse. The hysteresis loss is proportional to the area of the ellipse.
Dielectric loss	Dielectric loss is associated with loss density fields in AC Magnetic and Conduction solutions only. Dielectric loss or electric hysteresis loss represents power loss in ferroelectric materials (such as barium titanate, lead

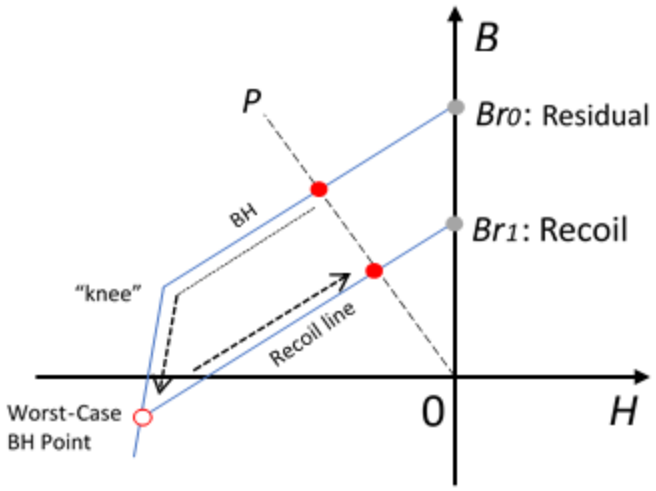
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	zirconate titanate) in alternating (sinusoidal) electric fields. This loss is due to a phenomenon called "electric viscosity" which causes the D and E fields to have a phase shift between them. In the D-E plane, for linear materials, the relationship between these two fields describes an ellipse. The dielectric loss is proportional to the area of the ellipse.
SurfaceLoss Density	The surface loss density is associated with the induced eddy current on the impedance boundary.
EM loss	In 2D and 3D AC Magnetic solvers only: EM_Loss combines the ohmic loss, dielectric loss, hysteresis loss, and StrandedLossAC (for Litz Wire only).
Edgeloss density	Edge loss density represents losses associated with impedance boundary applied on conductive surfaces on the boundary of the solution domain or on surface of excluded objects. It assumes an exponential spatial decay (toward the interior of the respective material) of induced current magnitudes and of associated losses. Integrating the loss along the edge produces the entire loss associated with the "edge".
Core loss	The core loss combines eddy current losses and hysteresis losses for a transient solution type. It is a postprocessing calculation, based on already calculated transient magnetic field quantities. It is applicable for the evaluation of core losses in steel laminations (frequently used in applications such as electric machines, transformers) or in power ferrites.
Solid Loss	The solid loss represents the resistive loss in a 2D or 3D volume and is calculated by $SolidLoss = \frac{1}{\sigma}\int_{vol} J^2 \cdot dV$ For **2DXY** designs, the length is given by **Set Model Depth**, while for **RZ** designs the volume is based upon the rotation of the cross-section around the symmetry axis.
Stranded Loss (3 types)	There are three types of stranded loss quantities: **StrandedLoss**, **StrandedLossAC**, and **StrandedLossR**, which represent the loss in a 2D or 3D winding utilizing multi-turn coil terminals. • **StrandedLoss** is dependent on the object's material conductivity of a stranded source. It represents the loss due to the DC resistance only and uses the cross-sectional area of the object that is assigned with the excitation. Where ***J*** is the current density, it is calculated by $StrandedLoss = \frac{1}{\sigma}\int_{vol} J^2 \cdot dV$ ◦ When Litz wire *is not defined*, StrandedLoss is computed by assuming fill factor = 100% (the ratio of net conductor area to coil terminal area). ◦ When Litz wire *is defined*, StrandedLoss is computed by using

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	the actual fill factor based on the specified Litz stranding and represents the DC loss only. ◦ StrandedLoss does not include the losses of end-coil resistance in 2D. ◦ StrandedLoss can be used to scale the loss density to a coupled thermal analysis. • **StrandedLossAC** is reported when using Litz wire model, and is dependent on the object's material conductivity of a stranded source. It represents the loss due to the DC resistance as well as AC eddy current losses due to proximity effects and skin effects. ◦ StrandedLossAC = StrandedLoss + AC eddy current losses due to proximity effects and skin effects. ◦ StrandedLossAC does not include the losses of end-coil resistance in 2D. ◦ StrandedLossAC can be used to transfer loss density to a coupled thermal analysis. • **StrandedLossR** represents the loss based on I^2 times the resistance **R**, by using the resistance value entered in the Winding properties for a voltage type winding only (Reported for a stranded voltage source not in an external circuit).
LossDensity	For Time Averaged Fields (for transient 2D and 3D solutions), the loss density is the time averaged loss density between a start time and end time, *i.e.*, the accumulated loss density from start time to end time divided by the time duration.

Demagnetization Coefficient Plots



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

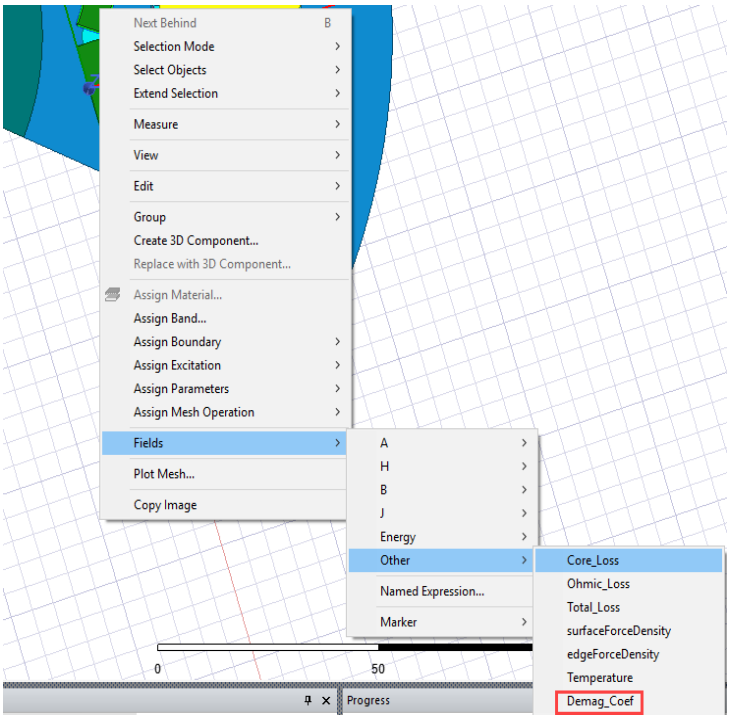
Demagnetization Coefficient: Demag_coef[%] = Br1/Br0*100%

When Demag_Coef = 1, the magnet behaves without any demagnetization at that location.

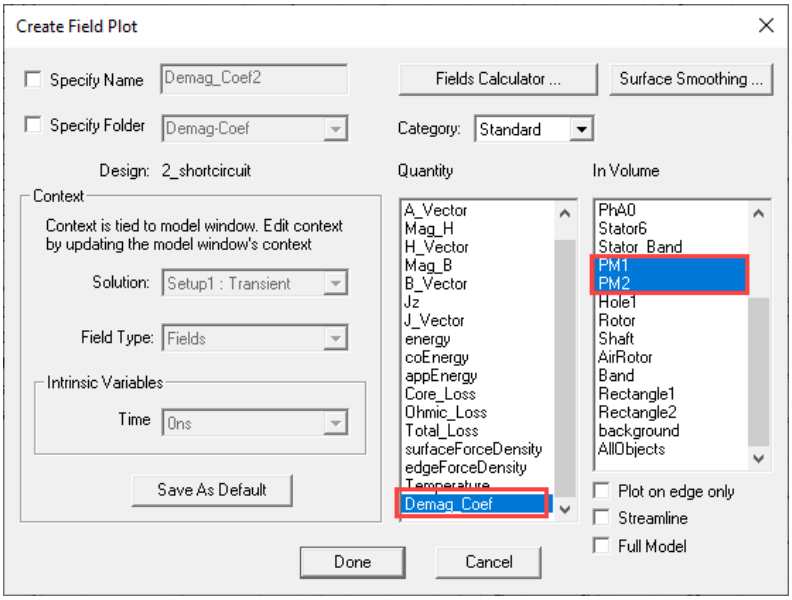
When Demag_Coef = 0, the magnet is fully demagnetized at that location.

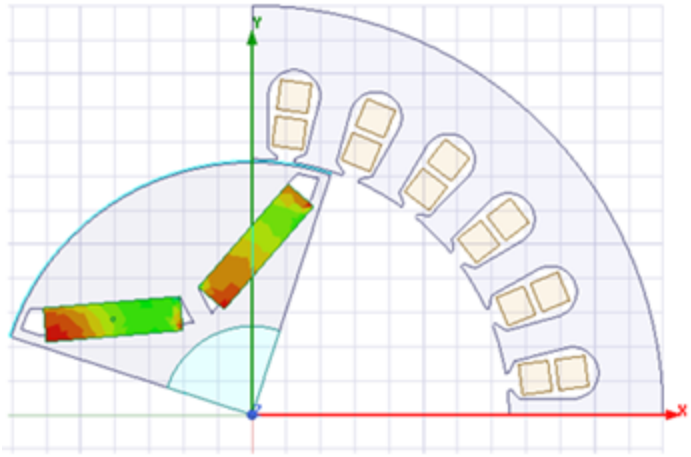
Demag_Coef Plot over the Geometry of Nonlinear Permanent Magnets:

Demag_Coef is a scalar value between 0 and 1, and is displayed as a color shaded plot inside nonlinear permanent magnets. **Demag_Coef** is applicable to 2D/3D transient solutions, and 2D/3D magnetostatic solutions - assign to objects by right-clicking them and selecting **Fields** > **Other** > **Demag_Coef**. Solvers provide the quantity at each node element-by-element to the post-processor for all nonlinear permanent magnetic objects at each timestep with the saved field.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

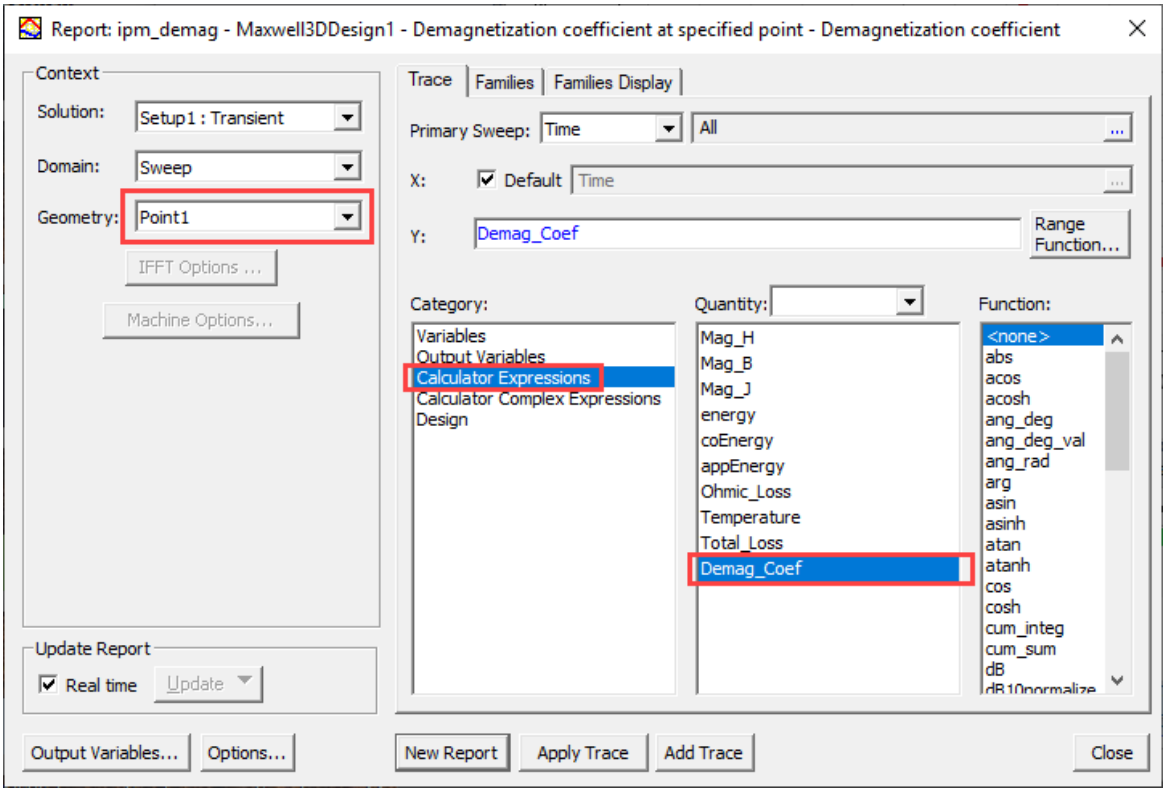


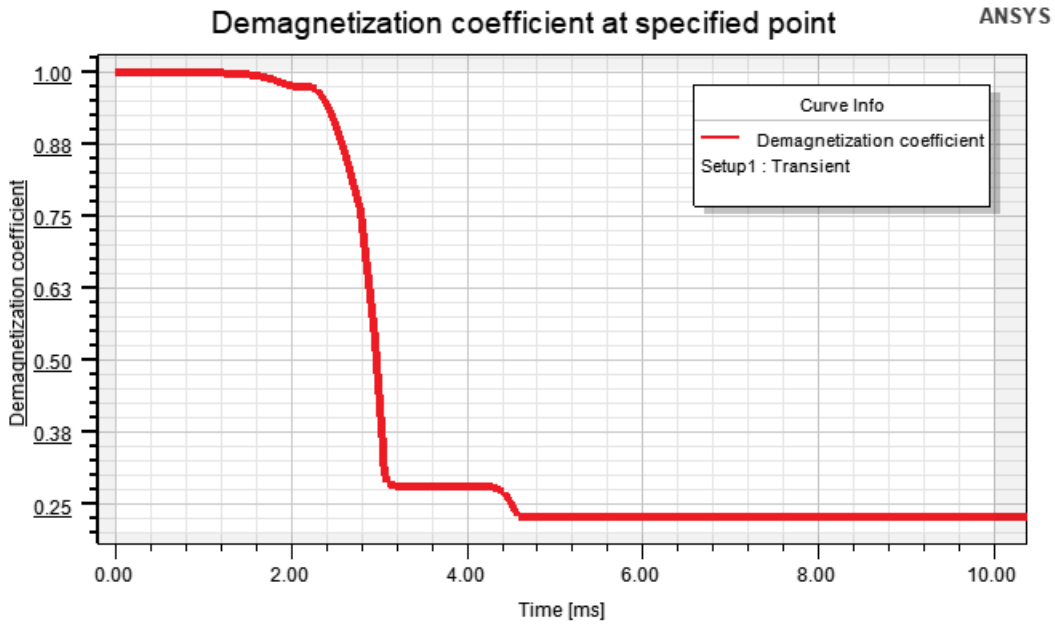


Demag_Coef Curve as a Function of Time at Specific Locations:

This function is provided in terms of the expression cache at any specified point in the magnet. This plot is only applicable to 2D/3D transient solutions.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

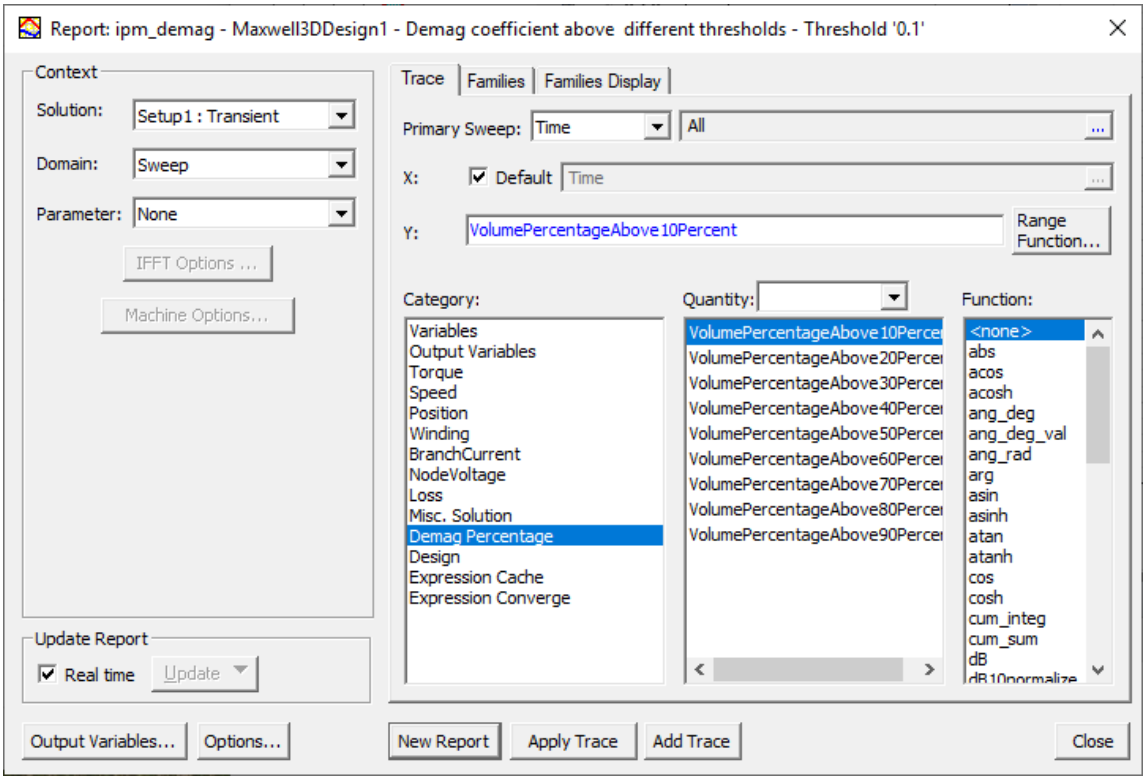




Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

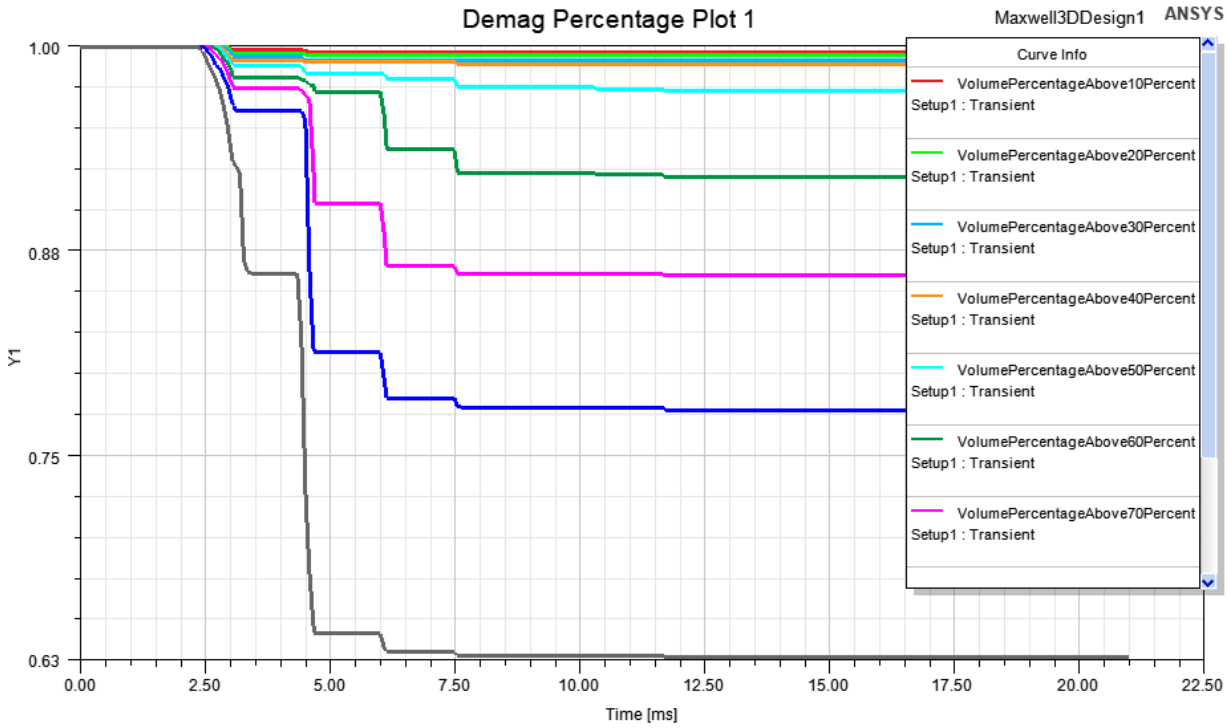
Demagnetization Percentage Statistics Curve as a Function of Time over All Magnet Volumes

For a specified Demag_Coef threshold, determine what the *percentage of total volume* for all magnets is, whose Demag_Coef is equal or greater than the specified Demag_Coef threshold. To determine this, the solver uses the Demag_Coef threshold values from 0.1 to 0.9, with 0.1 as the step interval, to compute the **Demag Percentage** statistics (**VolumePercentageAbove10Percent** through **VolumePercentageAbove90Percent**) at each timestep.



As a result, the output is a series of nine curves from 0.1 to 0.9 similar to the example shown below. This plot is only applicable to 2D/3D transient solutions.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Plotting Time Averaged Fields Overlays

Time Averaged Field overlays are representations of time averaged field quantities, such as LossDensity, on surfaces or objects between specified start and end times divided by the time duration.

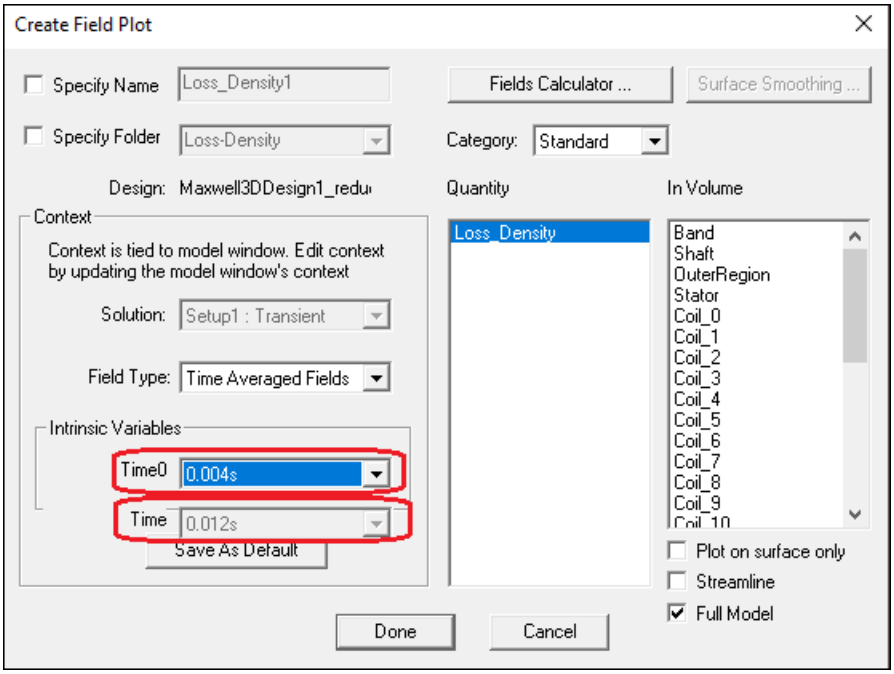
To plot a time averaged field quantity:

1. Select a point, line, surface, or object to create the plot on or within. You may also select a plane or object selection in the **History Tree**.
2. On the main menu bar, click **Maxwell 3D** or **Maxwell 2D** > **Fields** > **Time Averaged Fields** > **Fields** > **Time Averaged Fields**.
3. On the **Time Averaged Fields** sub-menu, click the field quantity you want to plot. Available selections depend on the solved solution. (You can also right-click on **Field Overlays** in the Project Manager and select **Time Averaged Fields** > *field_quantity*).

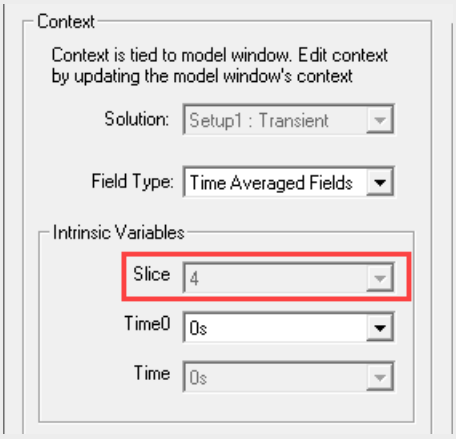
   If you select a scalar field quantity, a scalar surface or volume plot is created. If you select a vector field quantity, a vector surface or volume plot is created. If you select a vector quantity, you will be able to specify a Streamline plot. If the quantity you want to plot is not listed, see *Calculating a Derived Field Quantity*.

   After you select the field quantity to plot, the **Create Field Plot** dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note: For Maxwell 2D transient solutions in which **Use Skew Model is enabled on the Model Settings tab**, an additional **Slice** field displays in the **Intrinsic Variables** panel.



The **Specify Name** field shows a name based on the field quantity you selected, and the **Quantity** list shows the field quantity selected.

4. To specify a name for the plot other than the default, select **Specify Name**, and then type a new name in the box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. To specify a folder other than the default in which to store the plot, select **Specify Folder**, and then click a folder in the drop-down menu. Plot folders are listed under **Field Overlays** in the project tree. Plot folders let you group plots with the same quantity together. All field plots under the same folder share the same color key.

   > **Note:** All plots (field overlays) in the same folder have the same scale settings. To plot the same field with a different scale, you can create or move the new plot to a separate folder. Plots in different folders have a different plot keys.

6. Select the solution to plot from the **Solution** drop-down menu.
7. Select the fields type to plot in the **Field Type** to plot in the drop-down menu. Choices are **Fields** or **Time Averaged Fields**.
8. Under **Intrinsic Variables**, select the variable value(s) at which the field quantity is evaluated. Variables available vary depending on the solution type.

   > **Note:** For Maxwell 2D transient solutions in which **Use Skew Model** is enabled on the **Model Settings** tab, a **Slice** field displays in the **Intrinsic Variables** panel showing the slice currently selected for viewing in the modeler window.

9. If desired, you can choose a different field quantity to plot from the **Quantity** list.

   > **Note:** If you selected an *object*, you must check **Plot on surface only** to enable the **Surface Smoothing** button.

   For vector quantities, the **Surface Smoothing** button is disabled. For details on using the **Surface Smoothing** button, see Modifying Field Plots.

10. Select the volume or surface (region) in which to plot the field from the **In Volume** list.

    This selection enables you to limit plots to the intersection of a volume with the selected object or objects. You can select and deselect any items in the **In Volume** list. You can mix model objects with non-model boxes. For example you might want to see a plot from part of two model objects by restricting the region to a non-model box overlapping those parts.

    > **Note:** Multiple selection should be used when there is a discontinuous field on a surface. If not, the field on both sides of the surface is plotted and each interferes with the other.

11. If you selected a vector quantity, you can use the check box to select **Streamline** plot.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- Before creating the Streamline plot, select both starting and ending edges (in 2D) or both surfaces (in 3D).
- In the **Create Field Plot** dialog box, select **In Volume:** region, which is the region in which the streamlines will appear. Streamlines are outside of objects that are excited (sources).
- To show more streamlines after the plot is created, on the **Modify attributes...** dialog **Plots** tab, reduce the **Seeds density** value. If no streamlines appear, reduce this by a factor of 10 (or 100) because the default seeding was too large. RRefer to Setting Field Plot Attributes for adjusting the Streamline display parameters, and Setting Fields Reporter Options for setting Streamline defaults.
- For exporting Streamline plots, refer to Exporting Field Plots.

12. *Optionally*, you can select the **Plot on surface only** check box to obtain a plot around the surfaces of selected objects.
13. *Optionally*, for model objects in partial model designs, click the **Full Model** check box to calculate the fields on the full model based on the partial model solution.
14. *Optionally*, click **Fields Calculator** to open the fields calculator in which you can create named expressions, which can then be selected in the Create Field Plot dialog by choosing **Calculator** as the Category. The named expression(s) appear in the **Quantity** list.
15. Click **Done**.

    The time averaged field quantity is plotted on the surfaces or within the objects you selected. The plot uses the attributes specified in the **Plot Attributes** dialog box.

    The new plot appears in the view window. It is listed in the specified plot folder in the project tree. If you have created a field plot on a simulation in progress, the field plot is updated after the last adaptive solution.

    If you want to update the field overlay before then, to view progress in the solution, select the Field icon in the Project tree that contains the field plot of interest, right-click to display the short cut menu, and select **Update Plots**.

    To turn off the display of the plot, right-click on the plot and select **Plot Visibility** from the short-cut menu. Unchecking **Plot Visibility** turns off the plot display.

Plotting Noise Vibration Overlays *(Transient designs only)*

Noise vibration field overlays are representations of basic or derived noise vibration field quantities on surfaces or objects for the current design variation. You can set the design

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

variation via the **Set Design Variation** dialog box. This dialog box is accessible from the **Solution Data** window via by clicking the ellipsis button on the right of the Design Variation field, and via the **Results** > **Apply Solved Variation** command.

The **Noise Vibration** field type is available for Maxwell 2D and 3D Transient designs if element-based harmonic forces are set up. The **Noise Vibration** field type handles surface and volume force density in 3D, and surface and edge force density in 2D. **Frequency** and **Phase** intrinsic variables are used instead of **Time**. The **Noise Vibration** field overlays share the same Maxwell solution as the standard Fields overlays.

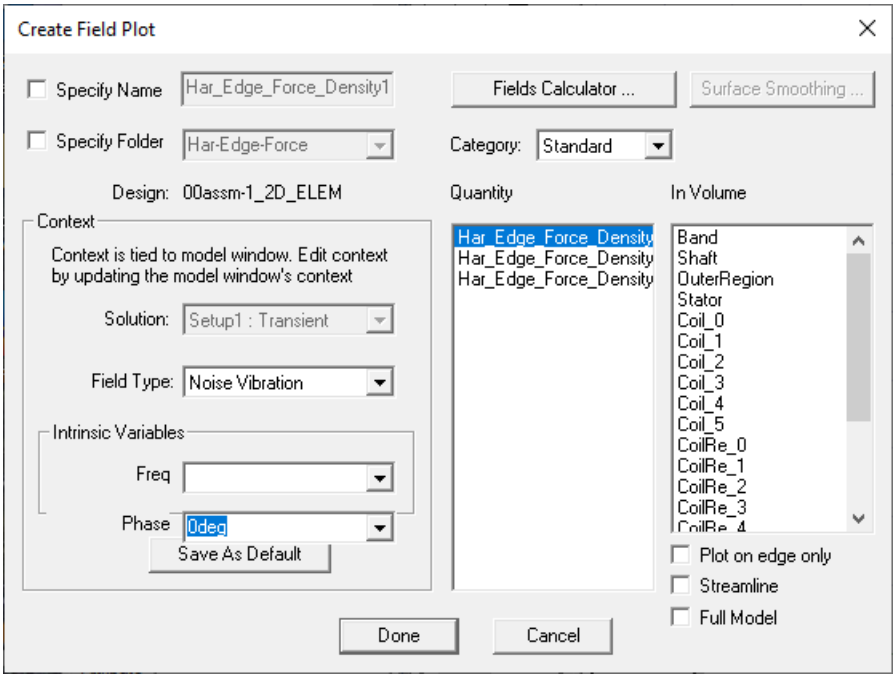
To plot a noise vibration field quantity:

1. Select a surface or volume for a 3D design; or an edge or surface for a 2D design, to create the plot on or within the selected objects.
2. On the main menu bar, click **Maxwell 2D** > **Fields** > **Noise Vibration** or **Maxwell 3D** > **Fields** > **Noise Vibration**.
3. On the **Noise Vibration** sub-menu, click the field quantity you want to plot. (You can also right-click on **Field Overlays** in the Project Manager and select **Fields** > **Noise Vibration** > *field_quantity*).

   If you select a scalar field quantity, a scalar surface or volume plot is created. If you select a vector field quantity, a vector surface or volume plot is created. If you select a vector quantity, you will be able to specify a Streamline plot. If the quantity you want to plot is not listed, see *Calculating a Derived Field Quantity*.

   After you select the field quantity to plot, the **Create Field Plot** dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Specify Name** field shows a name based on the field quantity you selected, and the **Quantity** list shows the field quantity selected.

4. To specify a name for the plot other than the default, select **Specify Name**, and then type a new name in the box.
5. To specify a folder other than the default in which to store the plot, select **Specify Folder**, and then click a folder in the pull-down list. Plot folders are listed under **Field Overlays** in the project tree. Plot folders let you group plots with the same quantity together. All field plots under the same folder share the same color key.

> **Note:** All plots (field overlays) in the same folder have the same scale settings. To plot the same field with a different scale, you can create or move the new plot to a separate folder. Plots in different folders have a different plot keys.

6. Select the solution to plot from the **Solution** drop-down menu.
7. Select **Noise Vibration** in the **Field Type** drop-down menu.
8. Under **Intrinsic Variables**, select the variable value(s) at which the field quantity is evaluated. For Noise Vibration, the variables available are **Freq**ency and **Phase**.
9. If desired, you can choose a different field quantity to plot from the **Quantity** list. For the **Har_Surf_Force_Density_Peak** quantity plotted on the surface of a geometry, the **Surface Smoothing** button is enabled. For the other two quantities, the **Surface Smoothing** button is disabled. For details on using the **Surface Smoothing** button, see Modifying Field Plots.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10. Select the volume or surface (region) in which to plot the field from the **In Volume** list.

    This selection enables you to limit plots to the intersection of a volume with the selected object or objects. You can select and deselect any items in the **In Volume** list. You can mix model objects with non-model boxes. For example you might want to see a plot from part of two model objects by restricting the region to a non-model box overlapping those parts.

    Note:

    - Multiple selection should be used when there is a discontinuous field on a surface. If not, the field on both sides of the surface is plotted and each interferes with the other.
    - Non-convex 3D non-model solids are not supported for Field Overlay Plots and Fields Calculator computations.
    - Object selections containing non-model object(s) are supported for Field Overlay Plots but are not supported for Fields Calculator computations.

11. *Optionally*, You can use the check box to select **Streamline** plot.

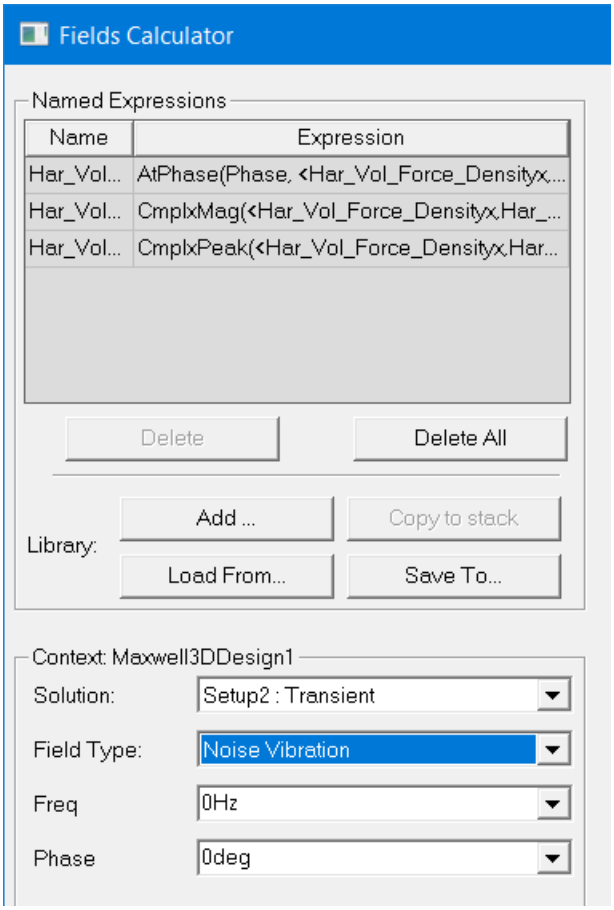
    Note:

    - Before creating the Streamline plot, select both starting and ending edges (in 2D) or both surfaces (in 3D).
    - In the **Create Field Plot** dialog box, select **In Volume:** region, which is the region in which the streamlines will appear. Streamlines are outside of objects that are excited (sources).
    - To show more streamlines after the plot is created, on the **Modify attributes...** dialog **Plots** tab, reduce the **Seeds density** value. If no streamlines appear, reduce this by a factor of 10 (or 100) because the default seeding was too large. Refer to Setting Field Plot Attributes for adjusting the Streamline display parameters, and Setting Fields Reporter Options for setting Streamline defaults.

    For exporting Streamline plots, refer to Exporting Field Plots.

12. *Optionally*, for 3D designs you can select the **Plot on surface only** check box to obtain a plot around the surfaces of selected objects. Similarly, for 2D designs, you can select **Plot on edge only** to obtain a plot around the edges of selected objects.
13. *Optionally*, for model objects in partial model designs, click the **Full Model** check box to calculate the fields on the full model based on the partial model solution.
14. *Optionally*, click **Fields Calculator** to open the fields calculator in which you can create named expressions.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



These can then be selected in the **Create Field Plot** dialog by choosing **Calculator** as the Category. The named expression(s) appear in the **Quantity** list.

15. Click **Done**.

The field quantity is plotted on the surfaces or within the objects you selected. The plot uses the attributes specified in the **Plot Attributes** dialog box.

The new plot appears in the view window. It is listed in the specified plot folder in the project tree. If you have created a field plot on a simulation in progress, the field plot is updated after the last adaptive solution.

If you want to update the field overlay before then, to view progress in the solution, select the Field icon in the Project tree that contains the field plot of interest, right-click to display the short cut menu, and select **Update Plots**.

To turn off the display of the plot, right-click on the plot and select **Plot Visibility** from the short-cut menu. Unchecking **Plot Visibility** turns off the plot display.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

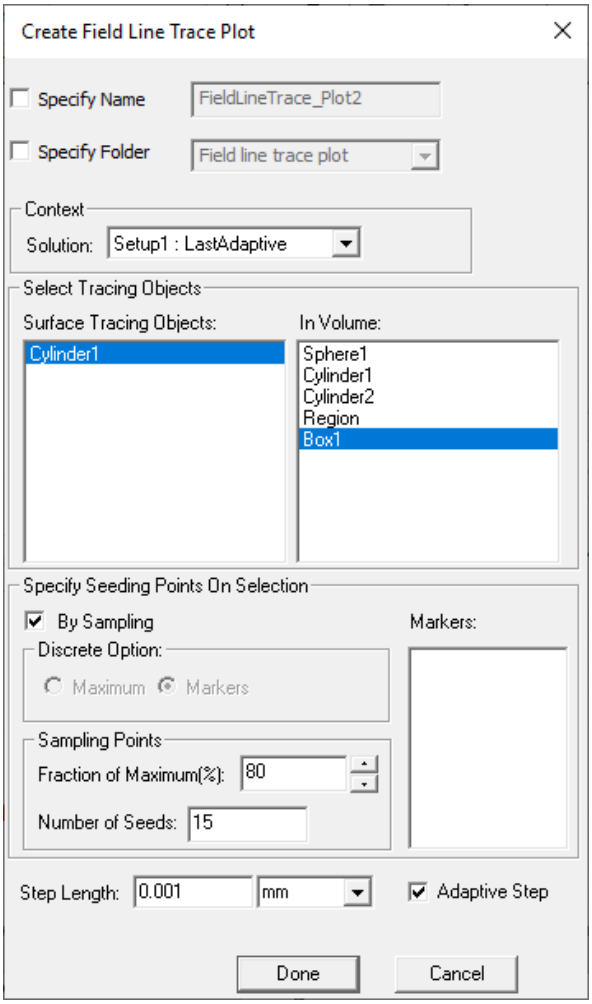
Plotting Field Line Trace Overlays for E Fields

For Maxwell electrostatic solutions, Field Line Trace overlays allow you to plot E field lines in 3D space and on insulator surfaces. The field lines are used as an indication of the high E field points and can be used to calculate the inception voltages for breakdown analyses.

> **Important:** This tool is not suitable for generating uniform field line plots.

To plot a field line trace overlay:

1. Select a surface from the geometry model or from a scalar surface plot.
2. Launch the **Create Field Line Trace Plot** dialog from the main **Maxwell** > **Fields** > **Field** > **Field Line Trace** menu, from the modeler context menu, or from the **Fields Overlay** context menu in the Project Manager tree.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. To specify a name for the plot other than the default, select **Specify Name**, and then type a new name in the box. The **Specify Name** field shows a default name based on the field quantity you selected.
4. To specify a folder other than the default in which to store the plot, select **Specify Folder**, and then click a folder in the drop-down menu. Plot folders are listed under **Field Overlays** in the project tree. Plot folders let you group plots with the same quantity together. All field plots under the same folder share the same color key.

   > **Note:** All plots (field overlays) in the same folder have the same scale settings. To plot the same field with a different scale, you can create or move the new plot to a separate folder. Plots in different folders have a different plot keys.

5. Select the desired **Surface Tracing Objects** and **In Volume** from the respective lists. You can use the **Ctrl** and **Shift** keys to select multiple items. The objects should be conductors within the gas material. The field line will be generated by tracing the surface of objects selected in **Surface Tracing Objects** and plotting the displacement of those surfaces only within the region selected **In Volume**.The field line tracing is from all the mesh grid points on the boundary of objects. If the electric field vector direction is towards the outside of selected object, the field line is not created.
6. The **Specify Seeding Points on Selection** panel allows you to manage the seeding points that will be used when calculating the trace lines:
   - If **By Sampling** is not selected, you can select either the **Maximum**, or **Markers** if markers for an E Field overlay plot have been set.
   - If **By Sampling** is selected, **Maximum** and **Markers** choices are disabled, and you can set a **Fraction of Maximum (%)** percentage value and the **Number of Seeds** to be used when calculating trace lines. **Fraction of Maximum** has a minimum value of 1%, and the maximum value for **Number of Seeds** is 99.
7. Enter the **Step Length** and unit. Step length controls the spacing between evaluation points along the E field direction. If **Adaptive Step** is checked, the step length will be inversely proportional to the E field strength.
8. Click **Done** to close the dialog box.

You can now analyze the design to generate the field line trace plot. After the field line trace plot has been generated, you can invoke the **Inception Voltage Evaluation** dialog box, which allows you to calculate the breakdown voltage - the voltage at which corona discharge will occur for the selected seed position and gas type.

Inception Voltage Evaluation

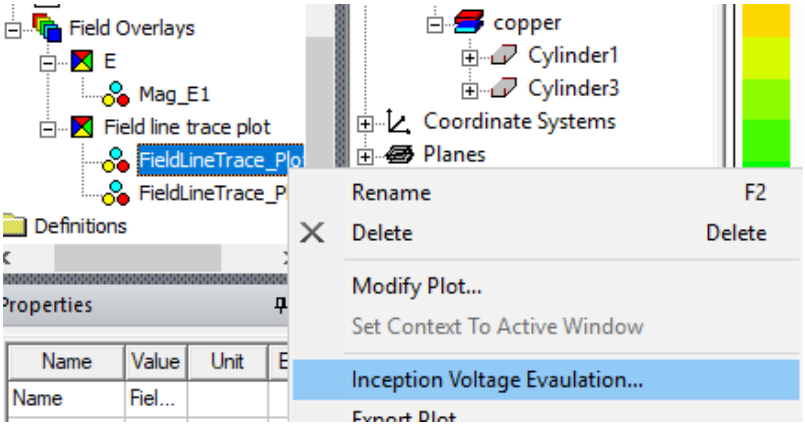
For Maxwell electrostatic solutions, Field Line Trace overlays allow you to plot E field lines in 3D space and on insulator surfaces. After a field line trace plot has been generated, you can invoke the **Inception Voltage Evaluation** dialog box, which allows you to calculate the breakdown

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

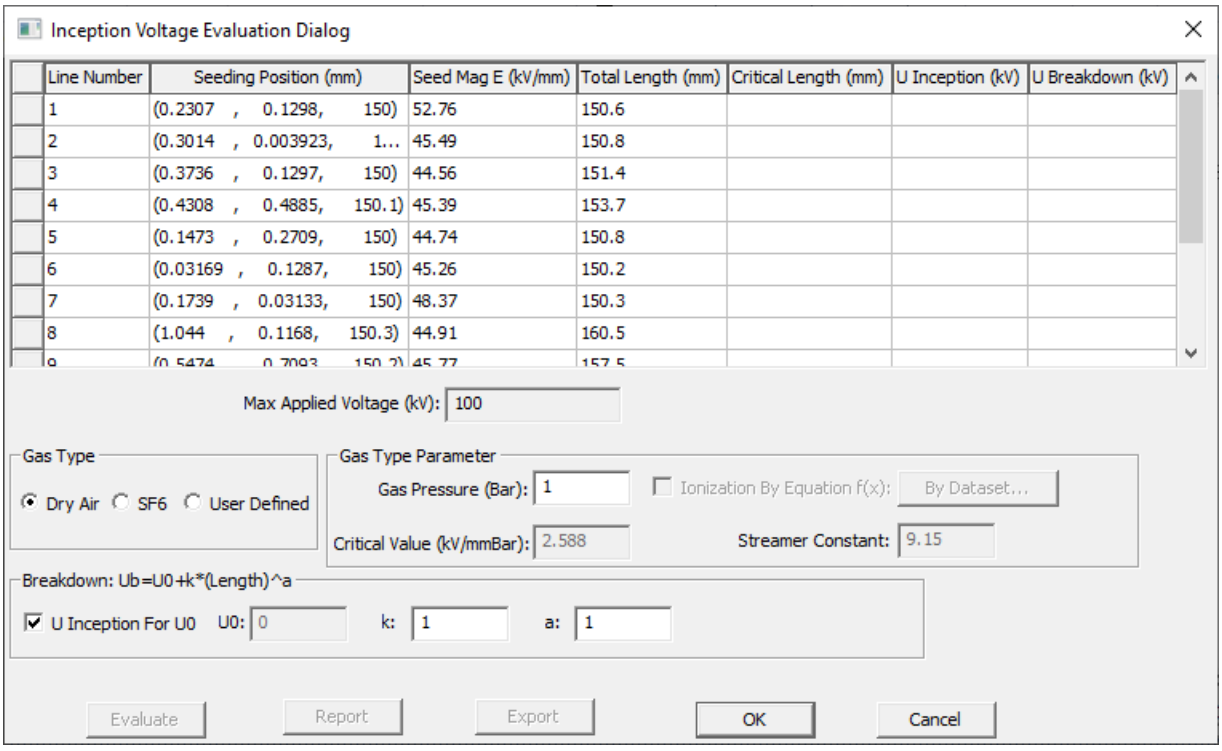
voltage – the voltage at which a corona discharge will occur for the selected seed positions and gas type.

To evaluate inception voltages:

1. For any solved field line trace plot, launch the **Inception Voltage Evaluation** function from the **Fields Overlay** context menu in the Project Manager tree.



The **Inception Voltage Evaluation** window opens.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following settings are available:

- **Seeding position**: the starting point of the field line
- **Seed Mag E**: the simulated electric field magnitude on the seeding point at max applied voltage difference
- **Total length**: evaluated streamer path total length
- **U Critical**: evaluated voltage at which the gas medium starts to ionize
- **Critical Length**: the streamer path length with electric field > critical value
- **U inception**: the evaluated voltage at which the streamer is formed with defined electron numbers; it is not the max applied voltage difference.
- **U breakdown (Breakdown voltage Ub)** – the evaluated voltage at which the gas changes from insulation state to conducting state
- **Max Applied Voltage Difference** – the difference between the highest defined voltage and lowest defined voltage
- **Gas Pressure** – Gas pressure used for evaluation
- **Critical Value** – the critical electric field value at which the gas starts to ionize
- **Streamer Constant** – the number related to the critical electron numbers of electrons in the streamer
- **Uo** – a constant voltage offset value
- **k,a** – constants, each type of gas has different value

2. The **Gas Type** panel allows you to choose either **Dry Air**, **SF6** (sulfur hexaflouride), or a **User Defined** gas. The selection controls which gas type parameters can be entered.

3. In the **Gas Type Parameter** panel:

   - For **Dry Air** and **SF6**, only the **Gas Pressure** value (in **Bar** units) can be adjusted. The **Critical Value** and **Streamer Constant** parameters are set to predetermined values.
   - For **User Defined** gas types, in addition to **Gas Pressure**, you can also set values for the **Critical Value** and **Streamer Constant** parameters. Additionally, you can click the **By Dataset** button to open an **Edit Dataset** dialog in which you can enter data points that define the **Ionization Coefficient Curve** for the gas type you are using. If you check **Ionization By Equation f(x)**, you can specify a function for Ionization (for example, 16.8*x –81.0). Typically, the **User Defined** values would come either from experimental test results or from a manufacturer.

4. In the **Breakdown** panel with **U Inception for U0** unchecked, you can enter the **U0**, **k**, and **a** parameters for the breakdown equation where the **Length** is the total length of the concerned trace line. When **U Inception for U0** is checked, **U0** is the evaluated inception value.

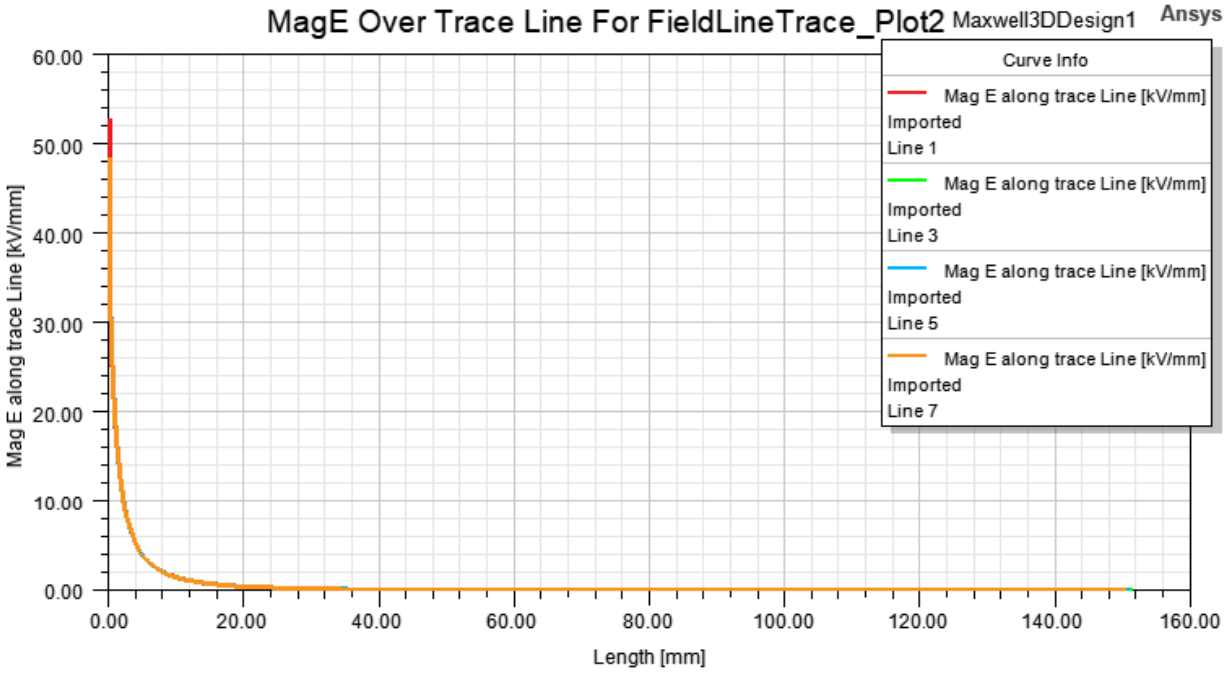
5. Select the row(s) for which you wish to evaluate **Critical Length (mm)** and **U Inception (kV)**, and **U Breakdown (kV)** values. You can use the **Ctrl** and **Shift** keys to select

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

multiple rows. You can select all rows by double-clicking any cell. You can sort by column by clicking the column header, click it again to reverse the order.

6. Click **Evaluate** to have Maxwell generate the **Critical Length (mm)**, **U Inception (kV)**, and **U Breakdown (kV)** values for the selected rows. This data can be useful when choosing the insulation for a design.

7. You can select one or more rows from the Inception table to create a XY plot by clicking the “Report” button. A curve will be created for each selected trace line. It will be a curve of MagE of each point on the trace line vs the segment length of the point to the starting point of the trace line. For example,



8. When **U Inception (kV)** and **U Breakdown (kV)** are computed for any trace lines, the **Export** button is enabled. You can select any rows to export or export all computed rows if there is no selection. (However, if any of the selected lines don’t have computed inception/breakdown data, the “Export” button will be disabled.) The data is exported as a text file.

   A sample exported text file (*.txt) is shown below, which includes the *Field Line Plot Inception Settings* that are used to compute the inception and breakdown voltages. It also includes the maximum applied voltage for the design and the inception table for the exported lines.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
# Ansys ElectronicsDesktop 2022.1.0
# Field line plot: FieldLineTrace_Plot1, Inception data file, version 1.0

Field Line Plot Inception Settings:
Gas Type: Dry Air
Gas Pressure: 1.0000000000000000e+00
Use Inception: true
Potential U0: 0.0000000000000000e+00
Potential K: 1.0000000000000000e+00
Potential A: 1.0000000000000000e+00

Max applied voltage: 1.0000000000000000e+02(kV)
Line ID, Table Index, Seed Position(mm), Seed Mag E(kV/mm), Total Length(mm), Critical Length(mm), U Inception(kV), U Breakdown(kV)
0, 1, (2.3070381908095705e-01, 1.2979182877146406e-01, 1.5001759493820100e+02), 5.2762025637338382e+01, 1.5057577619847515e+02, 1.1716558116651508e+00, 1.4075577105973363e+01, 1.6465135330444852e+02
2, 3, (3.7362326532583046e-01, 1.2973613679416152e-01, 1.5003999575422500e+02), 4.4555986426930822e+01, 1.5138148173590182e+02, 1.2050596224905987e+00, 1.4098477232850046e+01, 1.6547995896875187e+02
3, 4, (4.3076251393666537e-01, 4.8847738196739027e-01, 1.5010958571062298e+02), 4.5386888094631729e+01, 1.5371815568704838e+02, 1.2026495369371775e+00, 1.4128121527752524e+01, 1.6784627721480089e+02
4, 5, (1.4733129741472195e-01, 2.7087188864013190e-01, 1.5002441955487799e+02), 4.4740231766632597e+01, 1.5082887732437518e+02, 1.1796928016782176e+00, 1.4082413189396163e+01, 1.6491129051377135e+02
```

Named Expressions

Named expressions can contain any combination of scalar, vector, or geometry functions. When you plot a field overlay or create a report based on a named expression, only corresponding quantities are available.

For example, if you select a point for the **Geometry** and a **Category** that is a named expression based on the point (or other scalar, non-3D value), then the **Quantity** list only lists expressions that returns a single scalar value. If you have added named expressions that take the magnitude of the ScalarX(), ScalarY(), or ScalarZ() of a vector point value, then you can create output variables for those expressions only in this case.

Named expressions can be plotted in the following three ways:

- To create a field overlay of a named expression, the expression must be a real value (scalar or vector) that has values everywhere in space (or at least on every point in the geometry you plan to use for the field overlay).
- To create a 2D report from a named expression that evaluates in the Fields Calculator as a single scalar value, the expression must result in a single-valued, real, scalar value. A single-valued item could be the value on a single point in space or the result of a function that returns a single value (such as an integration, max/value, min/value, or other function).
- To create a 2D report from a named expression that is evaluated (in the Fields Calculator) along a polyline, the expression must be a real scalar that has values everywhere in space (or at least everywhere on the line object you want to use to sample the values). Named expressions can be created in the Fields Calculator.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plotting Derived Field Quantities

Derived field quantities are field quantity representations that have been deduced from the original field solution using the Fields Calculator.

1. Select a point, line, surface, or object to create the plot on or within. You may also select a plane or object selection in the history tree. If it does not exist, create it.

   > **Note:** For 2D Designs, a plane selection must be consistent with the drawing plane or an error will result.

2. Click **Maxwell 3D** or **Maxwell 2D** and select **Fields** > **Fields** > **Named Expression**.

   The **Selecting calculated expression** dialog box appears.

3. Select the derived quantity you want to plot, and then click **OK**.

   The **Create Field Plot** dialog box appears.

4. To specify a name for the plot other than the default, select **SpecifyName**, and then type a new name in the **Name** box.
5. To specify a folder other than the default in which to store the plot, select **Specify Folder**, and then click a folder in the drop-down menu. Plot folders are listed under **Field Overlays** in the project tree.
6. Select the solution to plot from the **Solution** drop-down menu.
7. Select the derived field quantity to plot from the **Quantity** list.

   > **Note:** If you select a point for the **Geometry** and a **Category** that is a named expression based on the point (or other scalar, non-3D value), then the **Quantity** list only lists expressions that returns a single scalar value. If you have added named expressions that take the magnitude of the ScalarX(), ScalarY(), or ScalarZ() of a vector point value, then you can create output variables for those expressions only in this case.
   >
   > If you select a polyline for the **Geometry** and a **Category** that is a named expression based on the line, then the **Quantity** list only lists corresponding expressions (*i.e.*, will not list scalar values when a line is selected as the geometry).

8. Select the volume or surface (region) in which to plot the field from the **In Volume** list.

   This selection enables you to limit plots to the intersection of a volume and the selected object.

9. Click **Done**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The derived field quantity you created in the Fields Calculator is plotted on the surfaces or objects you selected. The new plot is listed in the project tree under **Field Overlays**.

Creating 2D Reports from Named Expressions

You can create a 2D report from named expressions that evaluate as a single scalar value or that evaluate along polylines.

To create a report from a named expression evaluating as a single scalar value:

1. Click **Maxwell**, and then click **Results** > **Create Report**.

   The *Create Report* dialog box appears.

2. In the **Target Design** drop-down menu, click the design containing the solution data you want to plot.
3. Select **Fields** from the **Report Type** drop-down menu.
4. In the **Display Type** drop-down menu, select the type of report you want to create.
5. Click **OK**.

   The *Traces* dialog box appears.

6. In the **Solution** drop-down menu, click the solution containing the data you want to plot.
7. Add one or more traces to include in the report.
8. Click **Done**.

   The report appears in the view window and is listed in the project tree. Once you have created a report, additional options become available on the **Results** submenu.

To create a report from a named expression evaluating along a polyline:

1. Click **Maxwell** and then click **Results** > **Create Report**.

   The *Create Report* dialog box appears.

2. In the **Target Design** drop-down menu, click the design containing the solution data you want to plot.
3. Select **Fields** from the **Report Type** drop-down menu.
4. In the **Display Type** drop-down menu, select the type of report you want to create.
5. Click **OK**.

   The *Traces* dialog box appears.

6. In the **Solution** drop-down menu, click the solution containing the data you want to plot.
7. Select the geometry you want to plot from the **Geometry** drop-down menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Add one or more traces to include in the report.

   > **Note:**
   >
   > If you select a polyline for the **Geometry** and a **Category** that is a named expression based on the line, then the **Quantity** list only lists corresponding expressions. The list will not contain scalar values when a line is selected as the geometry.

9. Click **Done**.

   The report appears in the view window and is listed in the project tree. The default name is based on the Report Category you selected, (for example, S Parameter Plot *n* or rE Plot *n*). You can edit the plot names in the project tree and the plot header text in the report synchronizes. Once you have created a report, additional options become available on the **Results** submenu.

Creating Scalar Field Plots

A scalar plot uses shaded colors or contoured lines to illustrate the magnitude of field quantities on surfaces or volumes.

1. Do one of the following:
   a. To create a scalar surface plot, select the faces on which you want to plot the fields.
   b. To create a scalar volume plot, select the objects within which you want to plot the fields.
2. Click **Maxwell** > **Fields** > **Plot Fields**.
3. On the **Plot Fields** menu, click the scalar field quantity you want to plot.

   The **Create Field Plot** dialog box appears.

4. Follow the procedure for plotting field overlays.

The plot uses the attributes specified in the **Plot Attributes** dialog box.

The new plot will be listed in the specified plot folder in the project tree.

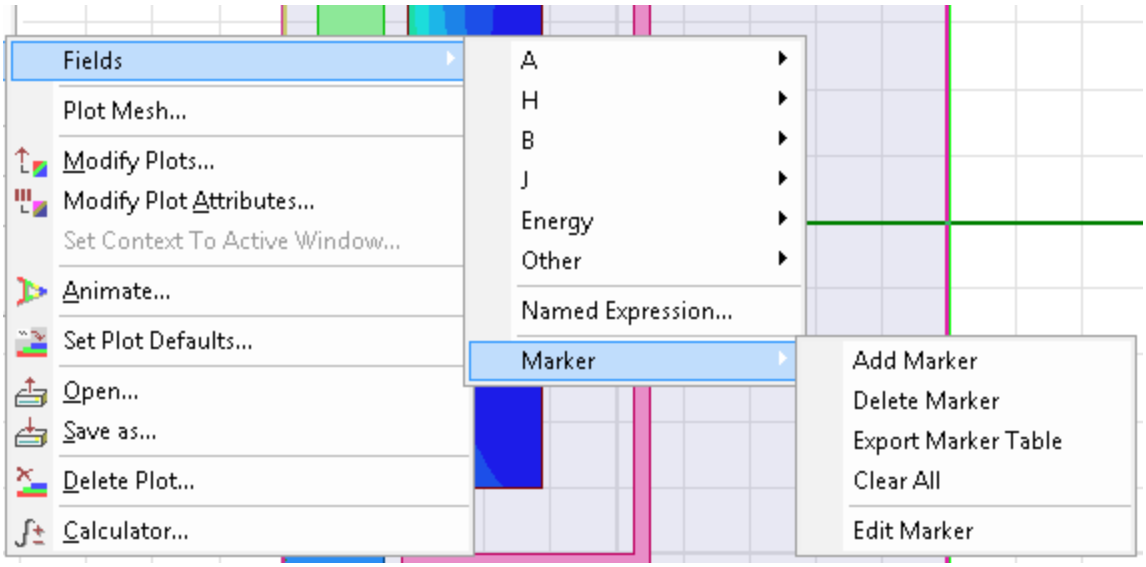
Working with Scalar Field Plot Markers

The field overlay plot marker feature enables you to create a marker at selected points in the scalar field overlay plot geometry, and to obtain the field value at that point. The fields Marker sub-menu enables you to:

- Add Marker
- Delete Marker
- Export Marker Table

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Clear All markers
- Edit Marker



Adding a Field Plot Marker

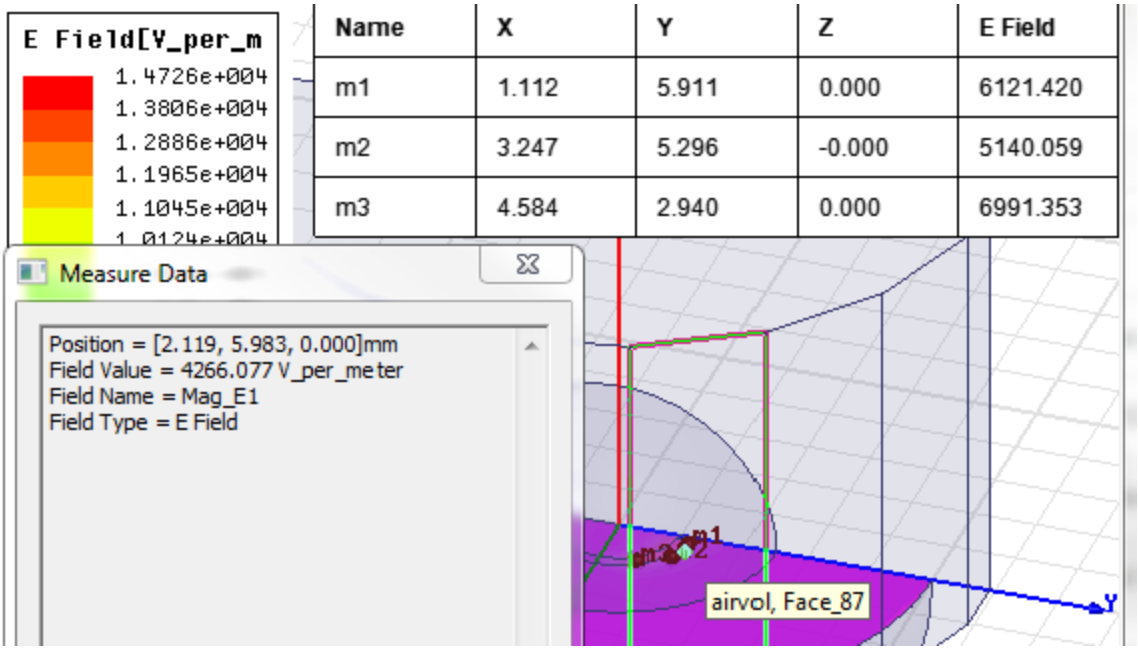
To add one or more field plot markers to a scalar field:

1. On the main menu click **Maxwell** and then select **Fields** > **Plot Fields** > **Marker** > **Add Marker**. Alternatively, you can right-click anywhere in the modeler window, or on the Field Overlays folder icon in the Project Manager, and select **Fields** > **Marker** > **Add Marker**.

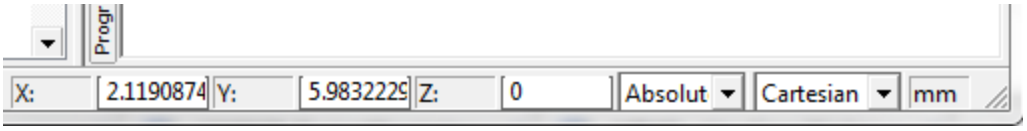
   The *Measure Data* dialog box opens, and a round dot appears at the tip of the cursor.

2. Drag the dot over the spot on the field overlay plot where you want to add a marker. The Measure Data dialog box shows detailed information for the spot currently under the dot.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Click the desired point in the field overlay plot to add the marker at that location. Alternatively you can select the position of the marker by entering the values manually in the edit fields at the bottom of the window as shown below.



A table showing the marker coordinates and associated field value is also created and added to the modeler window.

4. Repeat as desired to add additional markers.
5. Press the **Esc** key when finished adding markers.

Deleting a Field Plot Marker

To delete a field plot marker:

1. Click on the marker you want to delete to select it. The row corresponding to the selected marker will be highlighted in the marker table.

   Press and hold the **Ctrl** key and click to select multiple markers.

2. On the main menu click **Maxwell** and then select **Fields** > **Plot Fields** > **Marker** > **Delete Marker** to delete the selected marker(s).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Alternatively, you can right-click anywhere in the Modeler window, or on the Field Overlays folder icon in the Project Manager, and select **Plot Fields** > **Marker** > **Delete Marker**.
- You can also simply press the **Delete** key to delete the selected marker(s).

Exporting a Field Plot Marker Table

You can export a field plot marker table to either a comma- or tab-delimited file as follows:

1. On the main menu click **Maxwell** and then select **Fields** > **Plot Fields** > **Marker** > **Export Marker Table**.

   Alternatively you can right-click anywhere in the modeler window, or on the Field Overlays folder icon in the Project Manager, and select **Plot Fields** > **Marker** > **Export Marker Table**.

2. In the *Export As* dialog box, choose the export format, either **.csv** or **.tab**, and save the file in the desired location.

   The exported file can then be imported into another application such as a spreadsheet.

	A	B	C	D	E
1	Name	X	Y	Z	H[A_per_m]
2	m1	2	22.935	1.517	1234
3	m2	1.517	17.373	1.517	123
4	m3	2.488	23.516	1.517	234
5	m4	2.751	23.828	1.517	1324

Clearing All Field Plot Markers

To clear all field plot markers in the active modeler window do one of the following:

- On the main menu, click **Maxwell** and then select **Fields** > **Plot Fields** > **Marker** > **Clear All**.
- Alternatively, right-click anywhere in the Modeler window, or on the Field Overlays folder icon in the Project Manager, and select **Plot Fields** > **Marker** > **Clear All**.

Deleting all the markers also removes the marker table.

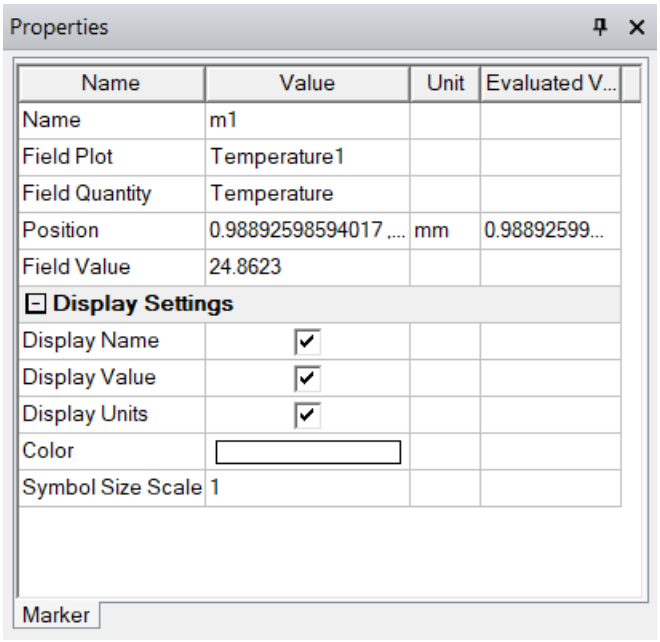
Editing Field Plot Markers

To edit field plot markers:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the marker table, click on the marker you wish to edit. The marker in the 3D Modeler window and its row in the table are highlighted. The properties of the marker are displayed in the **Properties** window.

> **Note:**
>
> Press **Ctrl** and select multiple marker rows in the table to edit marker properties simultaneously.



2. You can edit the following marker properties:
   - **Name**: Specify the name that appears next to the marker in the 3D Modeler window.
   - **Position**: Edit its coordinate values. The marker table changes to reflect the new value.
   - **Display Name**: Toggle to show or hide the marker's name.
   - **Display Value**: Toggle to show or hide the marker's value.
   - **Display Units**: Toggle to show or hide the marker's units.
   - **Color**: Click the color to change the marker's color in the **Color** dialog box.
   - **Symbol Size Scale**: Enter a different number to adjust the size of the marker in the 3D Modeler window.
3. Optionally, when a marker is selected, press the **Delete** key to delete the marker and its corresponding entry in the marker table.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

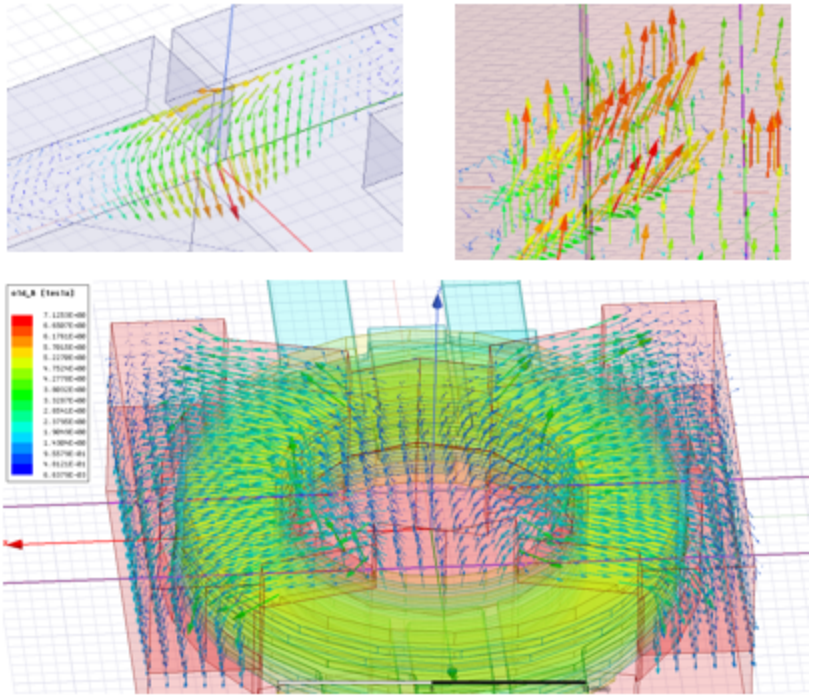
Showing Field Plot Markers

To display field plot markers in the 3D Modeler window, on the main menu click **Maxwell** and then select **Fields** > **Plot Fields** > **Marker** > **Show Marker**. The table showing the marker coordinates and associated field value is also displayed in the modeler window.

Alternatively, you can right-click anywhere in the modeler window, or on the Field Overlays folder icon in the Project Manager, and select **Plot Fields** > **Marker** > **Show Marker**.

Creating Vector Field Plots

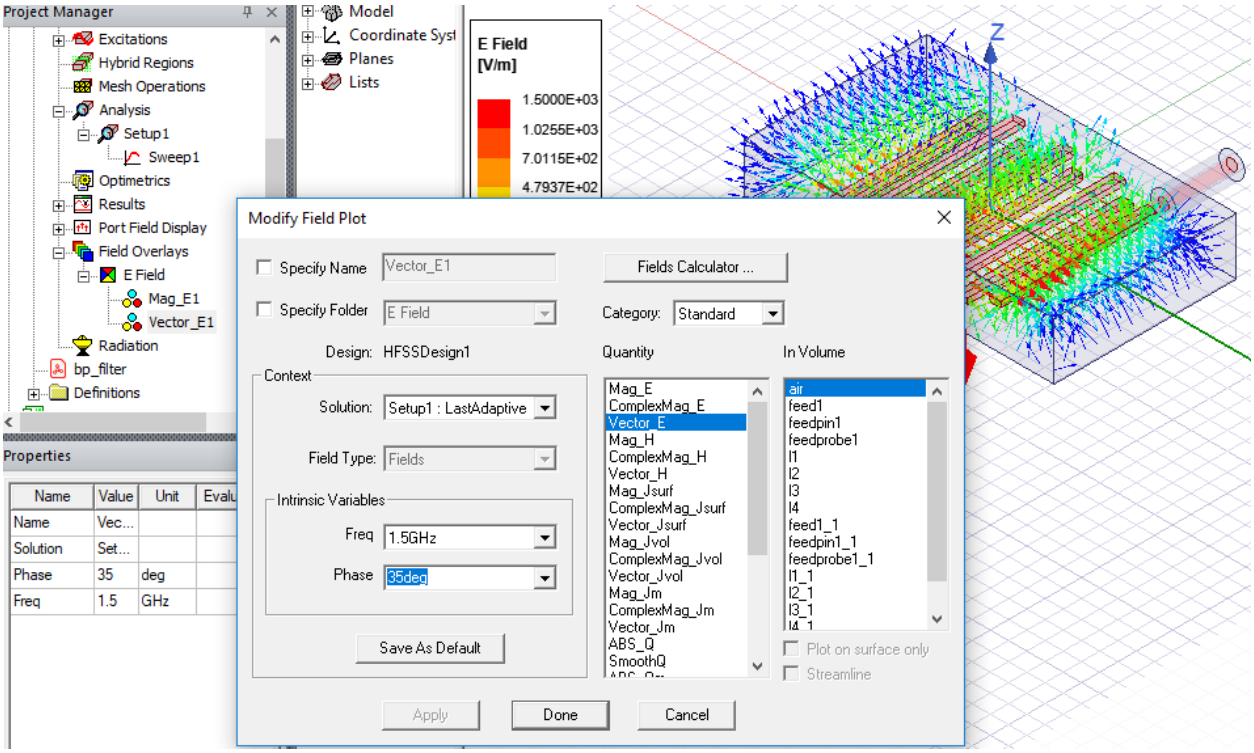
A vector plot uses arrows to illustrate the magnitudes of the x-, y-, and z-components of field quantities. Vector plots can be created on surfaces or volumes. The vector plots calculates a good initial spacing (min, max, and spacing range). The initial min, max spacing is based on a percentage of the plot geometry size and they are in model units. You can edit the vector min and max threshold spacing and the arrow representation.



1. Do one of the following:
    a. To create a vector surface plot, select the faces on which you want to plot the fields.
    b. To create a vector volume plot, select the objects within which you want to plot the fields.
2. Click **Maxwell** > **Fields** > **Fields**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

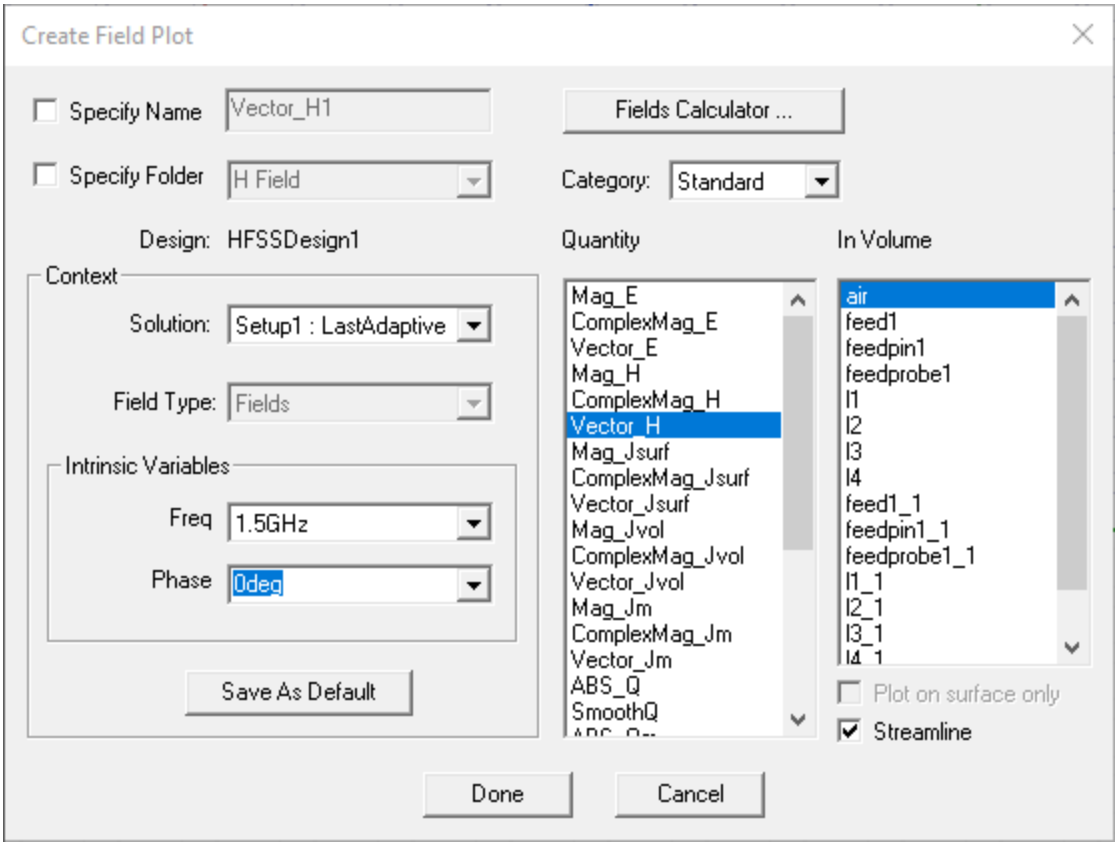
3. From the **Fields** submenu, click the vector field quantity you want to plot.



4. Follow the procedure for plotting field overlays.

If you select a vector quantity when creating a plot, you can also check Streamline for the plot.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

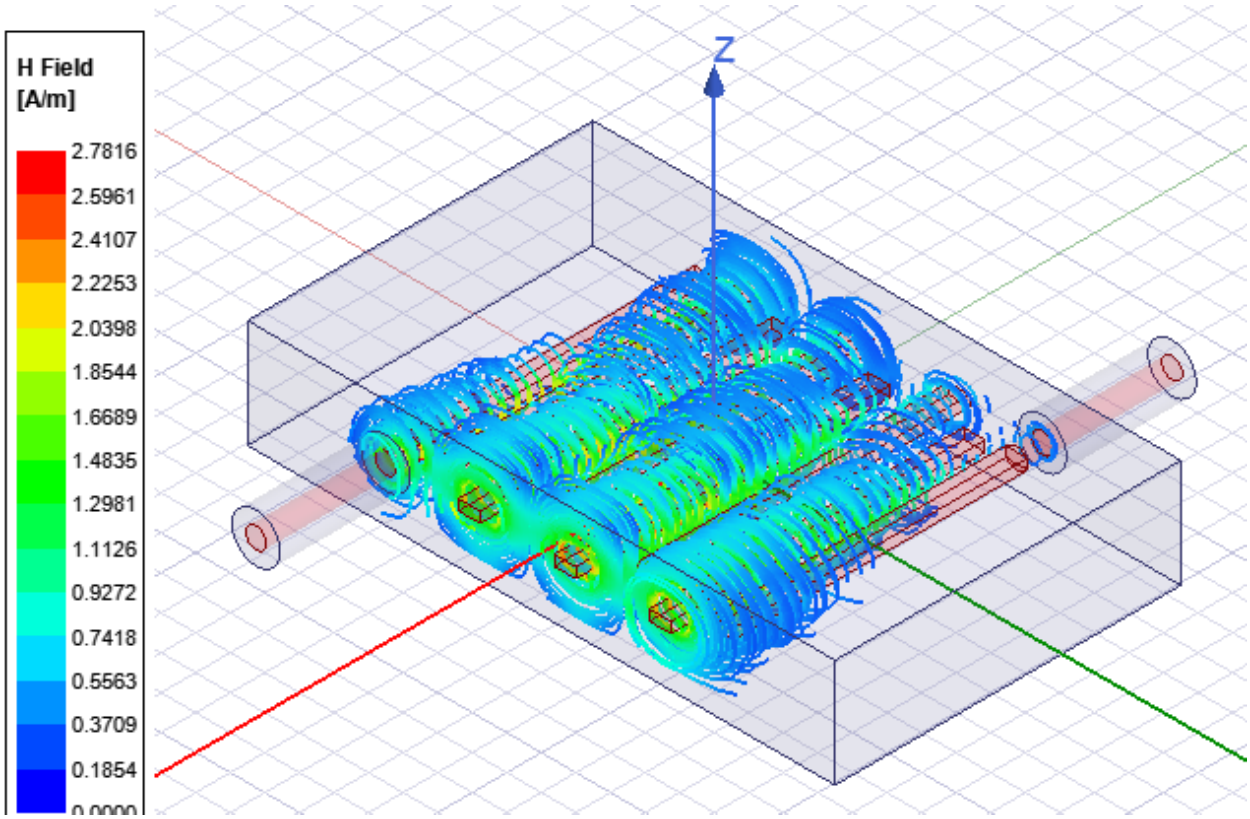


Notes for creating Streamline plots:

1. Before creating the plot, select both starting and ending edges (in 2D) or surfaces (in 3D).
2. In the Creating Field Plot menu, select "In Volume: Region" which is the volume in which the streamlines will appear and is outside of the sources.
3. After the plot is created, on the Attributes/Plot tab reduce "Seeds density" to show more streamlines. If no streamlines appear, reduce this by a factor of 10 (or 100) because the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

default seeding was too large.



Modifying Field Plots

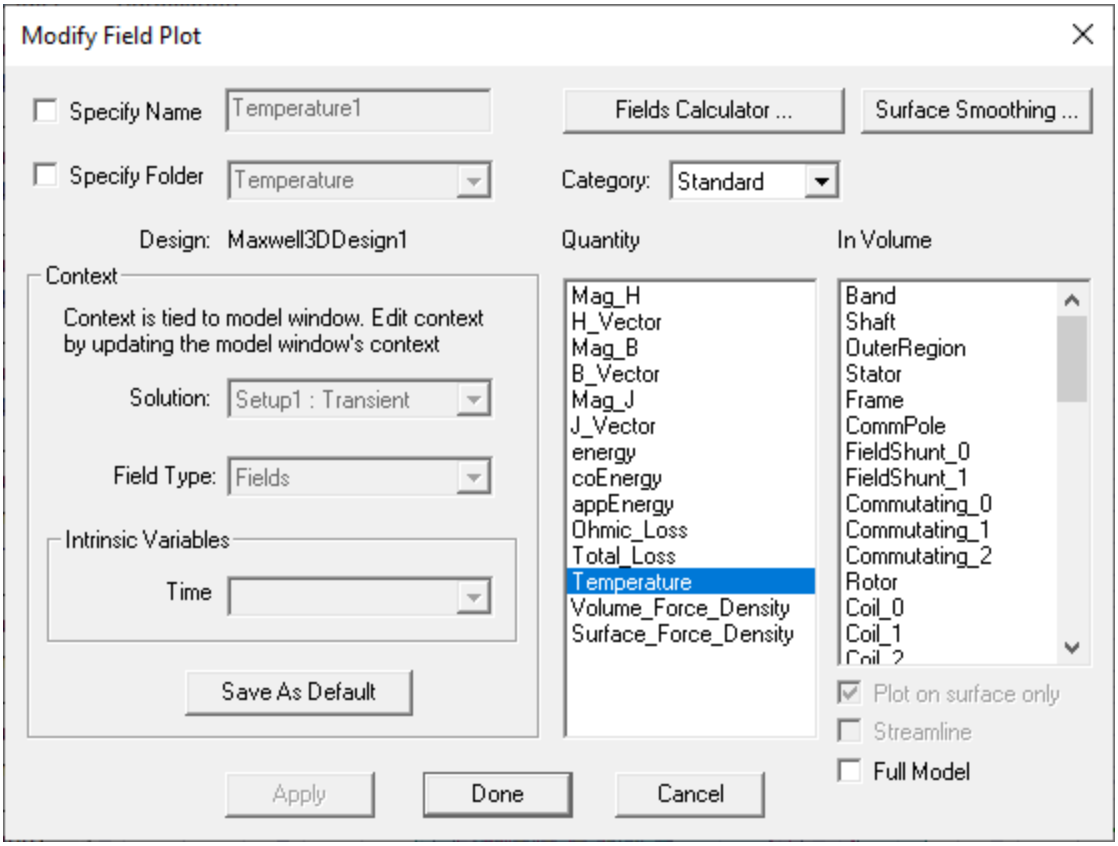
1. Click **Maxwell 3D** or **Maxwell 2D** and then select **Fields** > **Modify Plots** , or in the Project Manager tree, select the **Field Overlays** icon, right-click, and select **Modify Plots** or use the **M** hotkey.

   The **Select Field Plot(s)** dialog box appears.

2. Select the plot you want to modify from the **Select** column, and then click **OK**.

   The **Modify Field Plot** dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

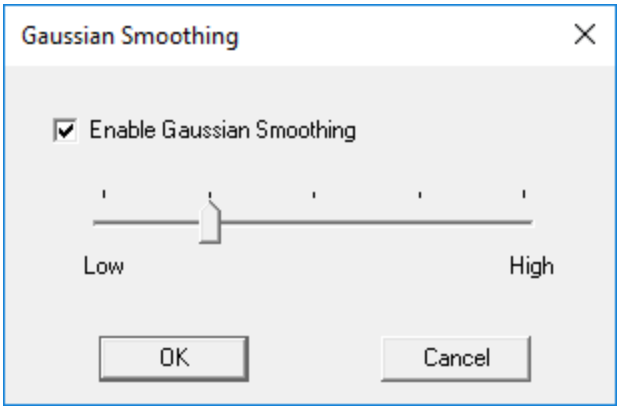


> **Note:** Optionally, you can right-click an existing plot listed under the Field Overlays in the Project tree, and select **Modify Plot** to open its **Modify Field Plot** dialog box.

3. Optionally, click a different solution to plot from the **Solution** drop-down menu.
4. Optionally, click a different field type (Fields or Noise Vibration) from the **Field Type** drop-down menu.
5. Optionally, click the **Specify Name** check box to enable the name field.
6. Optionally, select the **Specify Folder** check box to enable the drop-down menu to select a folder in which to store the plot.
7. Optionally, select a different field quantity to plot from the **Quantity** list.
    - To choose a calculated expression, select **Calculator** from the **Category** drop-down menu.
    - To choose a predefined field quantity, select **Standard** from the **Category** drop-down menu.
    - For scalar quantities plotted on the surface of a geometry, the **Surface Smoothing** button is enabled. For vector quantities, the **Surface Smoothing** button is disabled.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Smoothing applies a selected level of smoothing based on the Gaussian Blur algorithm. Clicking **Surface Smoothing** opens the **Gaussian Smoothing** dialog box.



Select the **Enable Gaussian Smoothing** check box to enable the feature and use the slider to set the degree of smoothing.

> **Important:** You must click **Apply** in the **Modify Field Plots** dialog box to see the changes.

8. Under **Intrinsic Variables**, select the variable value(s) at which the field quantity is evaluated. Variables available vary depending on the solution type.
9. Select the volume, or region, in which the field is to be plotted from the **In Volume** list. This selection enables you to limit plots to the intersection of a volume and the selected object.
10. For model objects in partial model designs, click the **Full Model** check box to calculate the fields on the full model based on the partial model solution.
11. Click **Apply**, and then click **Done**.

The field quantity is plotted on the surfaces or within the objects you selected. The new plot is listed in the specified plot folder in the Project Manager tree.

The plot uses the attributes specified in the **Plot Attributes** dialog box.

Setting Field Overlay Attributes

After creating a mesh or field overlay on a surface or volume, you can modify its appearance by changing the settings in the **Plot Attributes** dialog box. You will modify the settings for a plot folder and all plots in that folder will use the same attributes.

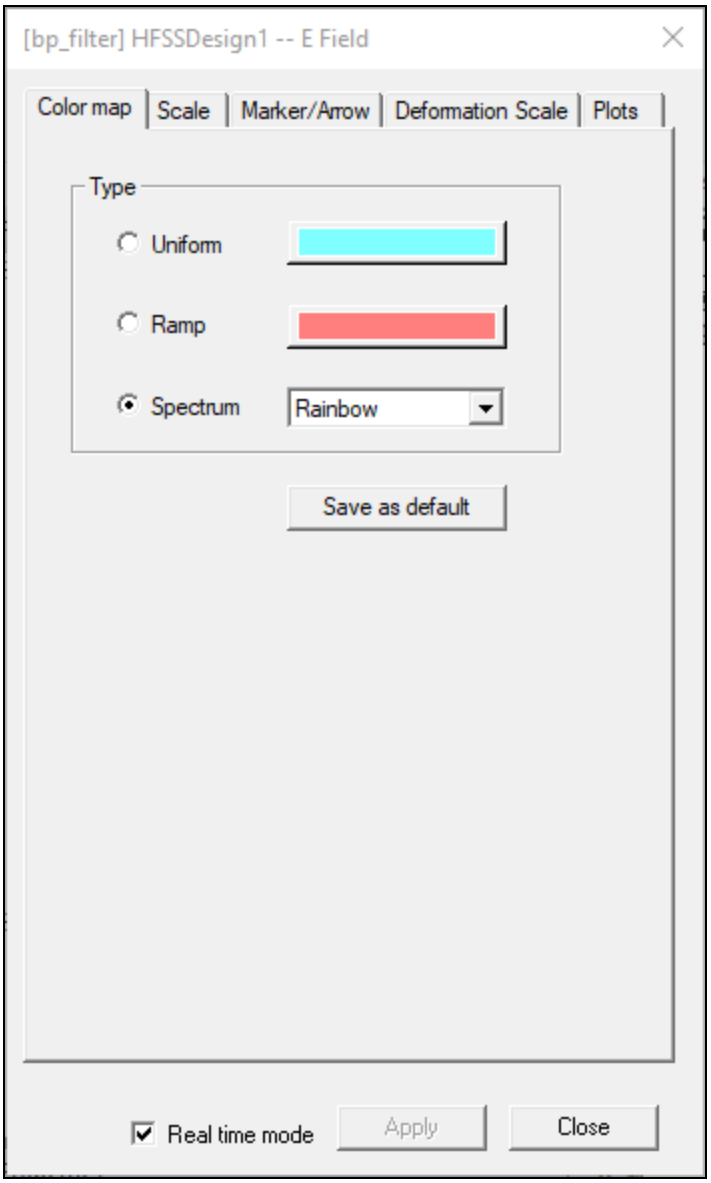
1. Click **Maxwell** > **Fields** > **Modify Plot Attributes**. Alternatively, right-click **Field Overlays** in the Project Manager and select **Modify Attributes** from the shortcut menu.

   The **Select Plot Folder** dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the **Select Plot Folder** dialog box, select the plot type that you want to modify and click **OK**. You can also right-click the specific plot in the Project Manager and choose **Modify Attributes** from the shortcut menu.

   A dialog box with attribute settings for the selected plot type (whether a field result or a mesh plot) appears.



3. For field result overlays (not mesh plots), the available plot attributes under each tab in the dialog box are listed in the following table:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Color map:**

  The number of colors used and how they are displayed. The field data must be available for the color key to appear.

- **Scale tab:**

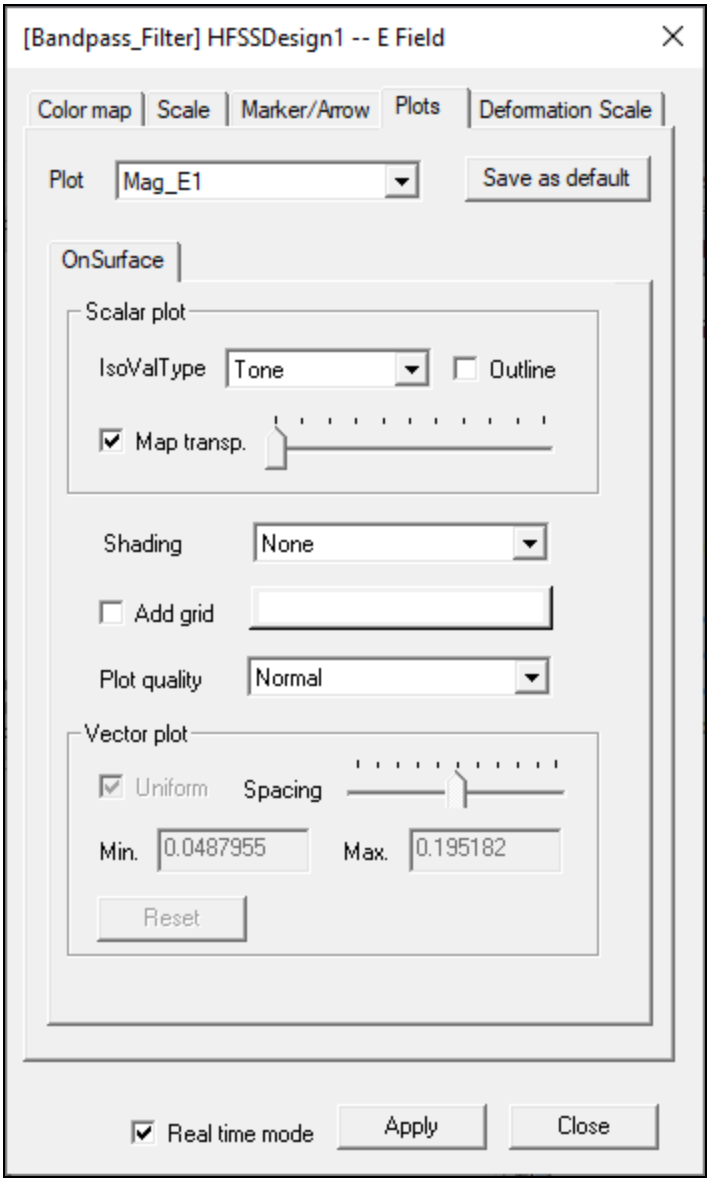
  The scale of field quantities, including the number of divisions in the scale, whether to use dB as the units, whether to use a linear or log scale, auto schall options, and plot number format.

- **Marker/Arrow**
  - The appearance of points (for scalar point plots).
  - The appearance of arrows (for vector plots).
  - Magnitude filtering (for vector plots). That is, you uncheck Map size, and specify a Min and Max Magnitude, or use a slider to set the Min threshold.
- **Deformation Scale:**

  This is for use with plots that include Stress feedback from Ansys Workbench Integration.

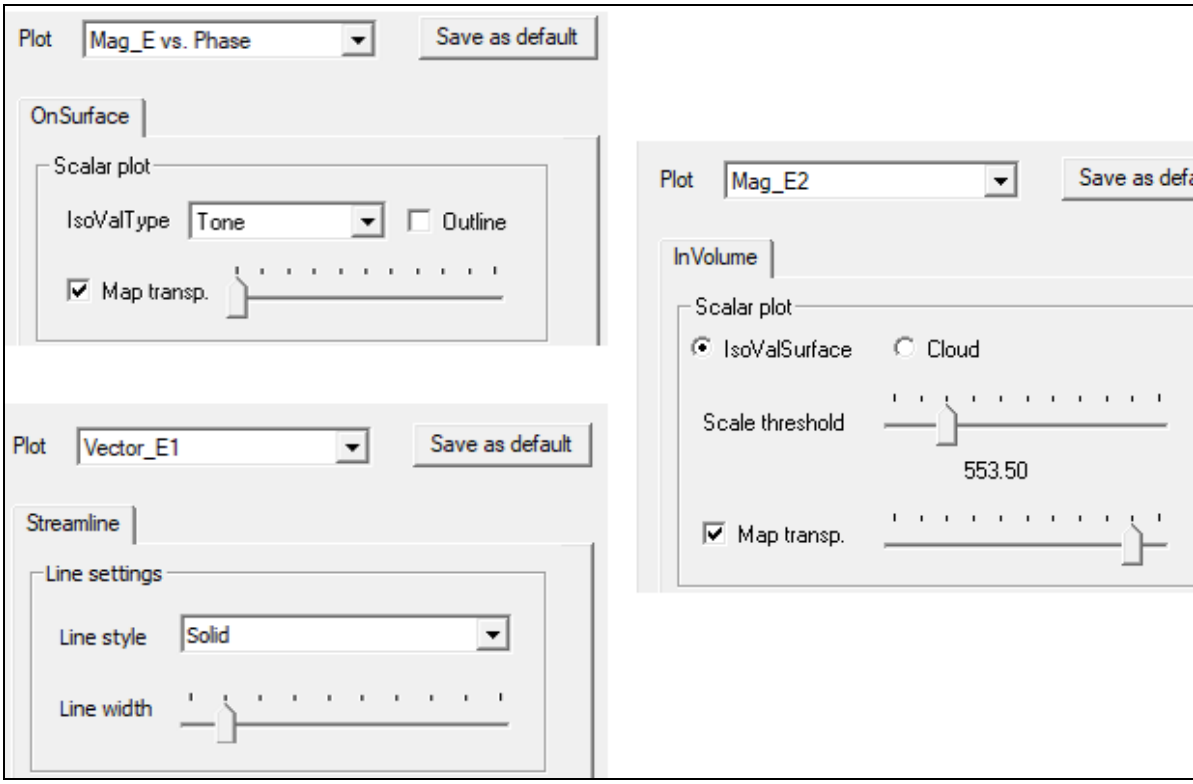
- **Plots (if not vector or streamline):**
  - The **Plot** selected. If multiple plots are available, you can select from a drop down menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Plots can be **OnSurface** or **InVolume** or **Streamline**. The type of plot you select determines the control options shown.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



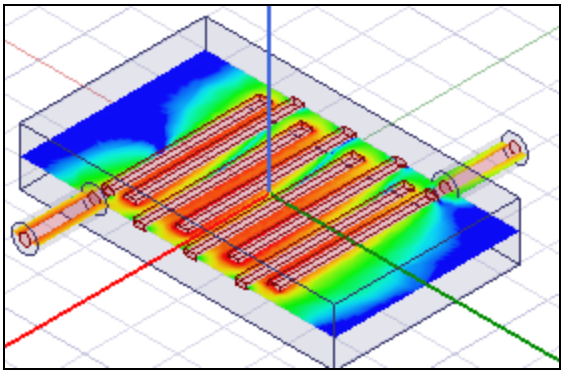
- For scalar surface plots you can select the type of isovalue display. For *Line, Fringe* and *Tone* **IsoValType**, *Outine* is enabled. For *Gourad*, it is disabled.
- Whether to use Shading (for scalar plots), if lighting is turned on. By default, the **Shading** is set to *None* which is equivalent to "Do not use lighting." Shading can be Flat or Smooth.
- **None** Equivalent to selecting **Do not use lighting** on the Lighting Properties dialog box.

  Flat Provides a flat rendering of the field plot.

  Smooth Provides a smooth rendering of the field plot.
- The transparency based on solution value.
- Whether to add a grid (that is, a mesh overlay), and to set the grid color.
- Specify the plot resolution as **Coarse**, **Normal**, **Fine**, or **Very Fine**.

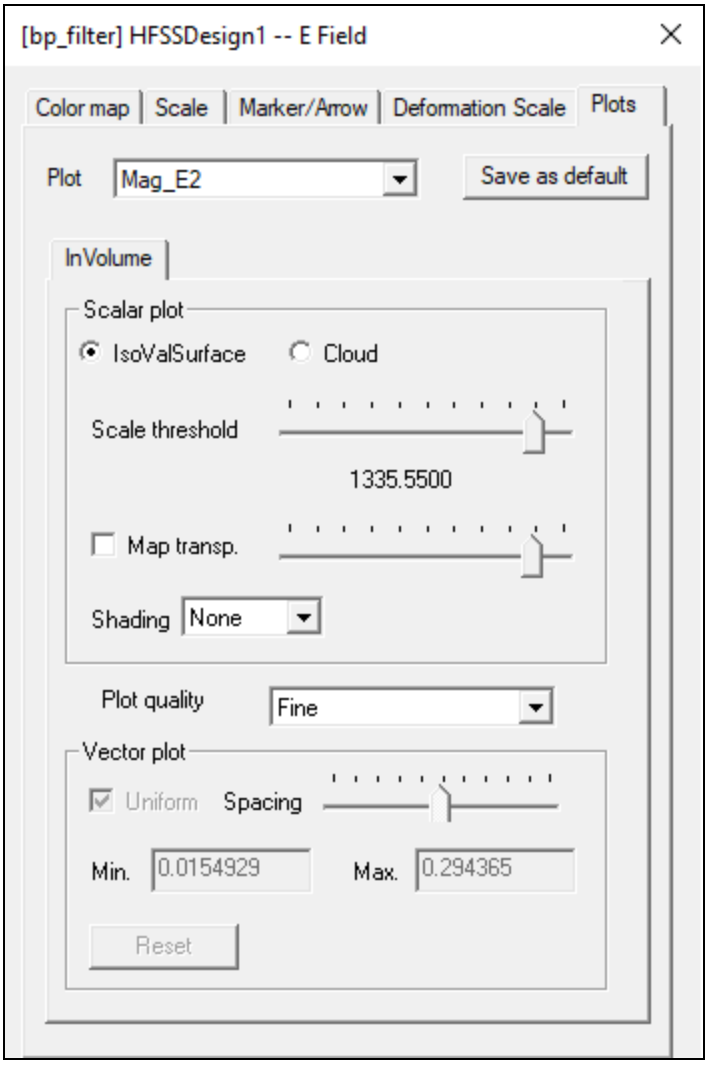
  This affects the use of memory for animating plots. For large plots with more frames to animate, use **Coarse** or **Normal** to reduce memory requirements and improve performance. For smaller plots with few frames, if higher resolution is required, use **Fine** or **Very Fine**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



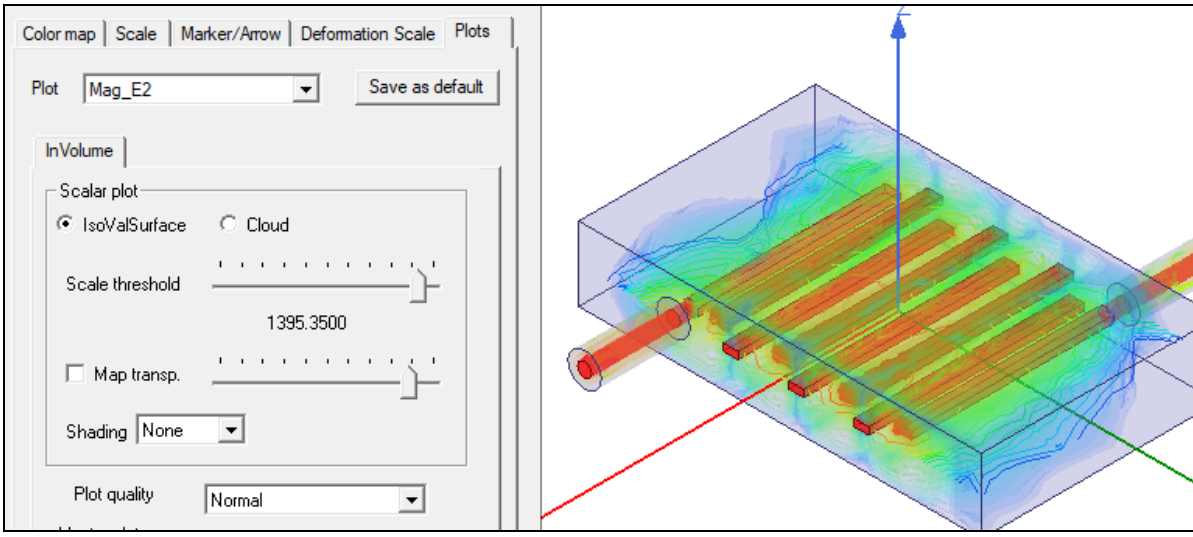
If you select an InVolume plot, display options are available to allow visibility of internal layers and objects, particularly those with high field values. For *IsoValSurface* for an *InVolume* Scalar plot, the choices are for **Scale threshold** and **Map transp**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

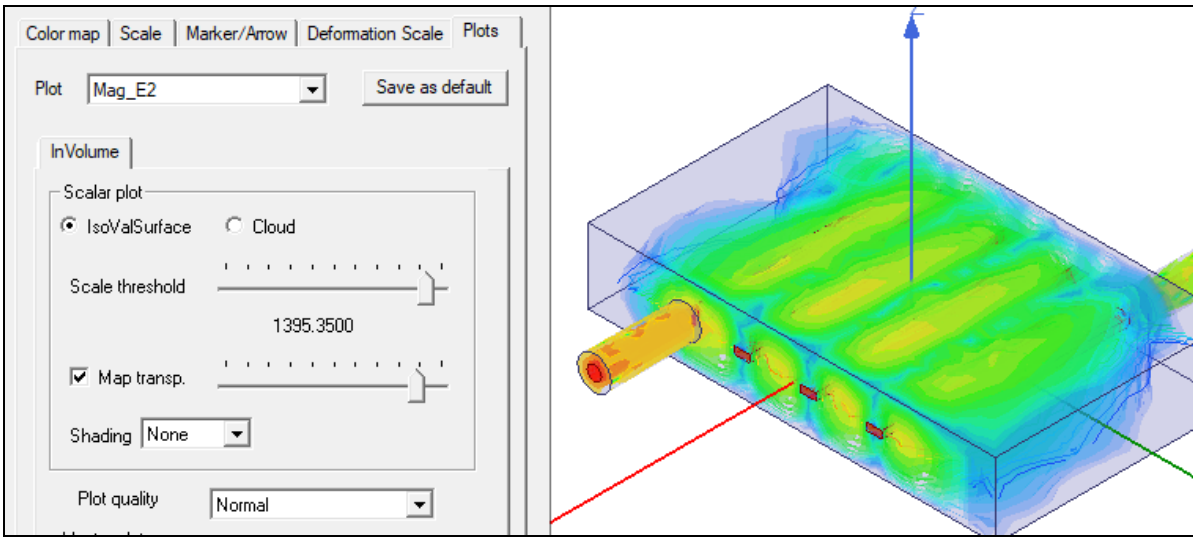


- The “**Scale threshold**” slider bar allows you to set a field value threshold so that iso surfaces with field value lower than the given threshold will show as translucent with given transparency specified by the slider bar below it.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



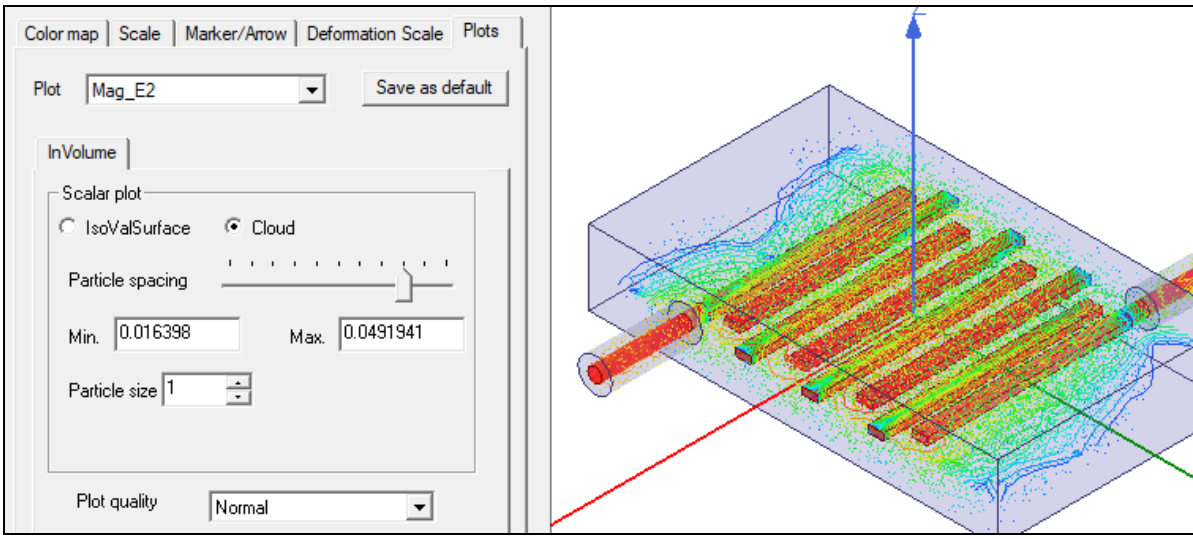
- Iso surfaces with field value higher than the threshold will show as opaque. The range of the "**Scale threshold**" is the same as the color key range from *Min* to *Max*.
- When "**Map transp.**" is checked, translucent iso surfaces field values map to [0.0, transparency] so lower field value map to higher transparency while higher field value to lower transparency. If it is unchecked, all translucent iso surfaces will have the same given transparency from the slider bar.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> When the **Map transp.** slider bar is at the left most position, transparency is 0. This feature is essentially disabled, and the "**Scale threshold**" slider bar will be disabled. Also, high fidelity transparency should be enabled for best results.

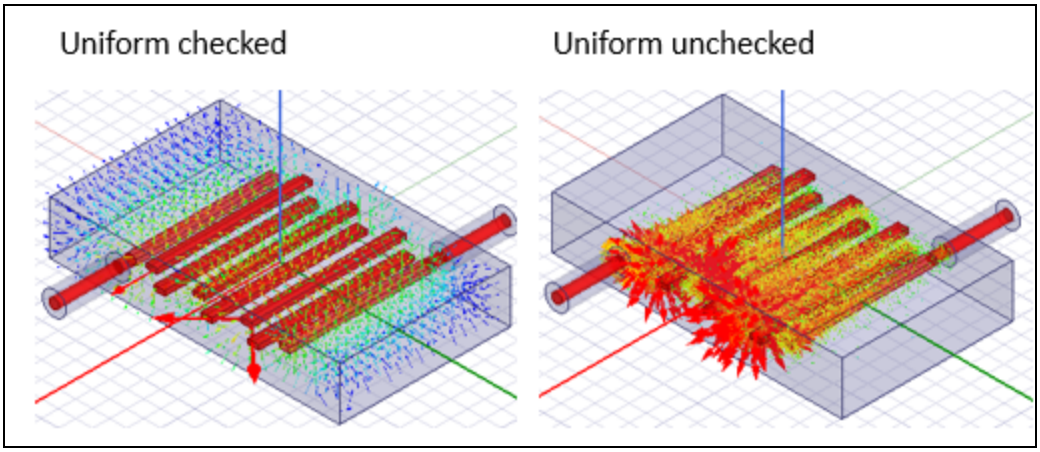
- For *Cloud* plots, field values are represented by points that illustrate the spatial distribution of the solution. You can use the **Particle spacing** slider bar to adjust the plot. The higher the solution value, the greater the cloud density. You can also specify **Min** and **Max** values (numeric values based on the active model length unit) and adjust **Particle size**.



- **Plots (for vector plots):**
  - The **Plot** selected.
  - The **Min** and **Max** spacing of arrows (for vector plots). These values are based on the model's active length unit and are only relevant when the **Uniform** option is selected.
  - **Plot quality**, as *Normal, Coarse, Fine,* or *Very Fine*.
  - **Uniform** vector spacing, based on the *Spacing* slider position and the *Min* and *Max* values.

    When Uniform is *not* selected, a vector is rendered at every node (both corner and mid-side nodes of the elements).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

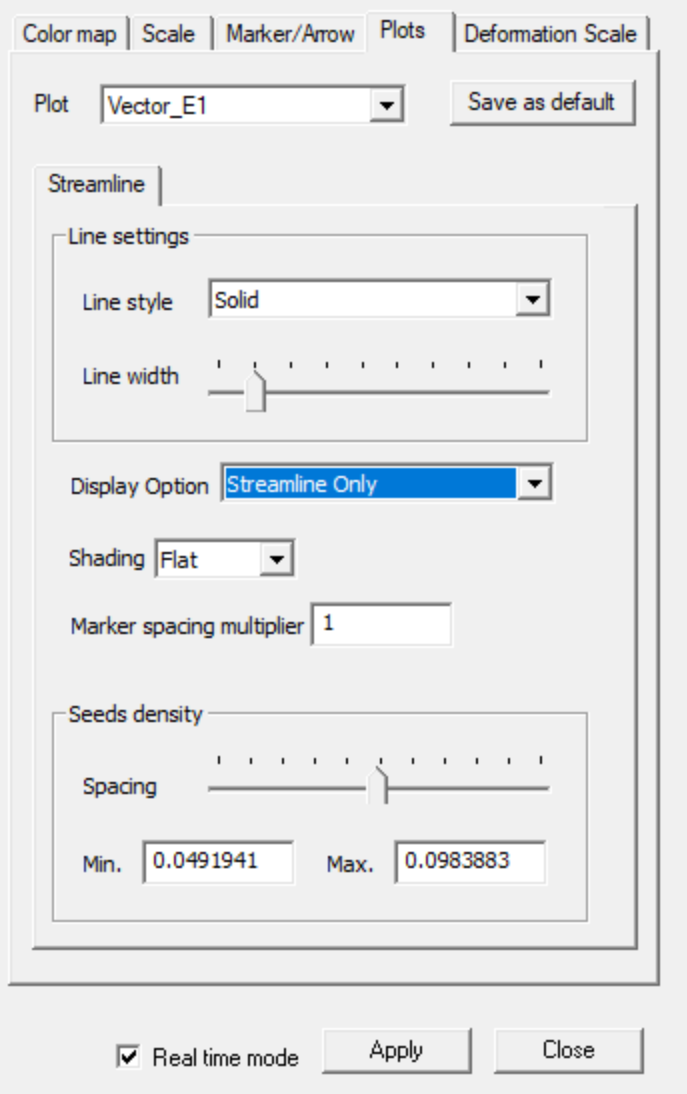


The **Reset** button resets a good initial spacing (and thus Min, Max spacing range) without the need to recreate the field plot.

- **Streamline Plots (if streamline is checked when you create the plot):**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **Plot** selected.



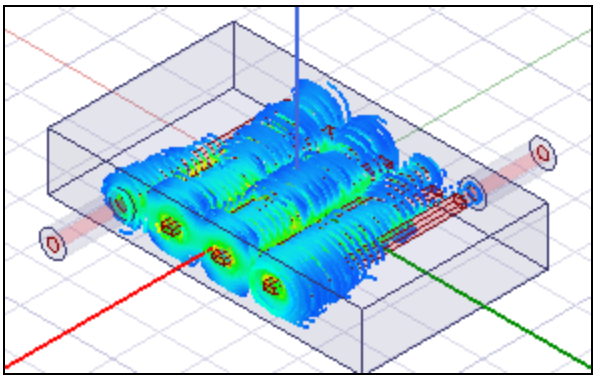
- The **Line style** as *solid* or *cylinder* from drop-down menu.
- **Line width**, specified using a slider.
- Display Option for whether to show Streamline Only, *Marker Only* or *Streamline and Marker*.
- Whether to use Shading, if lighting is turned on. By default, the **Shading** is set to *None* which is equivalent to "Do not use lighting." Shading can be Flat or Smooth.
- **None** Equivalent to selecting **Do not use lighting** on the Lighting Properties dialog box.

  Flat Provides a flat rendering of the field plot.

  Smooth Provides a smooth rendering of the field plot.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Seeds density** spacing. This affects the number of stream lines used to represent the quantity in the plot. Moving the slider to the left decreases the spacing and increases the number of stream lines. Moving the slider to the right increases the spacing and decreases the number of lines used to represent the quantity.
- **Min**. and **Max**. values represented (in *Magnitude filtering* settings).



a. Under each tab, click **Save as default** if you want the tab's settings to apply to field overlay plots created after this point.
b. Select **Real time mode** if you want the changes to take effect immediately in the view window.
c. If this option is cleared, click **Apply** when you want to see the changes.

Modifying Field Plot Colors

1. Click **Maxwell** > **Fields**> **Modify Plot Attributes** or, in the Project Manager, right-click **Field Overlays** and select **Modify Plots** from the short-cut menu (or use the "m" hotkey).

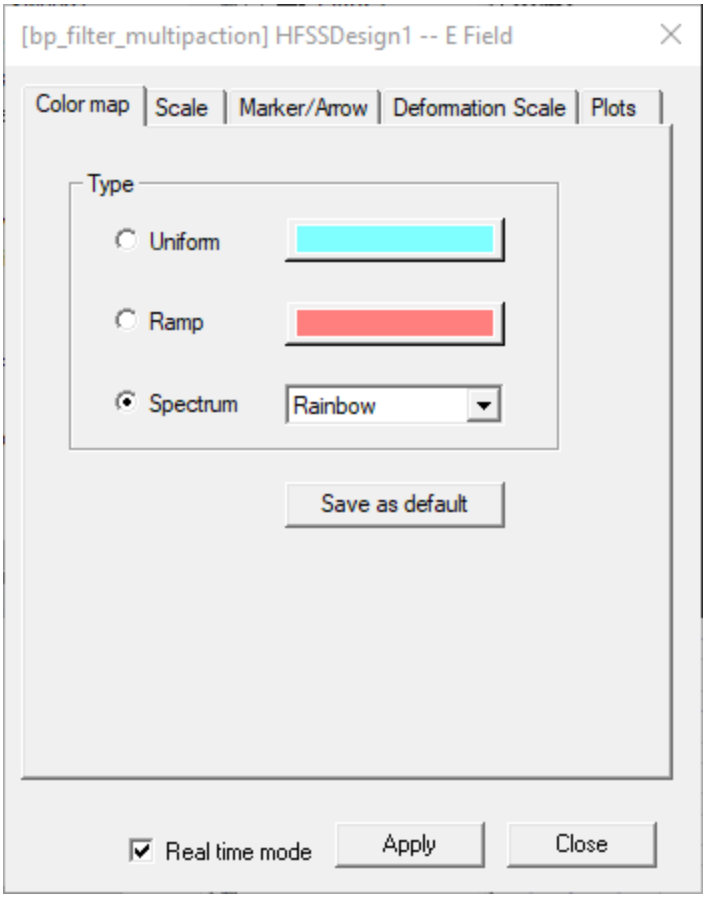
   The **Select Plot Folder** dialog box appears.

2. Select the plot folder you want to modify and click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.

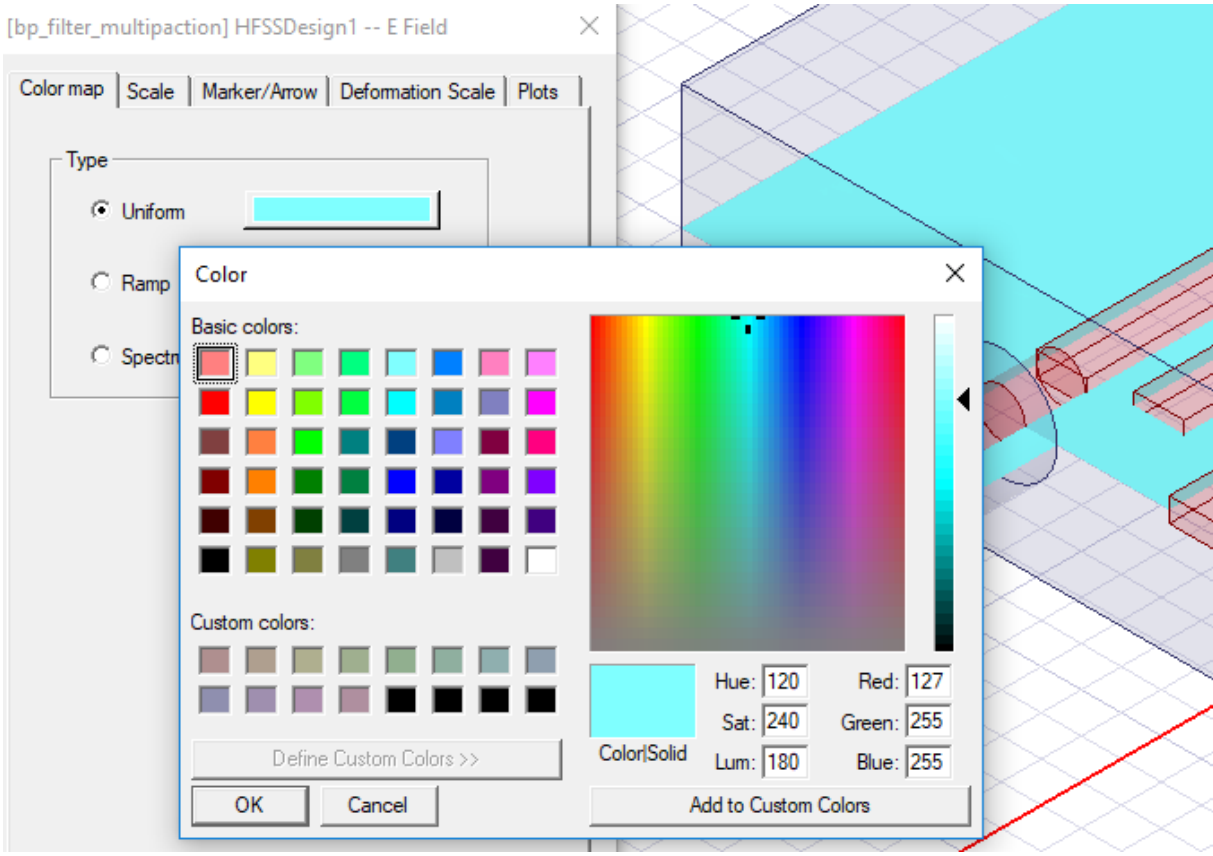
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Click the **Color Map** tab.
4. Select one of the following color types:

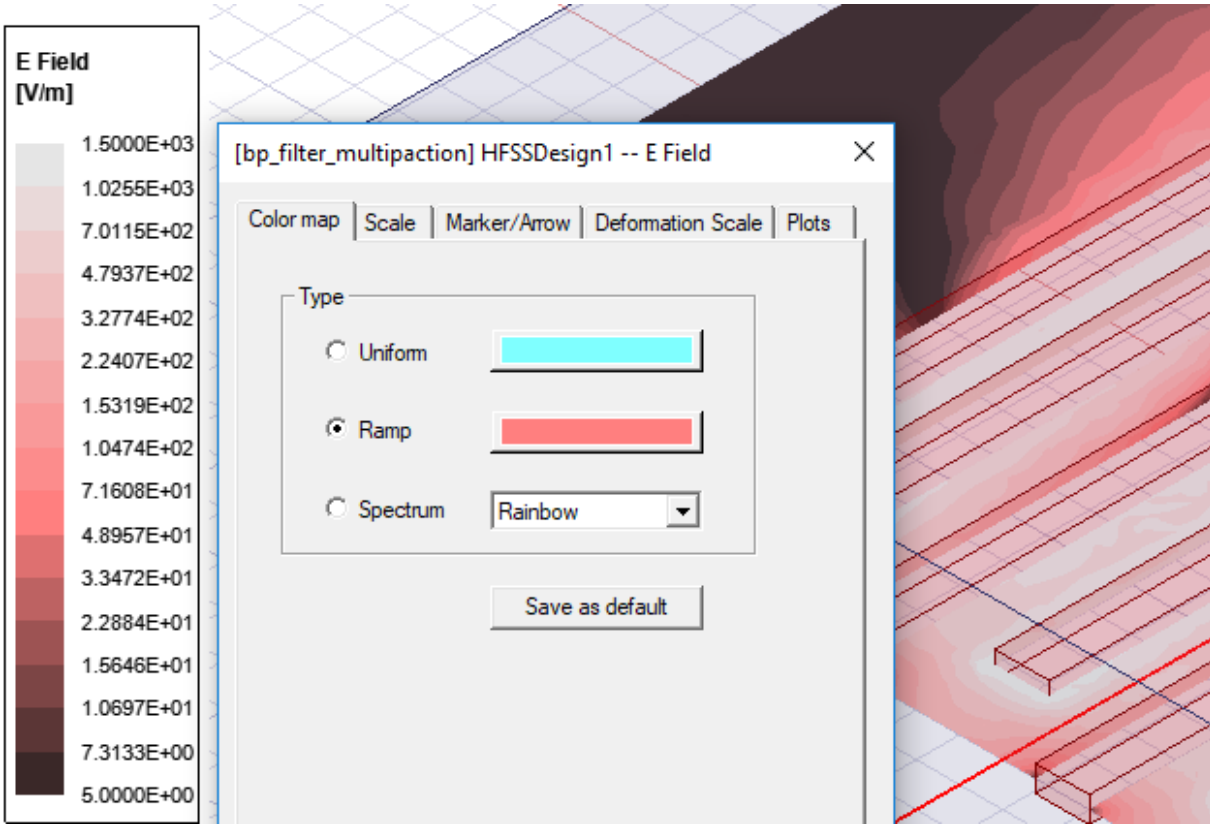
Uniform Field quantities are plotted in a single color. Click the button to choose the plot color from the **Color** button palette.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



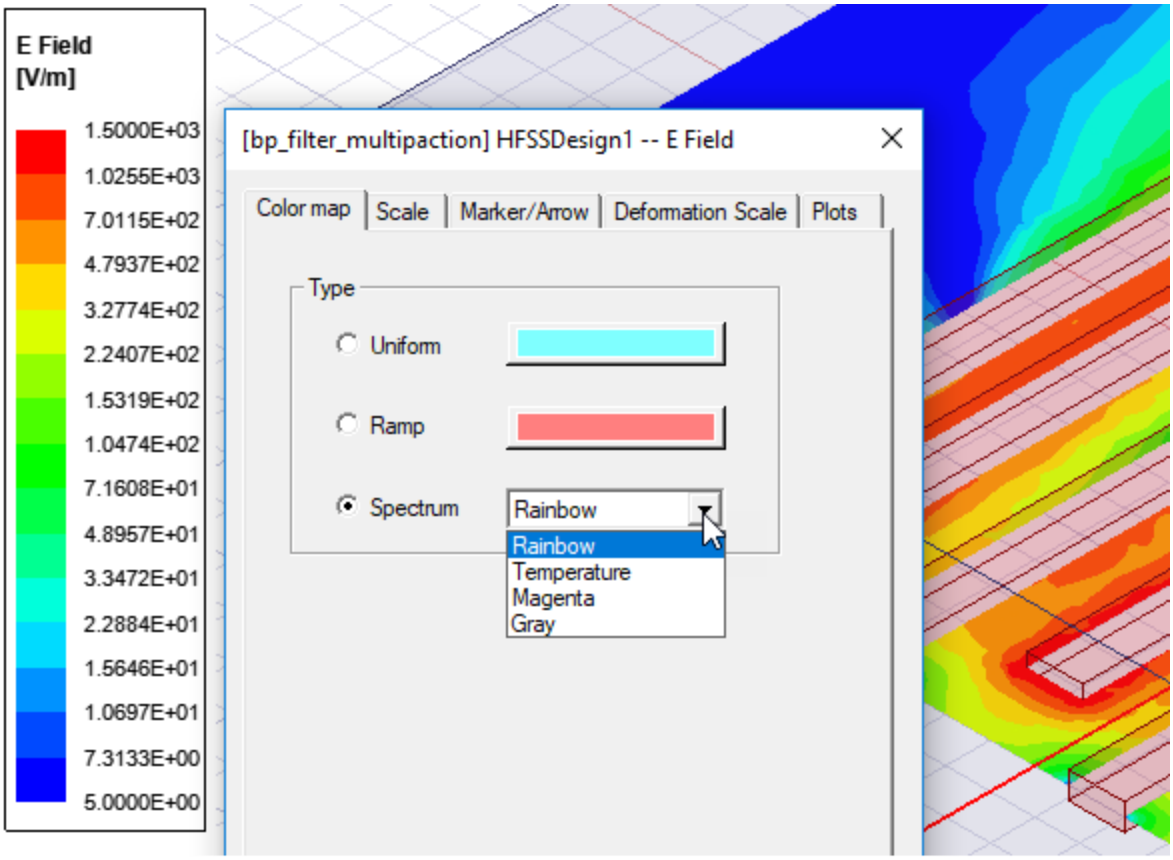
Ramp Field quantities are plotted in shades of a single color. Choose the plot color from the **Color** palette. The shade of the color corresponds to its field value.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Spectrum Field quantities are plotted in multiple colors. Choose a color spectrum from the drop-down menu. The values are Rainbow, Temperature, Magenta, and Grey. Each field value is assigned a color from the selected spectrum.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can choose **Save as Default**, if you want to use the current settings.

Select **Real time mode** if you want these, or subsequent changes to take effect immediately in the view window.

If this option is cleared, click **Apply** when you want to see the changes.

5. Click the **Scale Tab**.
6. In the **Num. Divisions** field, enter the number of colors to use in the plot.

   You can choose **Save as Default**, if you want to use the current settings.

   Select **Real time mode** if you want the changes to take effect immediately in the view window.

   If this option is cleared, click **Apply** when you want to see the changes.
7. Click **Close** to dismiss the dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color Key Settings

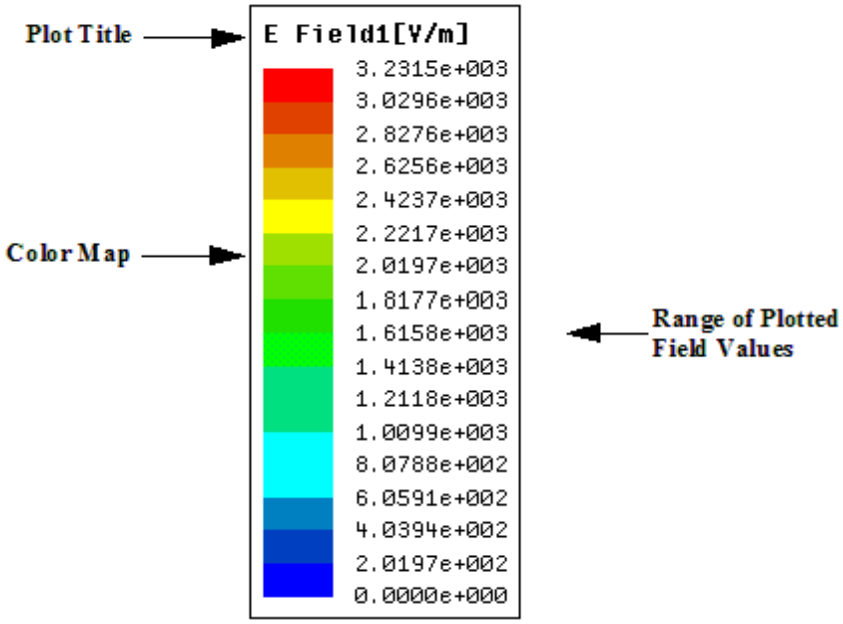
The color key (shown below) displays the range of plotted field values for a field overlay plot. It displays the colors that correspond to the range of field values on the plot. You can click and drag the color key to move it. In the case of no field data, the color key does not appear.

1. Click **View**> **Visibility**> **Active View Visibility**.

   The **Active View Visibility** dialog box appears.

2. Click the **Color Keys** tab.
3. In the **Visibility** column, select the field overlay or mesh plots in which you want to display the color key. Clear the plots in which you want to hide the color key from view.
4. Click **Done** to dismiss the dialog box.

Alternatively, to hide the color key, right-click on the color key in the active view window and choose **Hide** from the shortcut menu.



Color keys will only be visible for the plots that are selected in the **Active View Visibility** dialog box. If a hidden color key is shared by two or more overlay plots (such as a Heat Flux key, which is shared by heat flux magnitude and heat flux vector plots), then the color key will be hidden in all plots that share it. The **Active View Visibility** dialog box only lists a single incidence of a particular key, regardless of how many overlay plots share it. Use this dialog box to restore visibility of hidden color keys.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modifying the Field Plot Scale

To change how field quantities are scaled on the field overlay plot:

1. Click **Maxwell** > **Fields** > **Modify Plot Attributes** or, in the Project Manager, right-click Field Overlays and select **Modify Plots** from the shortcut menu (or use the "**m**" hotkey).

   The **Select Plot Folder** dialog box appears.

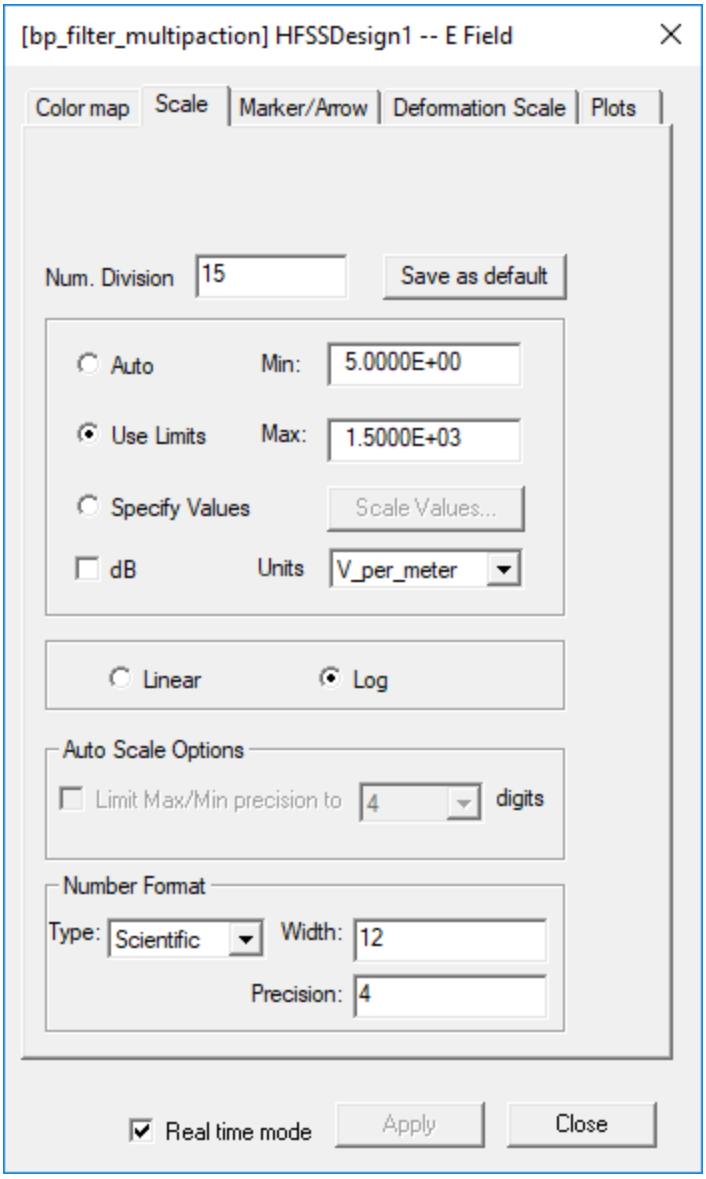
2. Select the plot folder you want to modify and click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.

3. Click the **Scale** tab.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Optionally, to change the number of divisions in the field plot scale, set the **Num. Division** field to a new value. You can click **Save as Default**, if desired.
5. Select one of the following scale options:

Auto The full range of field values will be plotted on the selected surface or volume. Selecting **Auto** enables the **Auto Scale Options** and disables the **Min** and **Max** fields. By default, precision is not limited and auto-min is the actual computed min on the plotted geometry.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use Limits	Only the field values between the minimum and maximum values will be plotted. Field values below or above these values will be plotted in the colors assigned to the minimum or maximum limits, respectively. Selecting **Use Limits** enables the **Min** and **Max** fields and disables the **Auto Scale Options**. Field values have a precision of at most 6 decimal places (field solution files are saved in floating precision), so Min/Max numbers are displayed to this precision.
Specify Values	This enables a **Scale Values** button.

Optionally, when dB is checked, dB scale is used for the plot. It disables the Units field and the **Linear** is selected.

6. Optionally, use the **Units** drop-down menu to select the default unit of measure for the plot.

   The units specified here appear on the Color map for the fields plot, and for the properties dialog for the field quantities.

7. If you selected **Use Limits**, enter the lowest field value to be plotted in the **Min.** text box and the highest field value to be plotted in the **Max.** text box.

   If you selected **Auto** or **Use Limits** without having dB checked, the **Auto Scale Options** are enabled. You should only changed for cases where auto-min is a small number. Use the 'Limits Max/Min precision to' check box to enable setting the drop-down menu for the precision limit.

   The auto-min is the greater of the following:

   - Actual computed Min
   - Max/pow(10, num digits of field precision)

   If you selected Specify Values, you can click the **Scale Values** button. This opens a dialog with an editable, scrollable list of the current scale values. To apply the changes you make, click **OK**. To close the dialog box without making changes, click **Cancel**.

8. If you selected **Auto** or use **Limits** or **dB**, you can select one of the following options:

Linear	Field values are plotted on a linear scale.
Log	Field values are plotted on a logarithmic scale. If field plots have negative and positive values and when auto-scale is selected, the log-scale choice automatically sets the Min value as the Max/Min Ratio. (If field plots have all negative values, **Log** is not allowed.)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. Specify the Number Format for the plot as Auto, Scientific Notation, or Decimal. You can also specify Width and Precision for the plot.
10. Select **Real time mode** if you want the changes to take effect immediately in the view window.

    If this option is cleared, click **Apply** when you want to see the changes.
11. Optionally, you can use the **Save As Default** button to save the following to the registry:
    - Whether to limit field precision
    - The number of digits of field precision
    - Whether to use log/linear scale
    - Whether to use dB

    Auto scale is the default for new plots. For scalar-in-volume plots, iso-surface (rather than cloud) is the default display
12. Click **Close** to dismiss the dialog box.

Modifying Vector Field Plot Arrows

To change the appearance of a vector field plot's arrows:

1. Click **Maxwell 3D** or **Maxwell 2D** and select **Fields** > **Modify Plot Attributes** .

   The **Select Plot Folder** window appears.
2. Select the plot folder you want to modify, and then click **OK**.

   > **Note:** All plots in the selected folder are modified.

   A dialog box with attribute settings for the selected folder appears.
3. Click the **Marker/Arrow** tab.
4. Under **Arrow options**, select one of the following arrow types from the **Type** drop-down menu:

Line	The arrows are displayed as 2D/flat.
Cylinder	The arrow tails are displayed as cylinders. The arrowheads are displayed as 3D/round.
Umbrella	The arrow tails are displayed as 1D lines. The arrowheads are displayed as 3D/round.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Use the **Size** slider to increase (move to the right) or decrease (move to the left) the length and dimensions of the arrows. The arrows are resized relative to the size of the model geometry.
6. Select **Map size** to scale the size of the arrows to the magnitude of the field quantity being plotted.
7. Select **Arrow tail** to include tails on all arrows.
8. Click the **Plots** tab.

   > **Note:** Maxwell plots arrows on a grid that is superimposed on the surface or object you selected for the plot.

9. Under **Vector plot**, use the **Spacing** slider to increase (move to the right) or decrease (move to the left) the distance between arrows (grid points.)
10. If you want the arrows to be spaced equally, select **Uniform**, and enter values in the Min and Max boxes.
11. Select **Real time mode** if you want the changes to take effect immediately in the view window. If this option is cleared, click **Apply** when you want to see the changes.
12. Click **Apply**, and then click **Close** to dismiss the window.

Setting the Mesh Visibility on Field Plots

To display or hide the mesh on field plots or to change the mesh's color:

1. Click **Maxwell 3D** or **Maxwell 2D** and select **Fields** > **Modify Plot Attributes** .

   The **Select Plot Folder** window appears.

2. Select the plot folder you want to modify, and then click **OK**.

   A dialog box with attribute settings for the selected folder appears.

   > **Note:** All plots in the selected folder are modified.

3. Click the **Plots** tab.
4. Select **Add grid** to display the mesh.
5. Optionally, select a color for the mesh from the **Color** palette.
6. Select **Real time mode** if you want the changes to take effect immediately in the view window.

   If this option is cleared, click **Apply** when you want to see the changes.

7. Click **Apply**, and then click **Close**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modifying Scalar Field Plot Isovalues

1. Click **Maxwell** > **Fields** > **Modify Plot Attributes**.

   The **Select Plot Folder** window appears.

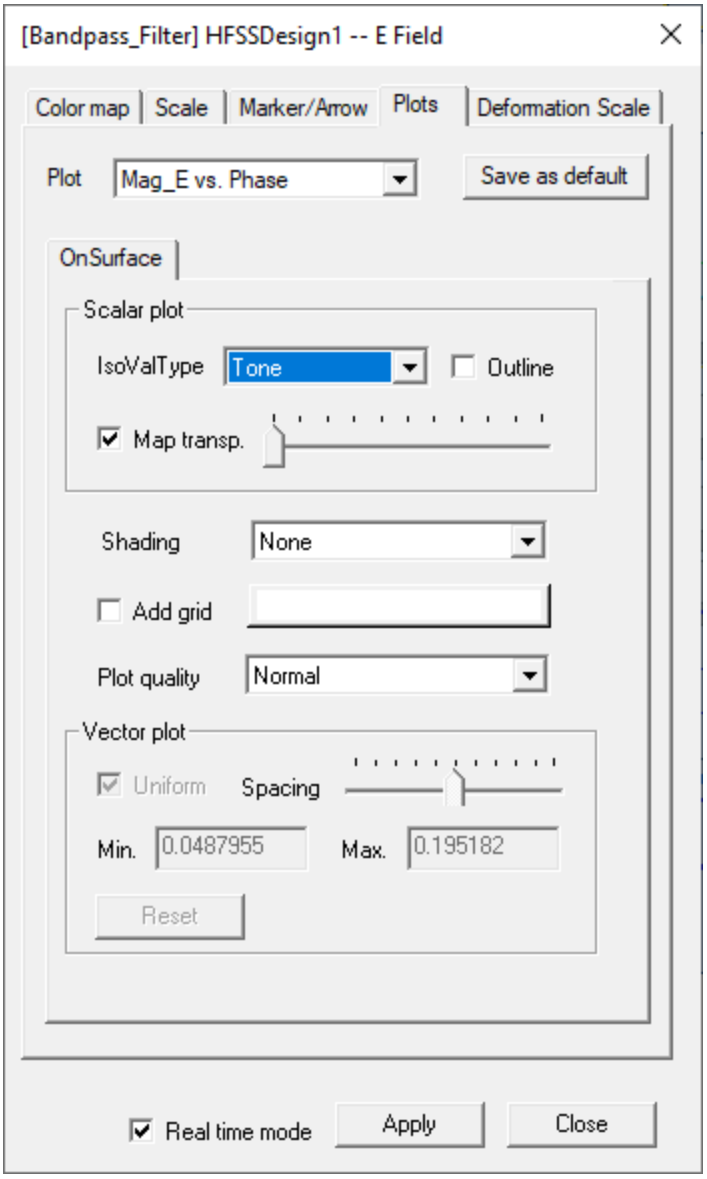
2. Select the plot folder you want to modify, and then click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.

3. Click the **Plots** tab. If you have more than one plot, a drop- down menu lets you select the desired plot. The **OnSurface** tab appears for surface plots and **InVolume** for volume plots.

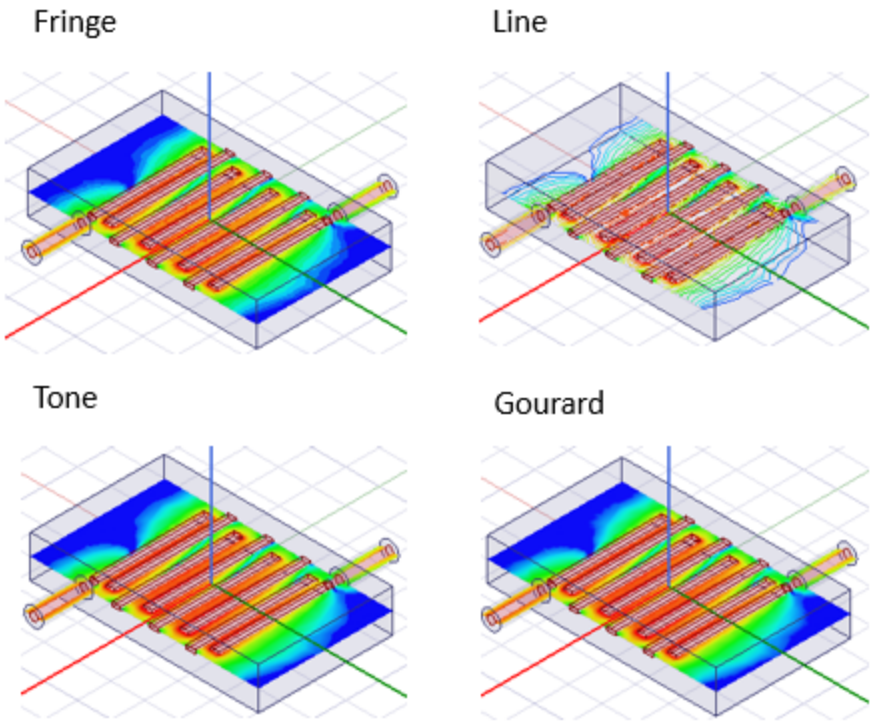
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. If the plot is a scalar surface plot, do the following:
   a. Select one of the following isosurface display types in the **IsoValType** drop-down menu:

Line	Lines are drawn along the isovalues.
Fringe	Color is constant between isovalues.
Tone	Color varies continuously between isovalues.
Gourard	Color varies continuously across the plot.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



b. Optionally, if you selected **Fringe** or **Tone**, select **Outline** to add a border line between isovalues.

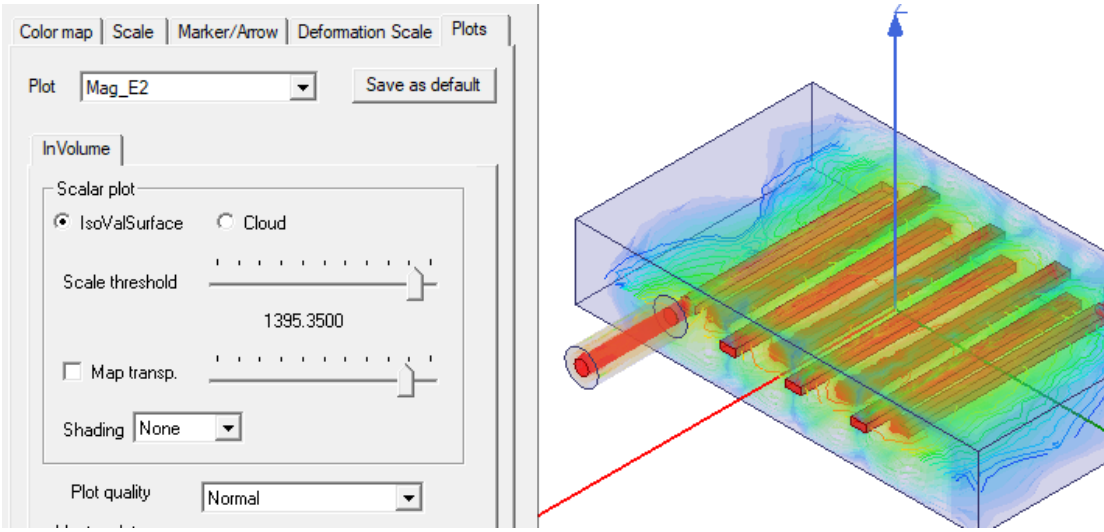
c. For shading on OnSurface plots, if you are using Lighting, you can adjust the Shading as None, Flat, or Smooth.

5. If the plot is a scalar volume plot, do the following:

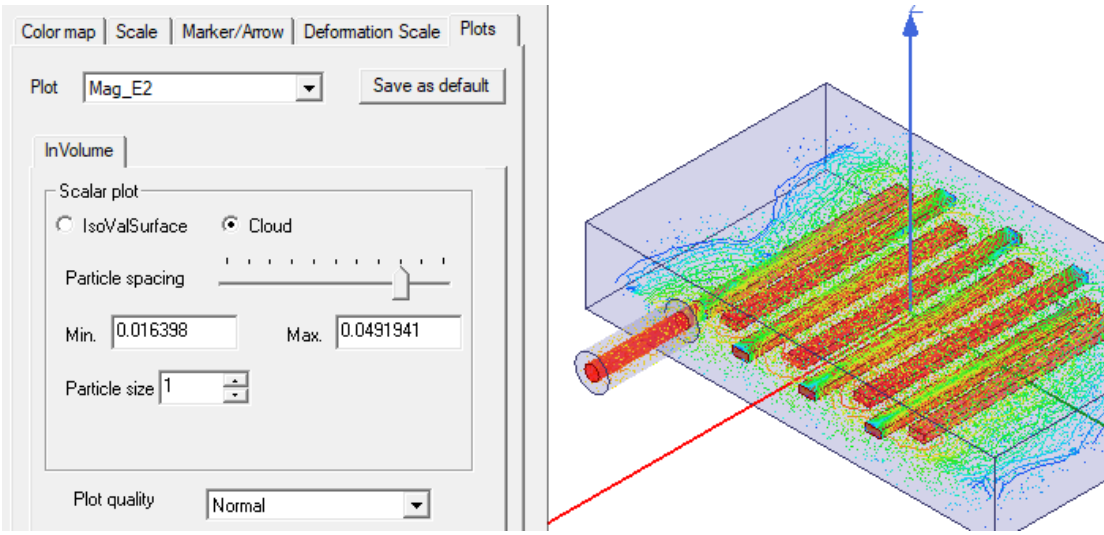
a. Select one of the following display types:

IsoValSurface	Color is drawn on the isovalues. This choice includes “Scale threshold” slider barthat allows you to set a field value threshold so that iso surfaces with field value lower than the given threshold will show as translucent with given transparency specified by the slider bar below it. When "Map transp." is checked, translucent iso surfaces field values map to [0.0, transparency] so lower field value map to higher transparency while higher field value to lower transparency. If it is unchecked, all translucent iso surfaces will have the same given transparency from the slider bar.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Cloud Field values are represented by points that illustrate the spatial distribution of the solution. The higher the solution value, the greater the cloud density. You can adjust particle spacing, Min. and Max, values, and Particle size.



6. Select **Real time mode** if you want the changes to take effect immediately in the view window.

   If this option is cleared, click **Apply** when you want to see the changes.

7. Click **Close** to dismiss the window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modifying Scalar Field Plot Shading

1. Click **Maxwell** > **Fields** > **Modify Plot Attributes**.

   The **Select Plot Folder** window appears.

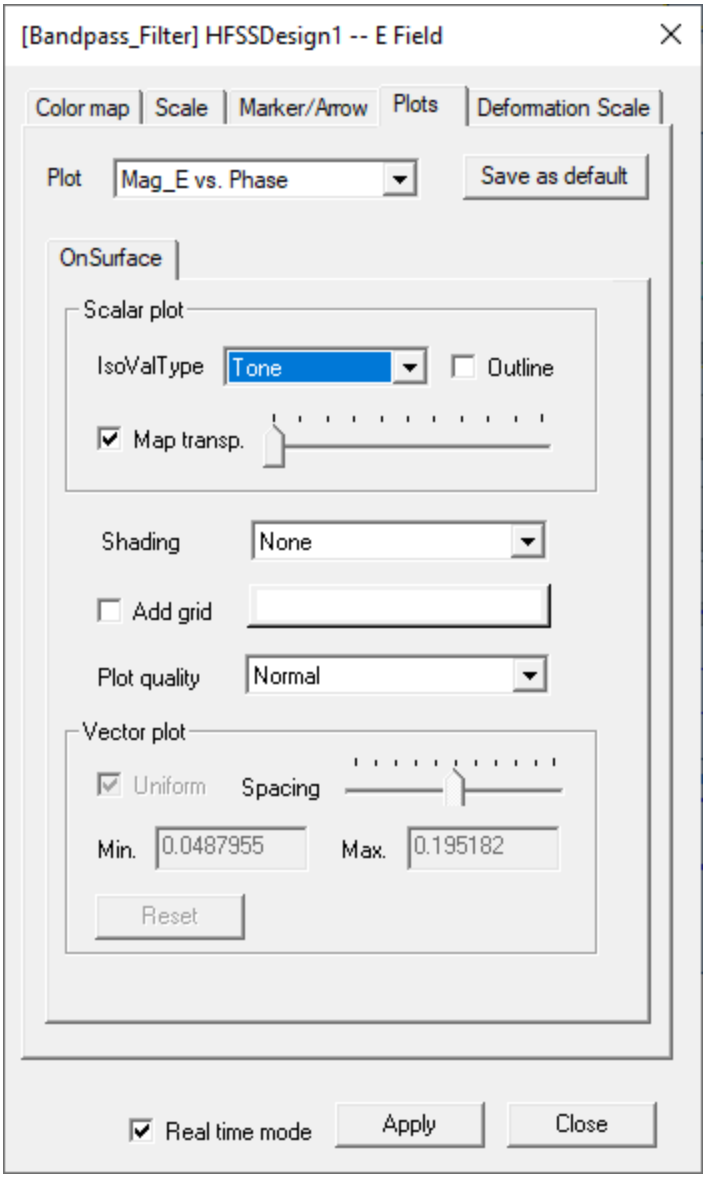
2. Select the plot folder you want to modify, and then click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.

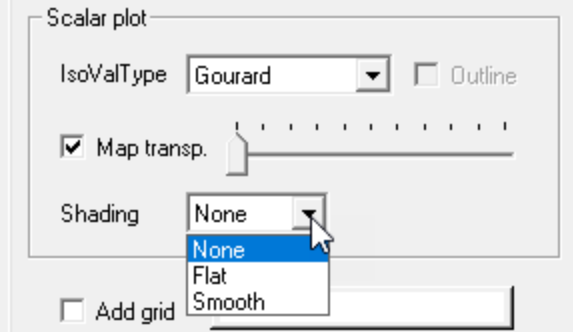
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click the **Plots** tab. Select the Plot to modify.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. If the plot is a scalar surface plot, do the following:
    a. Select one of the following from the Shading drop-down menu:



None	Equivalent to selecting **Do not use lighting** on the Lighting Properties dialog box.
Flat	Provides a flat rendering of the field plot.
Smooth	Provides a smooth rendering of the field plot.

5. Select **Real time mode** if you want the changes to take effect immediately in the view window.

    If this option is cleared, click **Apply** when you want to see the changes.

6. Click **Close** to dismiss the window.

Mapping Scalar Field Plot Transparency to Field Values

1. Click **Maxwell** > **Fields**> **Modify Plot Attributes**.

    The *Select Plot Folder* dialog box appears.

2. Select the plot folder you want to modify, and then click **OK**.

    All plots in the selected folder will be modified.

    A dialog box with attribute settings for the selected folder appears.

3. Click the **Plots** tab.
4. Use the **Map transp.** slider to increase (move to the right) or decrease (move to the left) the transparency of the plot.
    - If you select **Map transp.**, the transparency of field values increases as the solution values decrease.
5. Select **Real time mode** if you want the changes to take effect immediately in the view window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If this option is cleared, click **Apply** when you want to see the changes.

6. Click **Close** to dismiss the dialog box.

Modifying Markers on Point Plots

For scalar point plots, a marker is used to represent a field quantity at a selected point (for vector point plots, arrows are used).

Modify the shape and size of markers in the plot attributes window:

1. Click **Maxwell**> **Fields**> **Modify Plot Attributes**.

   The **Select Plot Folder** window appears.

2. Select the plot folder you want to modify, and then click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.

3. Click the **Marker/Arrow** tab in the plot attributes window.
4. Under **Marker options**, select one of the marker types to represent the field quantity at the point:
   - **Sphere**
   - **Box**
   - **Tetrahedron**
   - **Octahedron**
5. Use the **Size** slider to increase or decrease the size of the marker.
6. Select **Map size** to scale the size of the marker to the magnitude of the quantity being plotted.
7. Select **Real time mode** if you want the changes to take effect immediately in the view window.

   If this option is cleared, click **Apply** when you want to see the changes.

8. Click **Close**.

Viewing Data Markers

The difference between two markers can be viewed on a plot.

1. Click **Report2D** > **Data Marker**.
2. Click the first point on the graph.
3. Hold and drag the mouse and release at the second point.

Under the XY plot, you can see the difference between the two marker points.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modifying Line Plots

Field quantities can be plotted directly on a line object. Scalar quantities are plotted as 3D color-shaded lines. Vector quantities are plotted as arrows that are based on the line.

To modify the appearance of line plots:

1. Click **Maxwell** > **Fields** > **Modify Plot Attributes** .

   The **Select Plot Folder** window appears.

2. Select the plot folder you want to modify, and then click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.

3. Click the **Plots** tab.
4. Select one of the following isosurface display types in the **IsoValType** drop-down menu:

Fringe	Color is constant between isovalues.
Tone	Color varies continuously between isovalues.
Gourard	Color varies continuously across the plot.

5. Select one of the following styles for the line object in the **Line style** drop-down menu:

Cylinder	The line object is shaped like a cylinder.
Solid	The line object is a 3D solid.
Dash-Dash	The line object is represented by dashed black line segments.
Dot-Dot	The line object is represented by a series of dots.
Dash-Dot	The line object is represented by a series of alternating dashed black line segments and dots.

6. Use the **Line width** slider to increase (move to the right) or decrease (move to the left) the thickness of the line.
7. By default, a polyline object is divided into 100 equally spaced points for post processing. To modify the number of points on the line, type a new value in the **Number of points** text box.
8. Select **Real time mode** if you want the changes to take effect immediately in the view window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If this option is cleared, click **Apply** when you want to see the changes.

9. Click **Close** to dismiss the window.

Setting a Plot's Visibility

To display or hide a field overlay or mesh plot from view in the **3D Modeler** window:

1. Click **View** > **Visibility** > **Active View Visibility**.

   Alternatively, you can select the **Active View Visibility** icon on the **View** ribbon.

   The **Active View Visibility** dialog box appears.

2. Click the **FieldsReporter** tab.
3. In the **Visibility** column, select the field overlay or mesh plots you want to display. Clear the plots you want to hide from view.
4. Click **Done**.

Only the selected plots are visible.

Saving a Field Overlay Plot

Field overlay and mesh plots are saved in the project file; however, you can save a plot to Field Plot File format (.dsp) and then open it in Maxwell.

To save field overlay or mesh plot data to a .dsp file:

1. In the project tree, click the plot you want to export.
2. Click **Maxwell** > **Fields** > **Save as** .

   The **Select Field Plot(s)** dialog box appears.

3. Select the plots you want to export by checking the **Select** box, and then click **OK**.

   The file browser appears. **Field Plot Files (.dsp)** is the selected file type.

4. Specify the name of the .dsp file and the location in which to save it.
5. Click **Save**.

The plot is exported to the specified .dsp file.

To open the file you created, click **Maxwell** > **Fields** > **Open**.

Opening a Field Overlay Plot

To open a field overlay or mesh plot that you have saved to Field Plot File format (.dsp):

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click Maxwell > **Fields** > **Open** .
2. The file browser appears. **Field Plot Files (.dsp)** is the selected file type.
3. Browse to the location of the .dsp file you want to open, and then click the file name.
4. Click **Open**.

The plot appears in the view window. It is listed under **Field Overlays** in the project tree.

Deleting a Field Overlay Plot

1. Click **Maxwell** > **Fields** > **Delete Plot** .

   The **Delete Plots** dialog box appears.
2. Select the plots you want to delete by checking the **Delete** check box.
3. Click **OK**. The selected plot are deleted.

Alternatively, click the plot in the project tree that you want to delete, and then press **Delete** .

Setting Field Overlay Plot Defaults

Each new field plot uses the default plot settings specified in the **Set Plot Defaults** dialog box.

To modify the default plot settings:

1. If a plot folder has not been created, click **Field Overlays** in the Project Manager.
2. Click **Maxwell** > **Fields** > **Set Plot Defaults**.

   The **Set Plot Defaults** dialog box appears.
3. Select the solution to plot from the **Solution** drop-down menu.
4. Select the plot folder in which new plots will be stored from the **Quantity type** drop-down menu. Choose one of the following options:

New Folder	Each new plot will be stored in a separate folder in the project tree.
Automatic	Each new plot will be stored in a folder determined by Maxwell as the most appropriate based on the plotted field quantity. For example, all surface magnitude E plots will be stored in the same folder.
An existing folder	Select the existing folder in which you want to store new plots.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Plots stored in the same folder will use the same color key. The **Auto** scale setting will be based on the maximum field solution value present in a plot.

5. Under **Intrinsic Variables**, specify the frequency and phase angle at which the field quantity is evaluated.
6. Click **OK**.

Renaming a Plot

To rename a plot, do the following in the Project Manager tree:

1. Under **Field Overlays**, expand the field type, and right-click the specific plot you want to rename.

   A shortcut menu appears.

2. Select **Rename**.

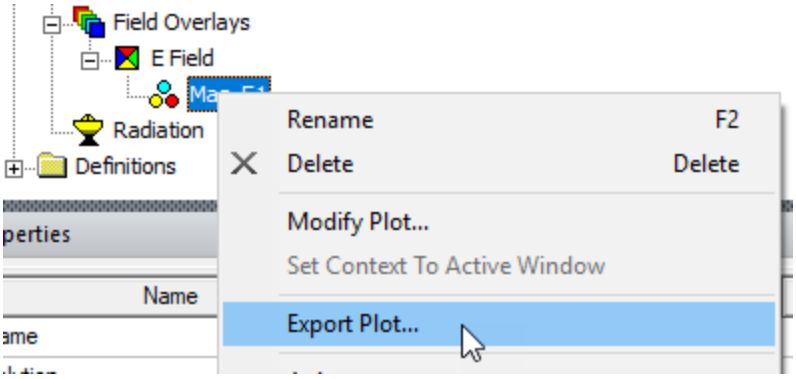
   The text becomes editable in the Project Manager tree.

3. Type the new name, and press **Enter**.

Exporting Field Plots

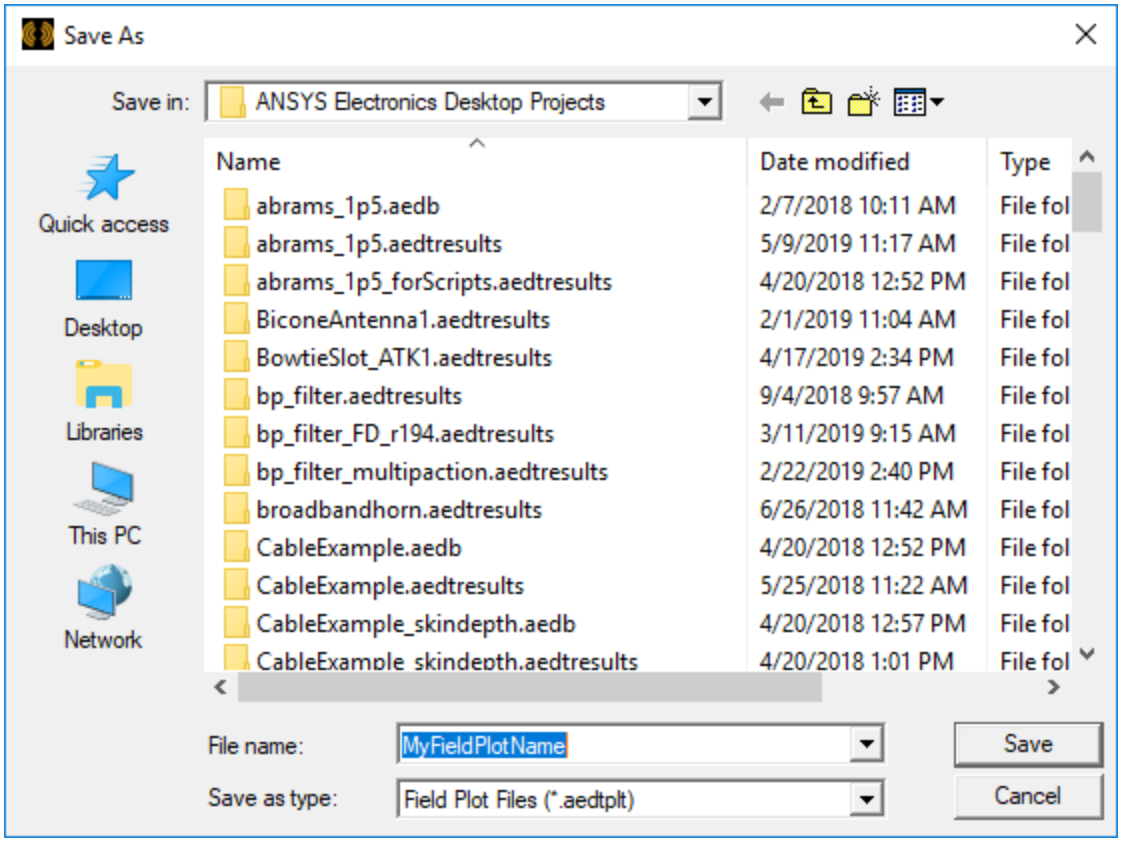
The **Export Plot...** command lets you export a field plot to a format that supports full data precision without the need to go through the Fields Calculator.

1. Create the field plot you want to export.
2. In the **Project Manager tree**, right click on the field plot icon and in the shortcut menu, click **Export Plot...**.

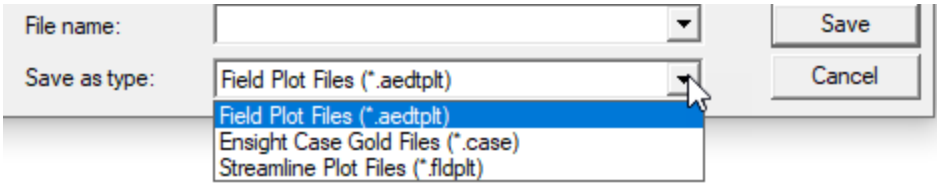


The **Save As** dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

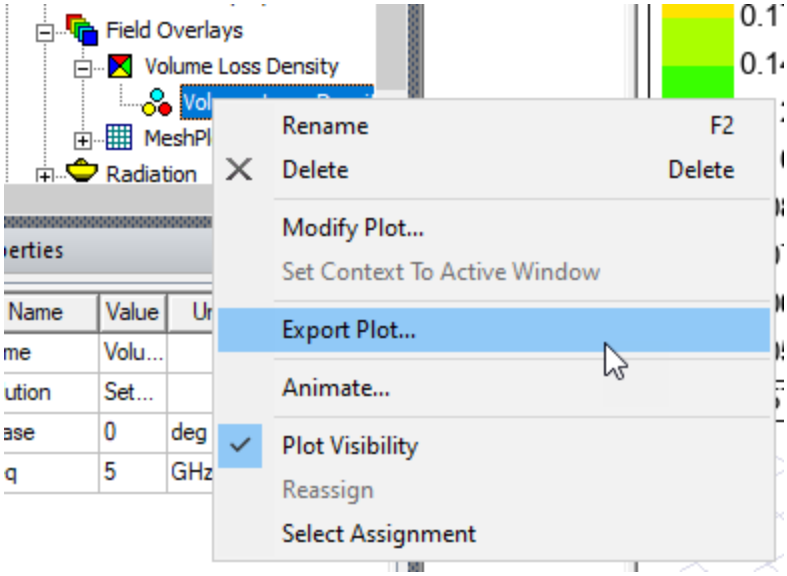


3. Specify the directory to **Save in**, the **File name** and use the **Save as type** as Field Plot Files (*.aedtplot) or Ensight Case Gold Files (*.case). If the Plot is a Streamline plot you can also choose Steamline files (*.fldplt). (See **Notes for creating Streamline plots** under Plotting Field Overlays).

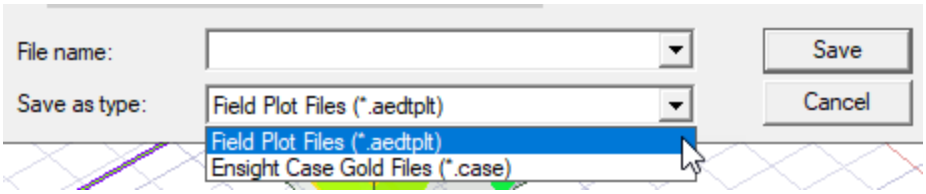


You can export a Streamline plot in .fldplt format by right-clicking on the plot in the Project tree, and selecting **Export Plot...**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This opens a dialog for you to specify a plot name and location.



The Field Plot is exported to the file format you specified.

Field Plot Data Export Format (*.aedtplt)

A sample first order file with 1 drawing:

```
# Ansys ElectronicsDesktop 2025.2.0
# Field plot export file (*.aedtplt), version 1.0
Number of drawing: 1
$begin Drawing_1
IsUniformMesh=true
HasCurvElem=false
BoundingBox(-1.2999999523162842e+00, 1.0000000000000000e+00, -
8.0000001192092896e-01, 1.3999999761581421e+00, -6.9999998807907104e-
01, 0.0000000000000000e+00)
Elements(27, 6, 4, 3, 3, 3, 10, 5, 23, 6, 24, 26, 7, 18, 19, 20, 3,
4, 3, 3, 3, 10, 5, 24, 7, 25, 27, 8, 18, 20, 21, 3, 4, 3, 3, 3, 10,
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
2, 14, 4, 16, 22, 7, 13, 17, 20, 3, 4, 3, 3, 3, 10, 7, 26, 6, 16, 15,
2, 20, 19, 13, 3, 4, 3, 3, 3, 10, 1, 9, 2, 12, 15, 6, 10, 13, 19, 3,
4, 3, 3, 3, 10, 6, 23, 5, 12, 11, 1, 19, 18, 10, 3)
Nodes(1.0000000000000000e+00, 1.4000000000000001e+00, -
7.0000000000000007e-01, 1.0000000000000000e+00, -8.0000000000000004e-
01, -7.0000000000000007e-01, -1.3000000000000003e+00,
1.4000000000000001e+00, -7.0000000000000007e-01, -
1.3000000000000003e+00, -8.0000000000000004e-01, -
7.0000000000000007e-01, 1.0000000000000000e+00,
1.4000000000000001e+00, 0.0000000000000000e+00,
1.0000000000000000e+00, -8.0000000000000004e-01,
0.0000000000000000e+00, -1.3000000000000003e+00, -
8.0000000000000004e-01, 0.0000000000000000e+00, -
1.3000000000000003e+00, 1.4000000000000001e+00,
0.0000000000000000e+00, 1.0000000000000000e+00, 3.0000000000000004e-
01, -7.0000000000000007e-01, -1.5000000000000013e-01,
1.4000000000000001e+00, -7.0000000000000007e-01,
1.0000000000000000e+00, 1.4000000000000001e+00, -3.5000000000000003e-
01, 1.0000000000000000e+00, 3.0000000000000004e-01, -
3.5000000000000003e-01, -1.5000000000000013e-01, 3.0000000000000004e-
01, -7.0000000000000007e-01, -1.5000000000000013e-01, -
8.0000000000000004e-01, -7.0000000000000007e-01,
1.0000000000000000e+00, -8.0000000000000004e-01, -
3.5000000000000003e-01, -1.5000000000000013e-01, -
8.0000000000000004e-01, -3.5000000000000003e-01, -
1.3000000000000003e+00, 3.0000000000000004e-01, -7.0000000000000007e-
01, -1.5000000000000013e-01, 1.4000000000000001e+00, -
3.5000000000000003e-01, -1.5000000000000013e-01, 3.0000000000000004e-
01, -3.5000000000000003e-01, -1.3000000000000003e+00,
3.0000000000000004e-01, -3.5000000000000003e-01, -
1.3000000000000003e+00, 1.4000000000000001e+00, -3.5000000000000003e-
01, -1.3000000000000003e+00, -8.0000000000000004e-01, -
3.5000000000000003e-01, 1.0000000000000000e+00, 3.0000000000000004e-
01, 0.0000000000000000e+00, -1.5000000000000013e-01,
3.0000000000000004e-01, 0.0000000000000000e+00, -1.5000000000000013e-
01, 1.4000000000000001e+00, 0.0000000000000000e+00, -
1.5000000000000013e-01, -8.0000000000000004e-01,
0.0000000000000000e+00, -1.3000000000000003e+00, 3.0000000000000004e-
01, 0.0000000000000000e+00)
ElemSolution(5.7354731844611963e-02, 2.9232772984855881e-01, 10,
2.1352990786467962e-01, 1.2797657961936443e-01, 1.9660806325882133e-
01, 1.1726155545957916e-01, 2.1885451415722262e-01,
2.4247258543532729e-01, 1.8703186257772084e-01, 1.1712555190711266e-
01, 1.0306551944577172e-01, 1.9222144469761518e-01,
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
1.6615464797768376e-01, 1.0804043743316410e-01, 6.6510438035030331e-
02, 1.7865861636680735e-01, 1.1040223877077059e-01,
2.5391393137507334e-01, 1.4691691383403402e-01, 9.2573922668716196e-
02, 1.7035856643167813e-01, 1.4641167161072380e-01,
1.4770701527152347e-01, 1.6261065519093762e-01, 2.3063204305627905e-
01, 1.2530038704664462e-01, 1.4614642445195816e-01,
1.1920618653295692e-01, 1.1233254336705870e-01, 1.2156273418014953e-
01, 9.0145063831456124e-02, 8.3219042948130054e-02,
1.2535082887941645e-01, 1.2753078005131815e-01, 1.3872888237187334e-
01, 1.2424658874600869e-01, 1.3005844785230156e-01,
1.4711693546682481e-01, 1.0390046361183673e-01, 1.2075181459543853e-
01, 1.1420762153323573e-01, 1.0510569422539225e-01,
2.5244183990075453e-01, 1.4468010704771497e-01, 1.1075980053186024e-
01, 1.0756449329788385e-01, 7.1852801171335884e-02,
5.7354731844611963e-02, 1.9153549687571236e-01, 1.2155254289403770e-
01, 1.0467070945529336e-01, 1.8204372604099889e-01,
1.6179743545861430e-01, 1.4548511943318404e-01, 1.7334644308737701e-
01, 1.1138082367857982e-01, 1.9054730731722422e-01,
2.9232772984855881e-01, 1.1136332643725760e-01, 1.5631026982928142e-
01, 2.1419476339746449e-01, 1.7884651132070065e-01)
ElemSolutionMinMaxLocation(1.0000000000000000e-03, -
8.0000000000000004e-04, 0.0000000000000000e+00, 1.0000000000000000e-
03, 1.4000000000000002e-03, -7.0000000000000010e-04)
ElemSolutionMinMaxIndices(4, 5, 5, 5)
$end Drawing_1
```

File Format Description

The first line starts with the symbol of “#” and continues with the product name and its version that is used to produce this file. The 2nd line also starts with the symbol of “#” and continue with the file format version. Currently it is version 1.0.

The next line, it shows the number of drawings in this file. The above example shows 1 drawing. The drawing has two big data blocks (the mesh data block and field data block). Each drawing starts with **$begin Drawing_*** ** and ends with **$end Drawing_*** ** (where ** *** ** should be replaced with the drawing index, starting from 1).

Mesh data block

It contains all the mesh information about current drawing include element type, number of nodes of each element, node coordinates and element node connectivity. At the beginning of the block, it shows if it has curve element and then it lists the bounding box of this mesh. The bounding box format is **BoundingBox(lowerX, lowerY, lowerZ, upperX, upperY, upperZ)**. They are six double values.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Next comes with elements part starting with **Elements(**. First integer is the total number of nodes for this mesh and the next is the total number of elements. After that, it repeats the following information for each element: element type, reserved integer 1, reserved integer 2 and reserved integer 3, number of nodes for this element, node index 1, node index 2 till to the number of nodes for this element. All these numbers are integers. Please refer to the end of this document to see the list of element types. This part ends with the close paren, ")".

The last part of the mesh data block is the nodes part. It contains the coordinates(x,y,z) of all the nodes for this mesh. It starts with Nodes(and continue with a list of tuples of 3 double values which are the x, y, z coordinates of the first, 2nd, last nodes and it ends with a close paren,")".

Field data block

It contains two parts. The first part includes node field data of each element. Node field data could either be scalar or vector data which means either it has one component or it has three components(x, y, z).

This part starts with **ElemSolution(**, the next two double values are the minimal and maximal field value of this drawing. The next integer shows the total number field data for each element. It should be a multiple of the number of nodes of each element (e.g. if field data is vector data, then it should be 3 times of the number of nodes of each element). For vector data, each element should have data like (**node1_x, node1_y, node1_z, node2_x, node2_y, node2_z, …**). This part ends with close paren, ")".

The 2nd part shows two location points. It starts with **ElemSolutionMinMaxLocation(**. The first 3 double values are the (x,y,z) coordinates of the location where the minimal field value is located while the next 3 numbers for the maximal field value. This part ends with close paren, ")".

Element types:

Element type description	Element type value
point	0
line	1
triangle	2
quad	3
tetradedron	4
pyramid	5
wedge	6
hexahedron	7

For second order element node index convention, Lagrange Parabolic is used.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using the Fields Calculator

The Fields Calculator is a very powerful tool for postprocessing. It is available for HFSS, Maxwell, Q3D, Icepak, and Mechanical solutions. While standard postprocessed results (such as S-parameters, Y or Z matrix, animated field plots, and near and far field patterns) serve most simulation requirements, the Fields Calculator enables you to perform further computations using basic field quantities. The calculator computes derived quantities from the general electric, thermal, or displacement field solution; writes field quantities to files; locates maximum and minimum field values; and performs other operations on the field solution. Using this calculator, you can perform mathematical operations on all saved field data in the modeled geometry at a single frequency. The resulting quantities can be plotted, tabulated, or exported.

The Fields Calculator includes predefined expressions appropriate for each solver and lets you create and save additional named expressions.

> **Note:** In **Maxwell 2D**, the Fields Calculator accounts for the difference between XY and RZ models when handling data and results. For example, the integral command will perform an XY integral on the model assuming a 1 meter depth, while for an RZ model it will perform an integral on the 360 degree equivalent volume.

To view information on a command or screen area, click over the button or screen area on the illustration below.

At the top-left corner of the calculator is a list of **Named Expressions**, which are standard or user-defined field quantities that are accessible from outside of the calculator. They can be added, copied to stack, saved to, or loaded from a library file using the buttons right beneath the list.

At the lower-left corner of the calculator is the **Solution Context** section, in which you can select the desired solutions, field types, frequency, and phase for the current session.

The top right of the calculator contains the **Data Stack**, in which calculator entries are held in stack registers. The data type in the Data Stack is denoted by its prefix abbreviation.

Immediately beneath the stack is the row of **Stack Command** buttons that define some basic operations for the data in the Data Stack.

The bottom half of the calculator holds the columns containing the actual calculator buttons, organized into columns, classifying them by the type of operation and the type of data upon which the operation can be performed. These columns are headed Input, General, Scalar, Vector, and Output.

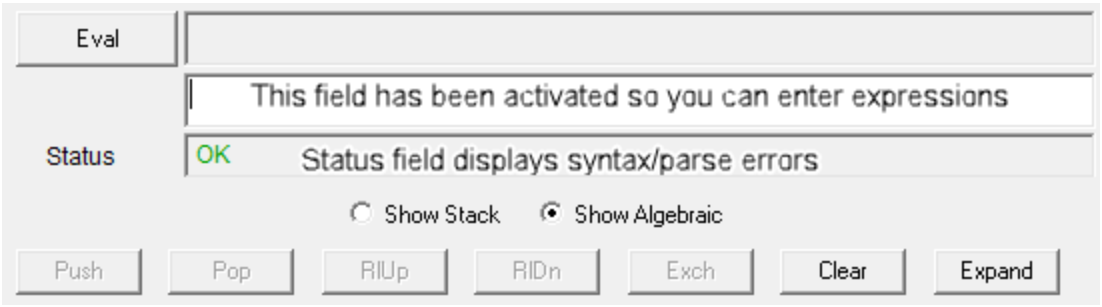
At the very bottom of the calculator is the button to exit, **Done**. The Fields Calculator interface is discussed in detail in subsequent sections.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The calculator does not perform the computations until a value is needed or is forced for a result. This makes it more efficient, saving computing resources and time; you can do all the calculations without regard to data storage of all the calculated points of the field. It is generally easier to do all the calculations first, then plot the results.

Support Expression Input in Fields Calculator

For Maxwell, you can create custom field quantities using algebraic expressions by activating the **Maxwell Support Expression Input in Fields Calculator** Beta option. When this beta feature is activated, the **Show Algebraic** option appears in the Fields Calculator window, and the evaluation field is activated so that you can enter expressions. For more information on this functionality, see Algebraic Mode.



Scripting Support

All calculator operations are fully scriptable. You can save the commands used in a Fields Calculator session by first clicking the **Tools** > **Record Script to File** menu and replay the same commands by clicking the **Tools** > **Run Script** menu in a later session.

Preliminary Considerations for Using the Fields Calculator

The following sections provides some consideration notes regarding use of the Postprocessor Field Calculator. Most of the statements below are fairly generalized, and may not apply to all 3D solver projects. When in doubt about the applicability of a particular consideration for a particular project, please feel free to contact your local HF Applications Engineer for further assistance.

Field Convergence and Accuracy

Maxwell use finite element method (FEM) field solvers, which arrive upon a solution via adaptive meshing convergence. There are different algorithms available for determining where in each given model mesh adaptation is performed, but convergence is always evaluated by comparison of S-parameters (for driven solutions), changes in overall scattering energy (for incident wave problems) or resonant frequencies (for eigenmode solutions) from pass to pass. Because these quantities represent the results of the model as a whole, they tend to converge more rapidly than

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the field values. Each point in the modeled space can be said to have converged to some value. As a result, specific field quantities at each mesh point are likely to be less accurate than the overall S-parameter or eigenfrequency result of a project solution.

In order to obtain high accuracy results from calculations on field data, we recommend that you take extra precautions to assure that the model's field data is dependable. This might include:

- Running the project to a tighter than usual convergence value.
- Seeding or manually refining the mesh in the areas to be used for calculations.
- Running parametric variations to isolate sensitivity to modeling parameters (such as adaptation frequency or circular cross-section facetization).
- Specifying expressions for output convergence.

As long as the accuracy of specific field data points to be used has been assured, the results of the Field Calculator operations should provide valuable information for your electromagnetic design tasks.

Fast Sweep and Dispersive Models

If a Maxwell solution includes a Fast Frequency Sweep, the Fields Postprocessor can be tuned to display field data at any point in the frequency band swept. The specific frequency selected for viewing need not even be a precise data point at which the S-parameters were calculated. While field calculator operations may be performed at any frequency to which the Fields Postprocessor is set, fast sweep solution field data (away from the center frequency of the sweep) may not be as accurate for lossy and dispersive media, within the interior of solid-meshed finite conductors, etc. For higher accuracy under these conditions, field calculator operations should be performed on a full matrix solution completed at the desired frequency.

Because materials assigned as part of a "Perfectly Matched Layer" (PML) model termination are anisotropic and highly lossy, performing field calculations on the surface of or interior to objects designated as PMLs is not recommended.

Inputs/Excitations

Use **Maxwell** > **Fields** > **Edit Sources** to set the field excitation appropriate to the calculation to be performed. In some cases (e.g., FSS calculations) picking the right field solution set (incident, scattered, or total) is also paramount to obtaining the intended result.

Any field calculation that has not yet been completed (such that the calculator stack still shows some form of "text" string rather than a simple numerical value) is merely a placeholder. Altering the field data loaded in the Postprocessor (by altering port excitations, changing frequency, or picking a different solution set using **Maxwell** > **Fields** > **Edit Sources**) will result in subsequent evaluation of the placeholder to the newly loaded data. To preserve a placeholder's association to an existing data set before altering the excitation to a different data set, you should export the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

register stack by using the **Write** button. You can bring the correctly associated quantity back into the stack by using **Read** after you have changed the field data set selection.

Units

All units in field solutions are expressed in the MKS system, regardless of drawing units. Therefore, E-mag is always in V/m, H-mag in A/m, etc. The exception is that when plotting along a geometry (e.g., along a line) the dimension along the X axis of the graph shows the position along the line in the drawing units, while the vertical (field quantity) axis will be in the MKS system.

Opening the Fields Calculator

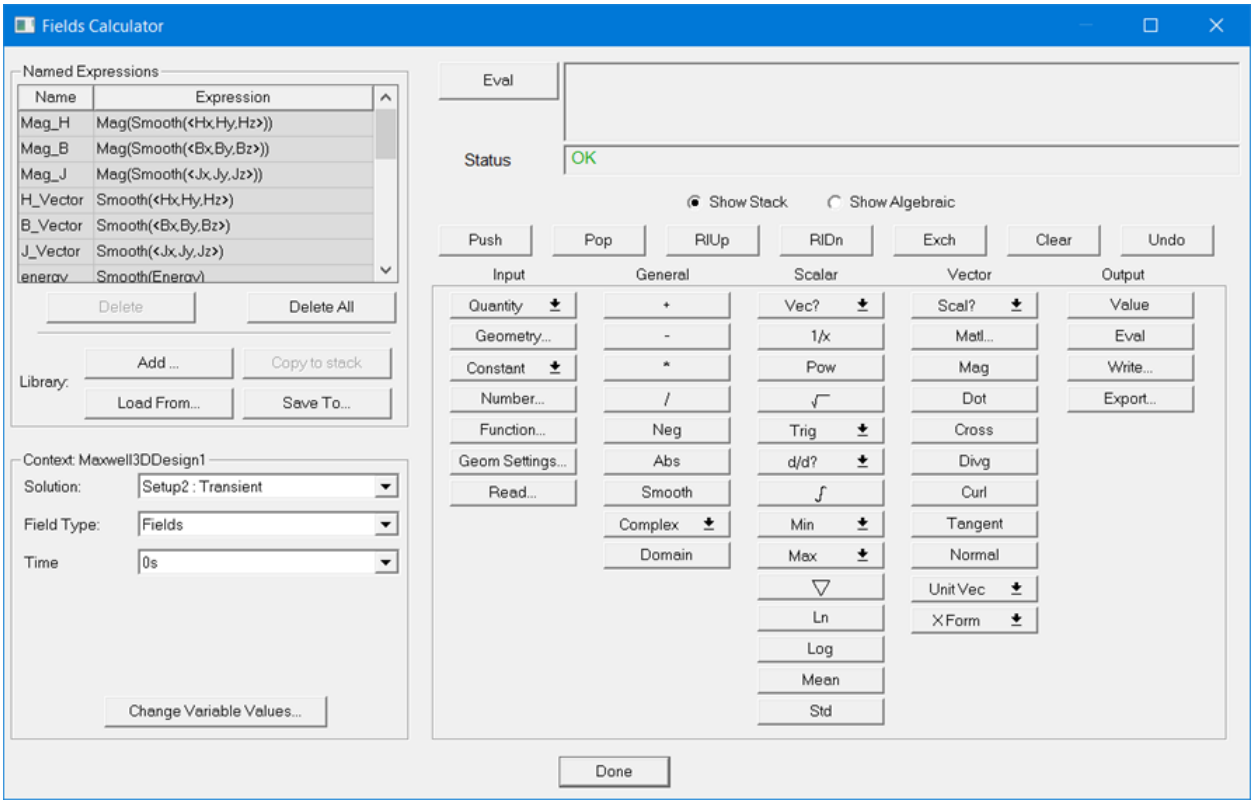
To open the Fields Calculator, do one of the following:

- Click **Maxwell 2D** or **Maxwell 3D** and select **Fields** > **Calculator**.
- Right-click **Field Overlays** in the Project Manager tree, and then click **Calculator** on the shortcut menu.

The **Fields Calculator** window appears.

To view information on a command or screen area, click over the button or screen area on the illustration below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



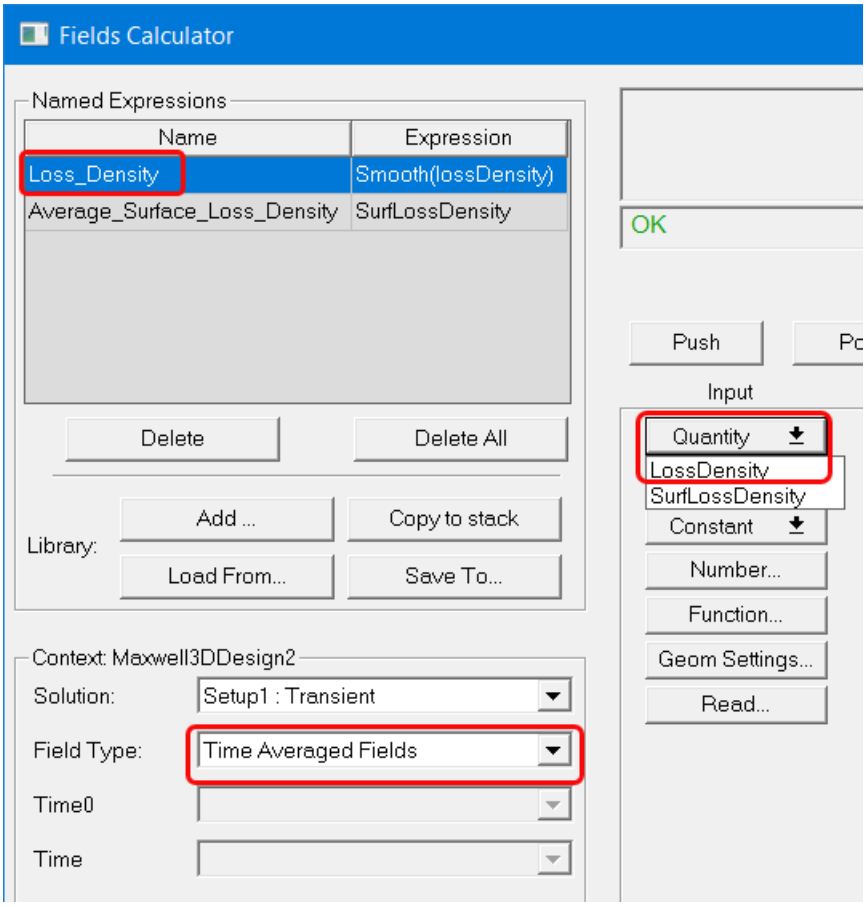
Context Area

The panel at the upper right of the window identifies the context to be used for the calculations. The top line identifies the design. Depending on the design, text entry boxes allow you to select a **Solution**, **Field Type**, **Phase**, or **Time**.

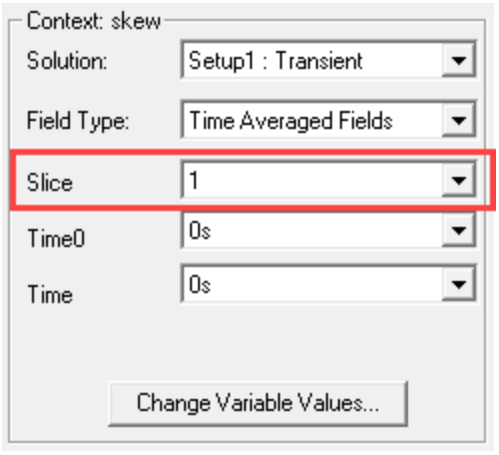
For Maxwell 2D and 3D transient solutions, the field type can be either **Fields** or **Time Averaged Fields**. Selecting **Time Averaged Fields** adds an additional **Time0** field, which is used when calculating time averaged losses. Selecting **Time Averaged Fields** also adds **Loss_Density** as a **Named Expression**, and **LossDensity** as a **Quantity command** selection.

When coupling a Maxwell transient solution with an Icepak design, the field type must be **Time Averaged Fields**. Selecting **Time Averaged Fields** adds an additional **Time0** field, which is used when calculating time averaged losses. Selecting **Time Averaged Fields** also adds **Loss_Density** as a **Named Expression**, and **LossDensity** as a **Quantity command** selection.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For Maxwell 2D transient solutions in which **Use Skew Model** is enabled on the **Model Settings** tab, an additional **Slice** field displays, which allows you to select the skew model slice to use for fields calculations.



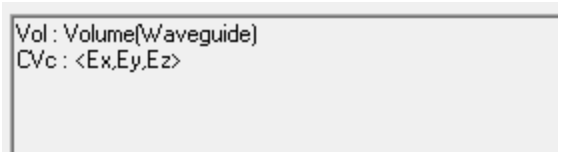
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Field Type here is not related to the edit sources. This is a general term among Ansys EM products (HFSS, Maxwell, Q3D, Icepak, and Mechanical). Some products have more than one field type for different solution types. If only one Field Type is available the drop-down menu is unavailable.

The **Change Variable Values** button opens a *Set Variable Values* dialog box. By default, it has *Use Nominal Design* checked. Unchecking the box lets you select another variable value. **OK** the dialog box to accept the selection.

The Calculator Stack

The calculator is made up of a stack of registers. Registers are displayed in the register display area at the center of the calculator window.

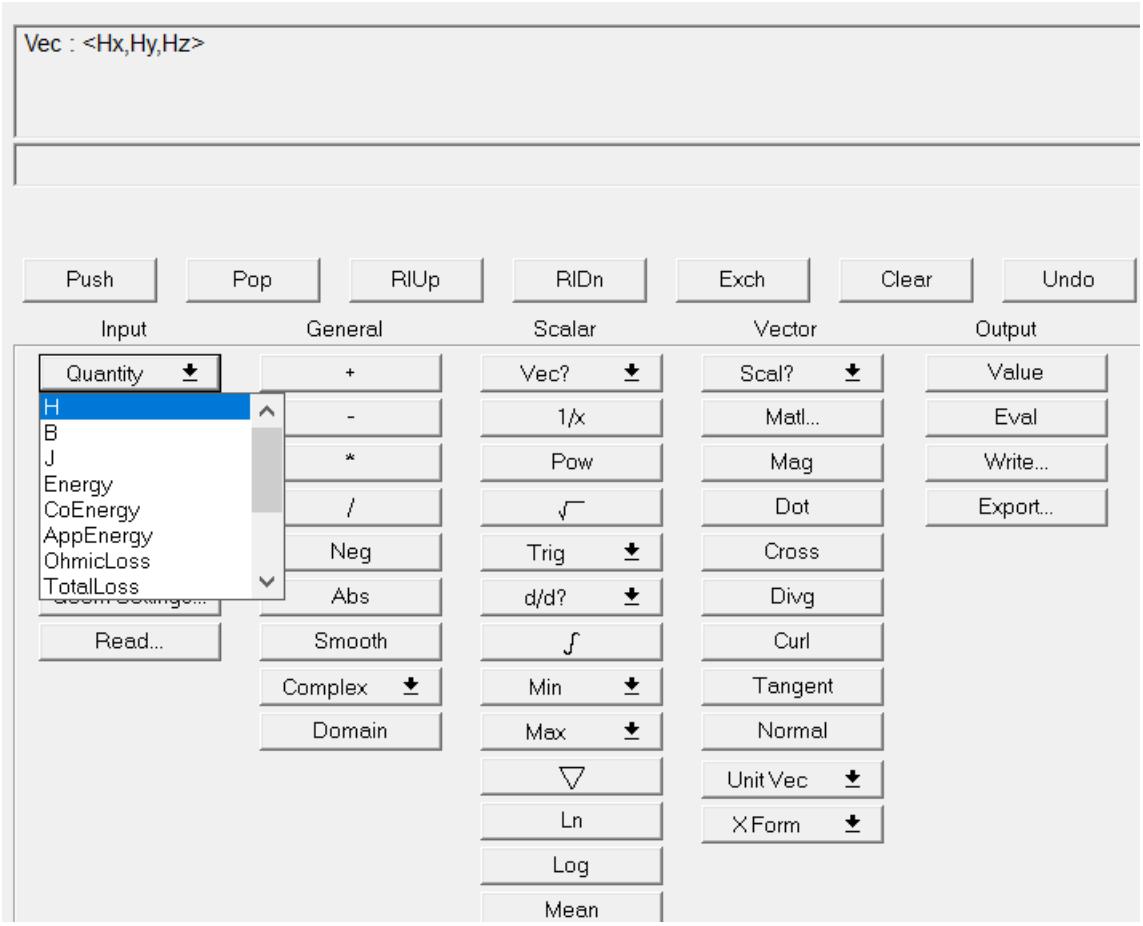


Each register can hold

- field quantities such as the H-field or E-field.
- functional or constant scalars and vectors.
- geometries — points, lines, surfaces, or volumes — on which a field quantity is to be evaluated.

To perform a computation on the field solution, you must first load a basic field quantity into a register on the stack:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Once a quantity is loaded into a register, it can be

- manipulated using mathematical operations such as curls, gradients, cross products, divergences, and dot products.
- integrated over lines, surfaces, or subvolumes of the solution region — either predefined surfaces, volumes, and lists, or lines, surfaces, and volumes that were defined using the **Draw** commands.
- exported to a file, allowing you to superimpose saved solutions.

Stack Commands

Stack Commands influence the entries in the calculator stack and their position. These commands are for manipulating stack contents only, such as copy/paste, delete, or rearrange. The names for these commands match standard stack manipulation conventions. Many are self explanatory, and some can be used in pairs.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Push

Reloads the quantity in the top register onto the top of the stack, creating a new register. The contents of the top two registers are identical.

Pop

Deletes the top register from the stack.

RlUp

Rolls the top register to the bottom of the stack, moving the other registers up the stack.

RlDn

Rolls the bottom register to the top of the stack, moving the other registers down the stack.

Exch

Exchanges the top two registers in the stack.

Clear

Clears the contents of the stack.

Undo

Use this command to undo the effect of the last operation you performed on the contents of the top register. Successive **Undo** commands act on any previous operations.

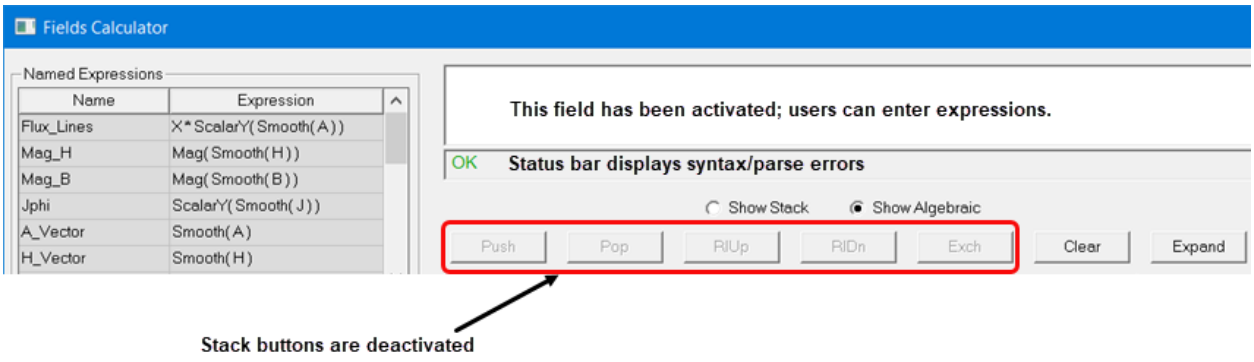
> **Note:**
>
> You cannot undo a simple operation such as loading a field quantity, constant, function, or geometry into the calculator. Instead, use the **Pop** or **Clear** commands to delete these items from the calculator stack.

Algebraic Mode

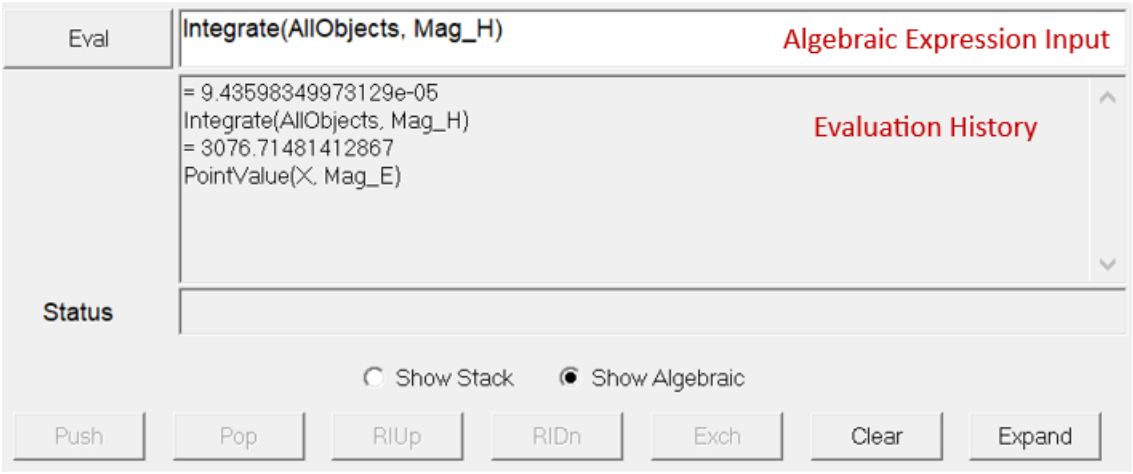
With the **Show Algebraic** option, users can create custom field quantities using algebraic expressions. Users can enter equations directly in the Data field, which is now editable. The text entered is in black when it parses and red when it does not. The Stack Command buttons can

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

also be used to define an algebraic expression; the input text will be inserted at the current insert point in the Data field.



A sample algebraic expression is shown below:



> **Note:** To activate the Show Algebraic option, you must select the **Maxwell Support Expression Input in Fields Calculator** option in the Beta Options window; see Using the Fields Calculator for more details.

An algebraic expression can be saved in the Named Expressions list. If **Show Algebraic** is selected, all the named expressions will be listed in algebraic format. If **Show Stack** is selected, the named expressions will be listed in stack format.

An algebraic expression is either a `namedExpression`, a `constant`, a `name`, an `operator`, two `expressions` separated by `binaryOperator`, or an `expression` enclosed in parentheses.

An `operator` is an *opName* followed by `arguments`.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

An `argument` is either empty parentheses, an `expression` enclosed by parentheses, or two `expressions` separated by a comma and enclosed by parentheses.

A `binaryOperator` is one of the following literals: '+', '-', '*', '/', '`%`', '^'

A `name` is either a project, design, or intrinsic variable (scalar or vector), or a geometry.

A *constant* can be one of the following:

- scalar literal; for example, "1.5e-10"
- complex number; for example, "3.5-1.0i" or "<(3.5, -1.0)>"
- 3D vector; for example, "<1.0, 2.0, 3.0>"
- complex 3D vector; for example, "<1.0-2.0i, 2.0-3.0i, 3.0-4.0i>" or "<(1.0, -2.0), (2.0, -3.0) ,(3.0, -4.0)>"

The grammar for an algebraic expression is summarized below:

```
expression : namedExpression
   | constant
   | name
   | operator
   | expression binaryOperator expression
   | '(' expression ')'
operator : opName arguments
binaryOperator : + | - | * | / | % | ^
arguments : '(' <empty> ')'
   | '(' expression ')'
   | '(' expression ',' expression ')'
```

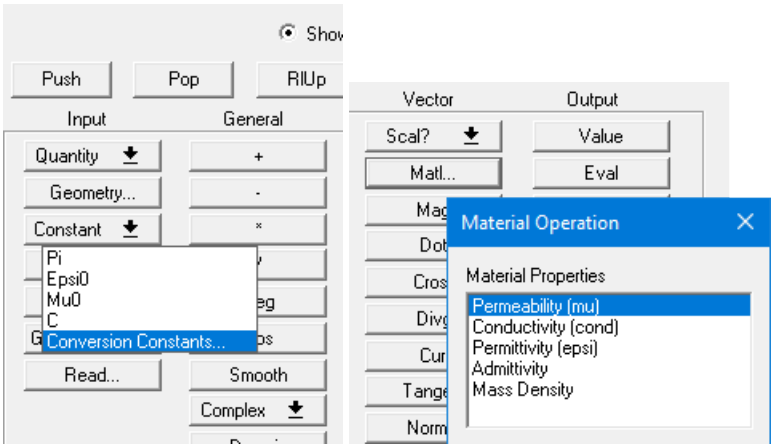
Note the following:

- `%` is used for the vector dot product; for example, `Integrate(AllObjects, Real(E%H))`
- `*` supports vectors on both sides as the operand and the cross-product. For example, `Integrate(AllObjects, Mag(E*H))` can be used instead of `Integrate(AllObjects, Mag(Cross(E,H))`.

Names

The following named constants and material properties are supported as input in the Fields Calculator:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note that Mass Density corresponds to keyword "`MassDensity`".

If there is a conflict between standard quantity names, user-defined named expressions, and user-defined variables, the default name conflict resolution order is

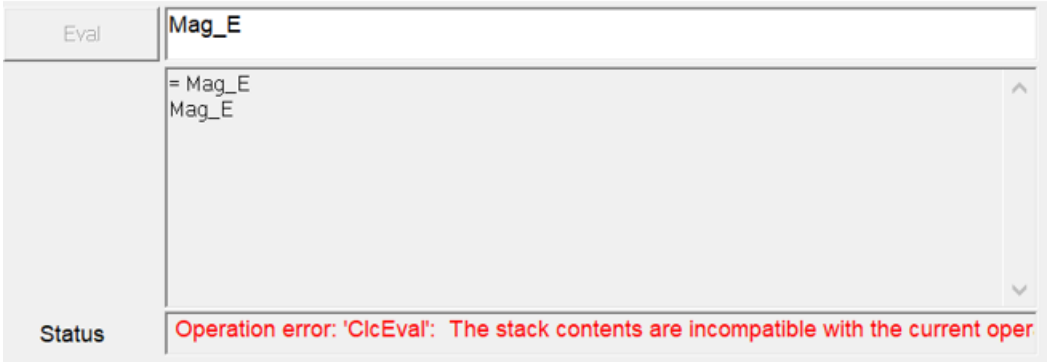
1. Standard quantities such as E and Mag_E
2. User-defined named expression
3. User-defined variables

Words not immediately followed by an open parenthesis will be treated as a quantity or variable. Therefore, a user-defined variable named `Real` not followed by open parenthesis will be recognized as a variable instead of the operator.

The `Var` or `Expr` operator can be used to override the default name conflict resolution. For example, a user-defined design variable named `Mag_E` would normally not be recognized. Instead, the standard quantity `Mag_E` will be assumed, and evaluation will fail:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Value	Unit	Evaluated...	
offset	0	in	0in	D
Mag_E	-2 - 2i		-2 - 2i	P
Real	3		3	P
Var	-1		-1	P
epsi0	-2		-2	P
epsi	-9		-9	P
E	3		3	P



With the `Var` operator, the user-defined variable can be parsed correctly and evaluated for the user -defined value:

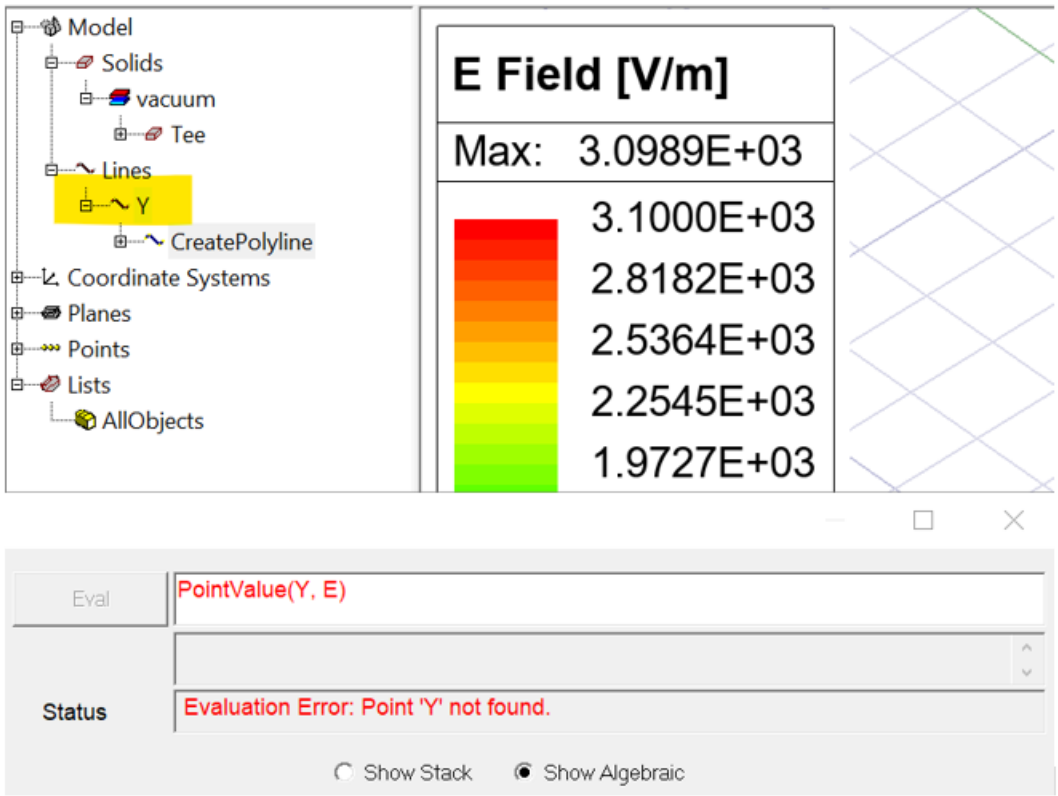
Here is another example that demonstrates how a user-defined variable can conflict with a standard named constant, such as epsi0, the vacuum permittivity, and how to use the `Var` operator to resolve the conflict:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Geometry

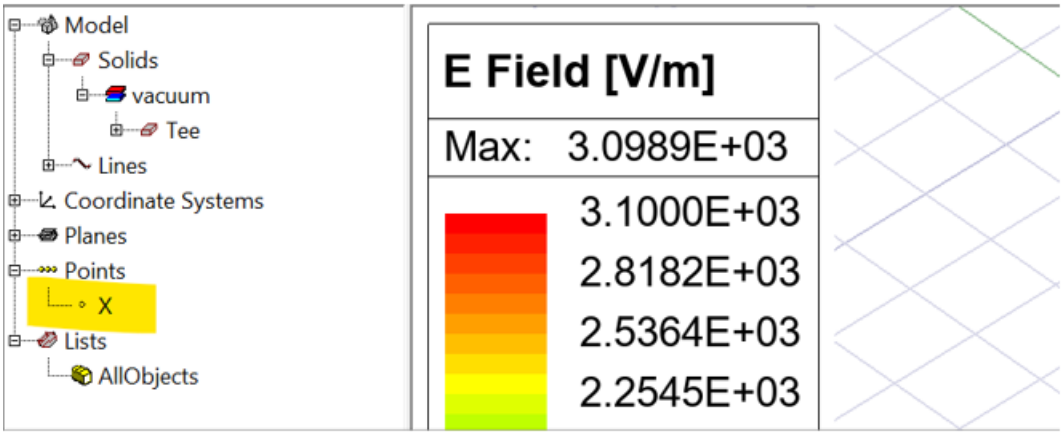
If an operator requires a geometry input, the geometry argument can be either first or second in the argument list, or in some cases, both. The operator automatically infers the geometry of the argument; no explicit specifiers are required. For example, `PointValue(GeomX, ExpressionY)` will assume `GeomX` is a point.

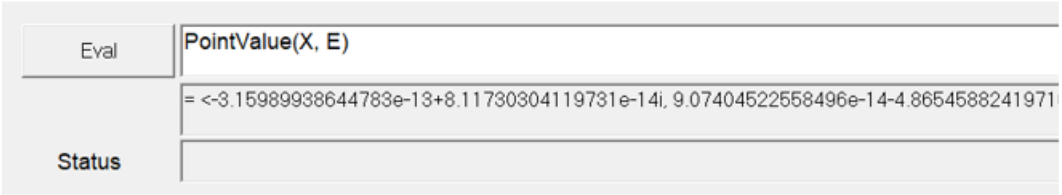
If the user specifies an operator, but uses the wrong type of geometry in the argument list, the Calculator will return an error. In the following example, the `PointValue` operator has a line in its argument list, so the Calculator generates an error:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following example shows the output when the geometry argument are specified correctly:





The following operators assume the geometry operand to be of the Volume type:

- `Integrate`
- `Mean`
- `Std`
- `Min`
- `Max`
- `MinPos`
- `MaxPos`

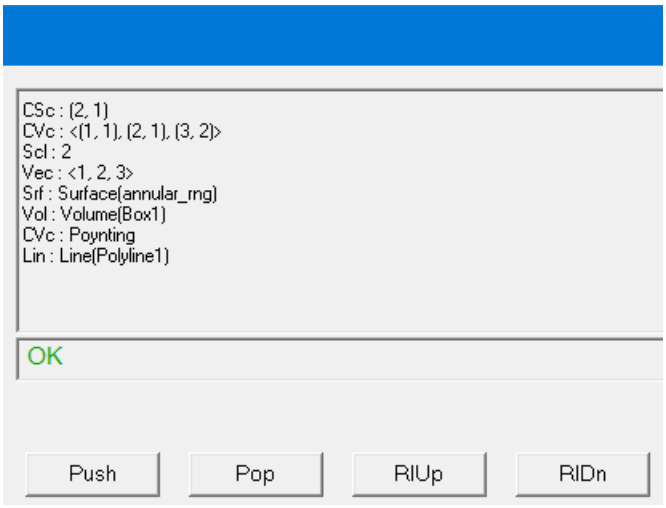
Registers

In the **Show Stack** mode, the Fields Calculator registers registers hold field quantities, numbers, vectors, and geometries. No registers are created until you load something into the calculator; therefore, this part of the window is initially blank. As items are loaded into the calculator, it creates new registers to hold them.

> **Note:** For Maxwell and HFSS users, make sure the Calculator is in **Show Stack** mode.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The calculator is capable of performing operations on a number of different data types. In many instances, a calculation requires certain type(s) of data to be present in the correct order in the stack register. Many operations result in a different data type than the inputs. In order to show you the type of data contained in each stack entry, calculator denotes its data type by a prefix indicator as shown in Stack Contents showing Data Type Indicators (at left), below.



The following table describes and defines each indicator, and provides guidance regarding operations which can convert data from one type to another.

Vec	Vector (non-complex) quantities, which have both direction and magnitude at each point in space. The x-, y-, and z-components of these quantities are stored in the register. Vectors are always evaluated in the coordinate system of the model. To convert a vector quantity to a scalar, use the Scal? drop-down menu from the Vector column. Suboptions ScalarX, ScalarY, and ScalarZ will take the appropriate scalar component of the vector data. Optionally, you can also Dot the vector with another vector to obtain the appropriate scalar result, or use the Tangent (return the tangential scalar component of) or Normal (return the normal scalar component of) operations to relate the vector quantity to a geometric data (Lin, Srf) stack entry. Convert to a Complex quantity using the CmplxR and CmplxI operations described in Scl, above.
Scl	Scalar quantities, which have a magnitude only. This is a simple numerical value. To convert a scalar to a vector quantity, use the Vec? drop-down menu in the Scalar column. The choices VecX, VecY, and VecZ convert the scalar data to vector data aligned with the X, Y, or Z unit vectors, respectively. You can also multiply the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	scalar quantity by a desired vector direction entered manually (Num drop-down in the Input column) or obtained using the Unit Vec button from the Vector column. To convert a scalar to a complex quantity, use either CmplxR (assign the scalar value as the real component of a complex quantity) or CmplxI (assign the scalar value as the imaginary component of a complex quantity), both found under the Cmplx drop-down in the General Column.
Cvc	Complex vector quantities, which are numerical values with real and imaginary components. Convert to a vector quantity using the same techniques described for Scl, above. Convert to a scalar using Real (take the real component), Imag (take the imaginary component), CmplxMag, (take the magnitude of the complex number) or CmplxPhase (take the phase of the complex number), all within the Cmplx dropdown in the General column.
Csc	Complex scalar quantities. This is a numerical value with real and imaginary components. Convert to a vector quantity using the same techniques described for Scl, above. Convert to a scalar using Real (take the real component), Imag (take the imaginary component), CmplxMag, (take the magnitude of the complex number) or CmplxPhase (take the phase of the complex number), all within the Cmplx drop-down in the General column.
Pnt	Points
Lin	Lines, which may be straight, curved, or “polylines” in three dimensional space. Lines may also be open (have two endpoints) or closed (ending vertex same as starting vertex).
Srf	Surfaces. Surfaces need not be planar, and may actually comprise a list of object faces (faces list) as well as planar slices through the entire model space (cutplanes).
Vol	Volumes, which may include sets of discontinuous object volumes created as an Object.
SclLin	Scalar value on a line
VecLine	Vector value on a line
SclSrf	Scalar value on a surface
VecSrf	Vector value on a surface
List	These indicators may exist alone, representing geometric data only, or in combination with one of the categories above, indicating a type of data applied to the geometric entity in question. For example, the notation SclSrf identifies a stack entry containing Scalar data on a Surface geometry set. To select only the portion of a given data entry which exists along, on, or within a given geometry quantity, use the Value button in the Output column of the calculator. Other operations (e.g., integration, the Normal button) operate when a data quantity is

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	in the second stack register and a geometric quantity is in the top stack register. Full descriptions of the register requirements for each individual command is available in the online help.

When examining calculator registers, keep the following in mind:

- To move or delete calculator registers, use the stack commands.
- To save a register to a disk file, use the **Write** command.

Users must be cautious concerning what type of data the Fields Calculator is manipulating and whether or not it is compatible with the desired operation. For example, the integral operation is often misused. Note that the integral sign is in the *Scalar* column, implying that, to integrate complex number/quantities, the user will have to integrate the real and imaginary components separately. In other words, performing integration on complex number/quantities must be achieved by parts.

Enlarging the Register Display Area

If there are too many registers to fit into the display area, do one of the following:

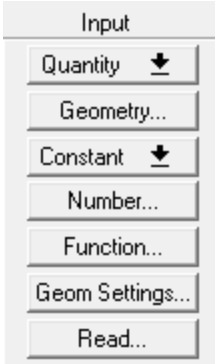
- Use the scroll bars to view the hidden registers.
- Enlarge the calculator window by dragging window's borders.

Units of Measure

Unless you are prompted specifically for the unit of measure, all measurements should be assumed to be in SI base units, not model units.

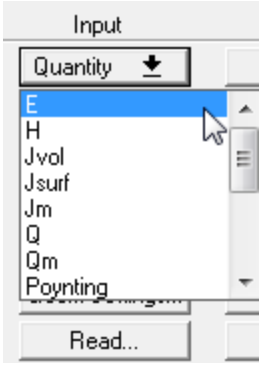
Input Commands

The *Input* Column contains all the calculator functions which place new values into the stack, such as field data, constant, geometry data, coordinate system selection, or user-entered vector or complex numbers.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Field data (e.g., E-field, H-field, Poynting vector) for the current project solution is input from the **Quantity** drop-down menu selection.



The quantities specifically available from the calculator are dependent on the solution type and include the E-field, H-field, J-vol (volume current), Poynting vector, Displacement, Temperature, Heat Flux, and Surface and Volume Loss Densities. All quantities are Peak Phasors, and not RMS quantities, with the phase information captured in the real and imaginary components and the field orientation captured in the vector components. Although the Poynting vector is automatically calculated by the interface as 0.5, it will appear in the calculator stack as a Complex Vector quantity. The imaginary portion should however be zero or ignored.

Use the following commands to load data onto the top of the calculator stack:

Quantity	Basic field quantities, such as E and H, and simple derived quantities such as volume current.
Geometry	Geometries such as planes, points, polylines, 3D lines, face selections, and volumes
Constant	Predefined constants such as π, ε_0, and conversion factors between various units of measurement.
Number	Vector and scalar constants, including complex numbers.
Function	User-defined or intrinsic variables
Geom Settings	Number of equally spaced points used to integrate fields and other quantities on a line.
Read	Read previously saved calculator registers containing field quantities.
Output Vars	This button appears only for eigenmode problems. Freq is the only value there. Evaluation for Freq returns a complex value.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These quantities can be manipulated using the **Stack** commands, **General** commands, **Scalar** commands, and **Vector** commands. The results of these calculations can then be examined using the **Output** commands.

Quantity Command

The **Quantity** command loads a field quantity into the top register of the calculator. The quantities available depend upon the solution and design type. These are the same field quantities available for plotting from the **Maxwell 2D** > **Fields** and **Maxwell 3D** > **Fields** menu.

Geometry Command

The Fields Calculator **Geometry** command opens a dialog box for selecting a geometry to load into the top register of the calculator.

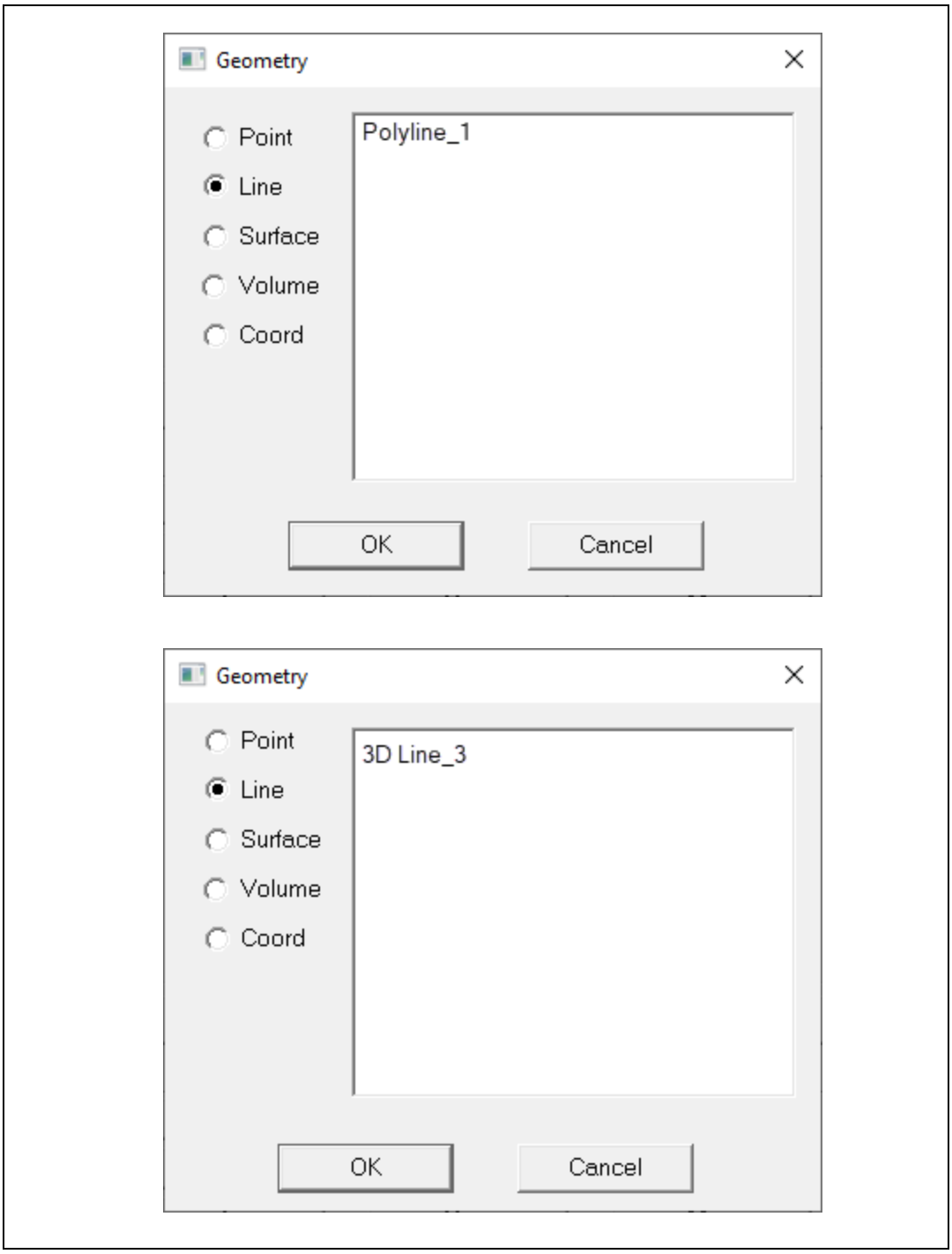
Do this to

- find the value of derived field quantities on any point, line, surface, or volume.
- plot quantities directly from the calculator.
- display a previously defined isosurface, maximum or minimum field point using the **Draw** command.

The following types of geometries are available:

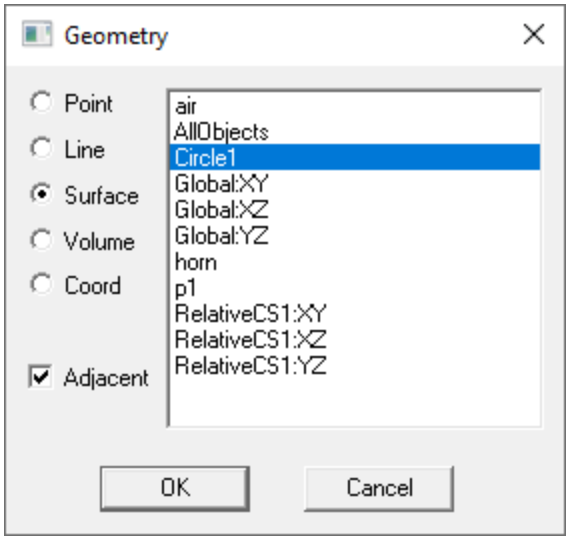
Point	See Drawing a Point Object. Drawn points are listed in the calculator's *Geometry* dialog box when **Point** is selected.
Line	See Drawing a Line Segment or Drawing a 3D Line Segment. Drawn lines are listed in the calculator's *Geometry* dialog box when **Line** is selected. To set the number of points on a line, see Geom Settings.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface



- Allow operation on the adjacent side surface geometry just like regular surface geometry.
- All calculation on this geometry will be done using data from the adjacent side of the surface.
- Support scripting, saving and loading expression with adjacent side surface geometry.

```
Srf : AdjacentSurface(Circle1)
CVc : <Ex,Ey,Ez>
```

Volume

3D objects, regions, and object selections (of 3D objects including AllObjects) are available in the calculator's *Geometry* dialog box when **Volume** is selected.

Coord

Coordinate systems are available in the calculator's *Geometry* dialog box when **Coord** is selected.

To load a geometry into the calculator:

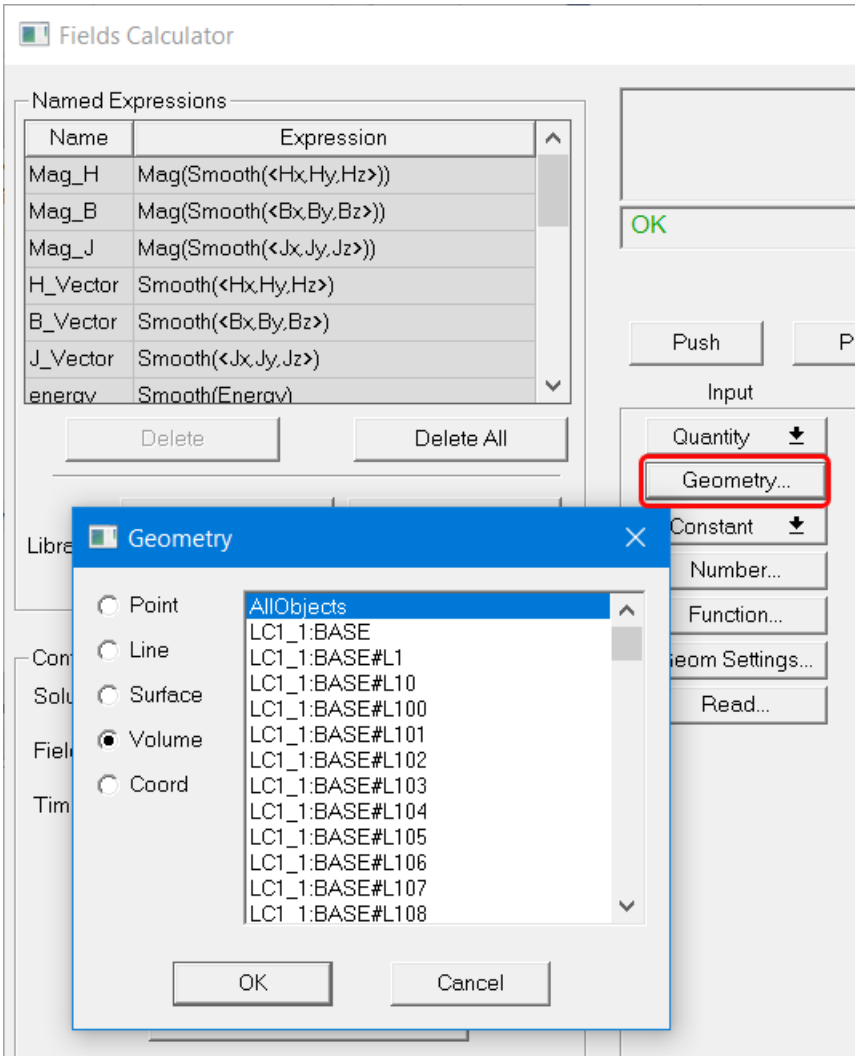
1. In the Fields Calculator, click **Geometry**.

   The **Geometry** dialog box appears.

2. Select a geometry type.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A list of all applicable geometries appears. If a design includes Layout Components, the Geometry list shows the available component geometries as well.



Note:

- Non-convex 3D non-model solids are not supported for Field Overlay Plots and Fields Calculator computations.
- Object selections containing non-model object(s) are supported for Field Overlay Plots, but are not supported for Fields Calculator computations.

In the modeler window, these Layout Component objects are drawn as visualizations without actual geometry. However, they can be used to calculate field quantities as shown below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These geometry operators can be used to perform computations just like one constructed from regular 3D geometry. These additional objects are generated only after reading a mesh, so they won't show up if there is no solution.

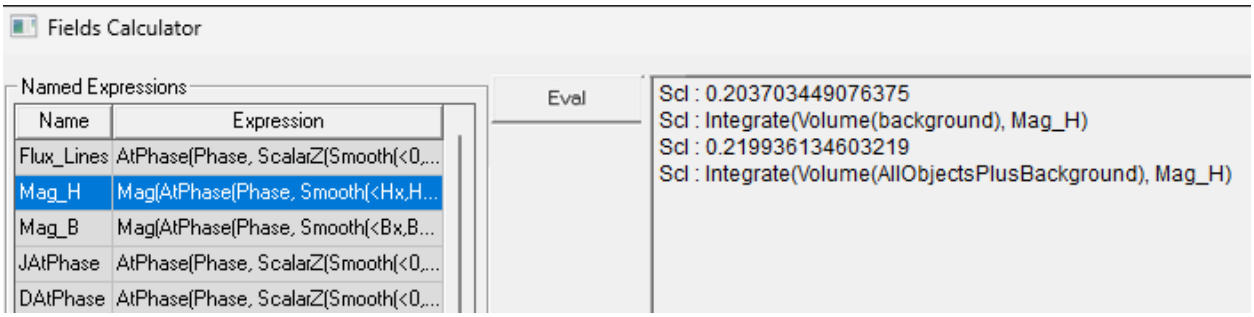
3. Click the geometry.
4. Click **OK** to load the geometry.

> **Note:** Consider a box (Box2) that is completely enclosed in a bigger box (Box1), so that no faces of Box2 are touching any faces of Box1. Box2 is actually implicitly subtracted from Box1 as is done in our solvers. So Box1 is used as if Box2 were already subtracted from Box1. Volume(Box1) is Box1 excluding Box2, and Surface (Box1) contains faces from both Box1 and Box2.

Background Geometry

For 2D solver simulations that include a Balloon boundary condition, the Geometry dialog box includes entries for **background** and **AllObjectsPlusBackground**. The background is the part that is outside the geometry but on which the solver generated a mesh.

Including the background in the calculation will affect the results as demonstrated in the following figure:

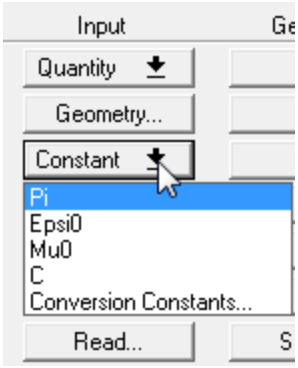


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: For a 2D solver simulation that does not include a Balloon boundary condition, but includes other boundary conditions, such as a VectorPotential, you may see **background** listed in the Geometry dialog box, but in this case, the background has no geometry and has no effect on the calculations.

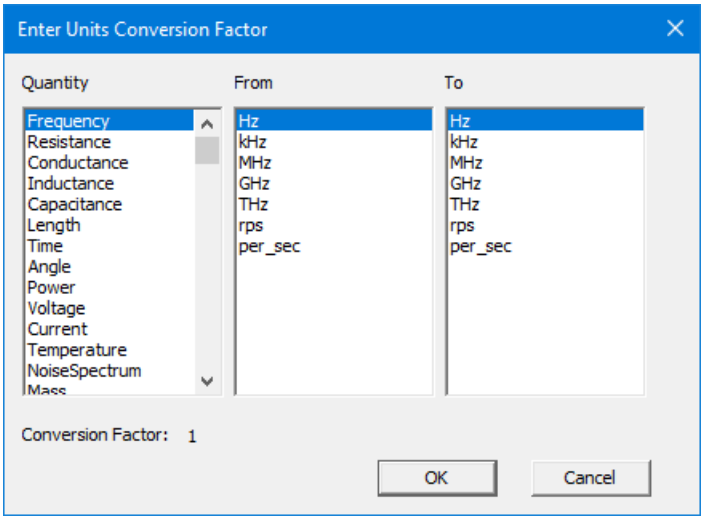
Constant Command

The **Constant** command loads one of the following four predefined constants or a selected conversion constant into the top register of the calculator:



Pi	π
Epsi0	The permittivity of free space, $\varepsilon_0 = 8.85418782 \times 10^{-12}$ C^2/Nm^2
Mu0	The permeability of free space, $\mu_0 = 4\pi \times 10^{-7}$ Wb/Am
c	The speed of light in vacuum, $c = 2.99792458 \times 10^8$ m/s
conversion constant	Displays the *Enter Units Conversion Factor* dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This lists a range of Quantities (such as frequency, resistance, and others) along with a list of Units (Hz to Thz, and rps) to convert From and To. The ratio of the Units From to the Units to is displayed for the selected values as **Conversion Factor**.

Number Command

The **Number** command enters one of the following into the top register of the calculator:

Scalar	A scalar constant. To enter a constant scalar number: 1. Click **Number**. The *Input Number* dialog box appears. 2. Select **Scalar**. 3. Type the scalar value in the **Value** text box. 4. Click **OK** to load the number into the top register.
Vector	A vector constant. To enter a constant vector: 1. Click **Number**. The *Input Number* dialog box appears. 2. Select **Vector**. 3. Enter the x-, y-, and z-components of the vector. 4. Click **OK** to load the vector into the top register.
Complex	A complex constant, entered in the form $C=A+jB$, where A represents the real part of the constant and B represents the imaginary part.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Number**.

   The *Input Number* dialog box appears.

2. Select **Scalar** or **Vector**.
3. Select **Complex**.
4. Enter the real and imaginary components of the number.
5. Click **OK** to load the number into the top register.

Function Command

The **Function** command enters a predefined scalar or vector function into the top register of the calculator:

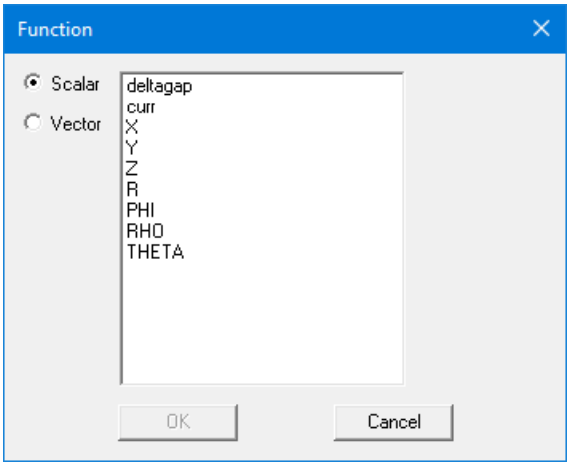
Scalar

A scalar function.

To enter a function:

1. Click **Function**.

   The *Function* dialog box appears.



2. Select **Scalar**.
3. Select the function from the list.
4. Click **OK** to load the functional scalar into the top register.

Vector

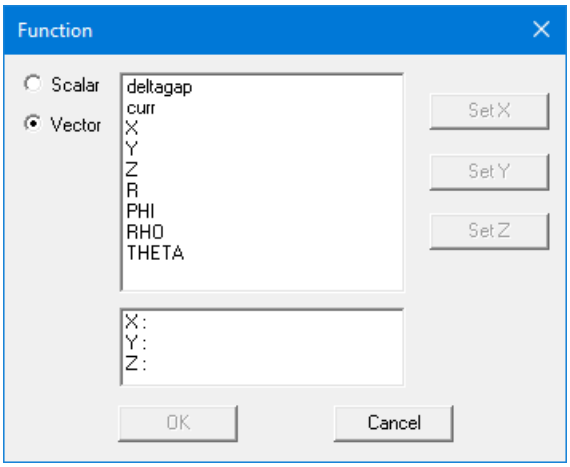
A vector function, in which the values of the vector's x-, y-, and z-components are given by functions.

To enter a functional vector:

1. Click **Function**.

   The *Function* dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select **Vector**.
3. Select the function from the list.
4. For each component of the vector, click **SetX, SetY,** and **SetZ**.
5. Click **OK** to load the functional vector into the top register.

Note:

The predefined variables **X, Y,** and **Z**; **RHO, THETA, R,** and **PHI**; and any functions that you created can be used to define functional scalar and vector quantities.

Use of the Global Coordinate System is assumed. Local coordinate systems are not used.

Geometry Settings

Clicking the **Geom Settings** button opens the **Geometric Settings** dialog box. The dialog box allows you to specify the line discretization, the number of equally spaced points used to integrate fields and other quantities on a line. The default is 1000 points.

To set the line spacing for geometry settings:

1. In the Fields Calculator, click **Geom Settings**.

   The **Geometry Settings** dialog box appears.

2. Enter a value in the **Line Discretization** box, and click **OK**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reading an External Input

This command copies the contents of a disk file into the top register. The register must be one that has been saved using the Write output command.

To read an external input into the register:

1. Click **Read**.

   The **Open** dialog box appears.

2. Use the file browser to specify the register's file name and directory path. A .reg extension is automatically assumed for register files, and click **Open**.

The contents of the file are copied to the top register in the stack.

General Commands

The General Column contains calculator operations which can be performed on many different data types (e.g., vector, scalar, complex). With the exception of the Complex menu, all are distinct functions. Most are self-explanatory, with the exception of Smooth which performs some data "smoothing" or statistical averaging on the top stack entry.

Use these Fields Calculator commands to perform operations on both vector and scalar quantities.

+ (Add)	/ (Divide)	Smooth
-- (Subtract)	Neg	Complex
* (Multiply)	Abs	Domain

+ (Add)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adds the quantities in the top two registers of the calculator.

- (Subtract)

Subtracts the quantity in the top register from the quantity in the second register. The two registers must hold the same type of quantity (both scalar or both vector). You cannot subtract a scalar from a vector (or vice versa).

> **Note:** Consider a box (Box2) that is completely enclosed in a bigger box (Box1), so that no faces of Box2 are touching any faces of Box1.
>
> If you explicitly subtract Box2 from Box1, any calculation on the surface (faces) of Box1 will use the 6 exterior faces and the 6 interior faces. Any calculation on the volume of Box1 will use the difference in volume between Box1 and Box2.
>
> If you do not explicitly subtract Box2 from Box1, the inner box is only implicitly subtracted. Any calculation on the surface of Box1 in this case will use only the 6 exterior faces of Box1. Any calculation on the volume of Box1 will use the entire volume without subtracting the volume of Box2.

*** (Multiply)**

Multiplies the quantity in the top register by the quantity in the second register. One of the two registers must contain a scalar value; the other register can be either a scalar or a vector.

/ (Divide)

Divides the quantity in the second register by the quantity in the top register. The second register must contain a scalar value; the top register can be either a scalar or a vector.

Neg

Changes the sign of the quantity in the top register.

Abs

Takes the absolute value of the quantity in the top register.

Smooth

Smooths the quantity in the top register. Because of the numerical solution technique used, field values are not always continuous across the boundaries of the individual elements that make up the finite-element mesh. Smoothing makes the values continuous by taking a weighted average from all of a node's neighboring elements. The weights are based on angles, so elements with larger angles provide larger contributions. In general, use smoothing before plotting a quantity.

Complex

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These commands perform operations on a complex quantity in the top register. Complex quantities are indicated by a **C** at the beginning of the register label. They can be represented in terms of real and imaginary components, or in terms of magnitude and phase:

$$C = A + jB = Me^{j\phi}$$

where:

- *A* is the real part of the complex number.
- *B* is the imaginary part of the complex number.
- *M* is its magnitude, which is equal to $\sqrt{A^2 + B^2}$.
- ϕ is its phase, which is equal to **atan**(B/A).

The **Complex** commands includes a drop-down menu that let you do the following:

Real	Takes the real part of the complex quantity (A).
Imag	Takes the imaginary part of the complex quantity (B).
CmplxMag	Takes the magnitude of the complex quantity (M). Due to interpolation issues, the sequence of calculations may cause a loss of accuracy. It is best to define the points , separately obtain the value of the real part, then the imaginary part, and use those values to calculate the magnitude and and phase. For the sequence for using the Fields Calculator to obtain the real and imaginary parts, see the procedure here.
CmplxPhase	Takes the phase of the complex quantity(ϕ).
Conj	Takes the complex conjugate of the quantity in the top register. If a complex number is given by *C* = *A* + *jB*, its complex conjugate is given by *C** = *A* - *jB*.
AtPhase	Lets you specify the phase angle, θ, at which an field quantity is evaluated. These quantities can be represented in the form $A(x,y,z,t) = A(x,y,z)\cos(\omega t + \theta(x,y,z))$. where • ω is the angular frequency at which the quantities are oscillating, specified during the solution. • θ(x,y,z) is the phase angle (the offset from a cosine wave that peaks at t=0). Entering the phase angle lets you compute the real part of the field's magnitude at different points in its cycle.
CmplxReal	Converts the real scalar of the top register to the real part of a complex number.
CmplxImag	Converts the real scalar of the top register to the imaginary part of a complex number.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CmplxPeak	Calculates the peak value of a given complex vector. Intuitively, this calculates the maximum magnitude of the equivalent real vector in a waveform.

Domain

This limits a calculation to the volume you specify. The domain filter works for scalars, vectors, complex scalars and complex vectors. This operation requires the top two entries of the stack to be a volume geometry and a numeric field quantity. To do this:

1. Load the field quantity into the top register, and perform any necessary operations on it.
2. Load the volume using the **Geometry** command.
3. Click **Domain**.

The **Domain** command is often used to limit a calculation or plot to the intersection of a surface and an object or group of objects. If you export a domain filtered numeric, points that are filtered out by the domain will not be written out.

Steps for Calculating the Complex Vector Electric Field

These are the field calculator steps to obtain the real part, the imaginary part, and the magnitude of the x-directed, y-directed, and z-directed components of the phasor electric field. For each of these vector components, the magnitude should be equal to sqrt(real^2+imag^2), but the need to interpolate values and the calculation sequence means that Maxwell does not give this value unless the specified location is directly on a mesh element node.

1. Calculate real part of complex vector electric field (in x, y, and z directions):
    a. Qty > E
    b. Complex > Real
    c. Geometry > Point > fieldcalc_point
    d. Value
    e. Eval
2. Calculate imaginary part of complex vector electric field (in x, y, and z directions):
    a. Qty > E
    b. Complex > Imag
    c. Geometry > Point > fieldcalc_point
    d. Value
    e. Eval

Use the real and imaginary components to manually calculate the magnitude as the sqrt (real^2+imag^2).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scalar Commands

The Scalar Column contains calculator operations which can only be performed on scalar stack entries.

Drop-down menus in this column include Vec? (convert scalar to vector), Trig (trigonometric, containing sin, cos, etc. functions), d/d? (derivative with respect to...), Max and Min (self-explanatory), Gradient, ln (natural logarithm base e), log (logarithm base 10), mean, and Std (standard deviation). Note that the calculator's Integrate function is located in the Scalar column. The implication is that integration can only be performed on scalar quantities. To perform integration upon complex quantities, you must perform the integration separately on real and imaginary subcomponents.

Use these commands to perform operations on scalar quantities.

Vec?	Makes the scalar quantity in the top register a vector component.
1/x	Takes the inverse of the scalar quantity in the top register.
Pow	Raises a scalar quantity to the power you specify.
√? **(Square Root)**	Takes the square root of the quantity in the top register.
Trig	Performs a selected trigonometric operation on the value in the top register of the calculator stack (Sin, Cos, Tan, ASin, ACos, ATan). Angles are in radians.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Additionally, the ATan2 operation is available, which requires two arguments. This operator calculates the arc tangent of the following ratio: The second register in the stack divided by the top register. **Example:** If the top register value is 1.5, and the second register value is 2, the ATan2 function returns ATan(2/1.5) = 0.5880 radians. All other Trig functions operate on a single argument (top register only).
d/d?	Takes the partial derivative of the quantity in the top register.
∫? (Integral)	Takes the integral of a scalar quantity over a volume, surface, or line. To perform integration upon complex quantities, you must perform the integration separately on real and imaginary subcomponents.
Min	Computes the minimum of a scalar field quantity on a line, surface, or volume relative to your choice of Value or Position.
Max	Computes the maximum of a scalar field quantity on a line, surface, or volume relative to your choice of Value or Position.
∇ (Gradient)	Takes the gradient of the scalar quantity in the top register.
ln	Takes the natural logarithm (base e) of the scalar quantity in the top register.
log	Takes the logarithm (base 10) of the scalar quantity in the top register.
Mean	Takes the mean of the quantity in the top register.
Std	Takes the standard deviation of the quantity in the top register.

Vec? Command

Makes the scalar quantity in the top register a vector component. Choose from the following:

VecX	The x-component of a vector.
VecY	The y-component of a vector.
VecZ	The z-component of a vector.

1/x (Inverse) Command

Takes the inverse of the scalar quantity in the top register.

Pow Command

Raises a scalar quantity to the power you specify.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To raise a scalar quantity to a power:

1. Enter the quantity into the calculator.
2. Enter the exponent to which it is to be raised into the calculator.
3. Click **Pow**.

   The results are displayed in the top register.

Square Root Command

Takes the square root of the quantity in the top register.

Trig

Takes one of the following trigonometric values of the value(s) in the top register(s) of the calculator stack:

Sin	Sine
Cos	Cosine
Tan	Tangent
ASin	Arcsine
ACos	Arccosine
ATan	Arctangent
ATan2	A two-argument version of the **ATan** function. Takes the y and x coordinates of the point (in the top two stack registers).

d/d? (Partial Derivative) Command

Takes the partial derivative of the quantity in the top register:

d/dx	Takes the partial derivative of the quantity with respect to x.
d/dy	Takes the partial derivative of the quantity with respect to y.
d/dz	Takes the partial derivative of the quantity with respect to z.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Integral Command

Takes the integral of a scalar quantity over a volume, surface, or line. The top register must contain a geometry and the second register must contain the scalar quantity to be integrated.

To perform an integration:

1. Load a quantity into the top register of the calculator, and perform any required operations on it.
2. Use one of the **Geometry** commands to load the line, surface, or volume over which the quantity is to be integrated.

   > **Note:**
   >
   > If you computed the tangent or normal of the quantity to be integrated, you do not have to load any geometry selection onto the calculator stack. Maxwell integrates the tangential or normal component of the quantity over the line on which you computed its tangent, or the surface on which you computed its normal.

3. Choose the command to integrate the scalar quantity over the geometry.

To find the numerical results of an integration, use the **Eval** command.

> **Note:** In **Maxwell 2D**, the Fields Calculator accounts for the difference between XY and RZ models when handling data and results. The integral command will perform an XY integral on the model assuming a 1 meter depth, while for an RZ model it will perform an integral on the 360 degree equivalent volume.

Min Command

Computes the minimum of a scalar field quantity on a line, surface, or volume. Two options are available:

Value	Finds the magnitude of the minimum value of the field.
Position	Finds the point where the minimum field value occurs. You can then: • Plot the minimum field value at the point. • Plot basic field quantities at the point. • Load the point into the calculator. • Change the point's location.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These commands operate in the same way as the **Max** commands. Use the **Eval** command to display the actual minimum field value or the coordinates of the point where it occurs.

Max Command

Computes the maximum of a scalar field quantity on a line, surface, or volume. Two options are available:

Value	Finds the magnitude of the maximum value of the field.
Position	Finds the point where the maximum field value occurs. You can then: • Plot the maximum field at the point. • Plot field quantities at the point. • Load the point into the calculator. • Change the point's location.

To compute the maximum field value:

1. Load a field quantity into the calculator, and perform any necessary operations on it. Keep the following in mind:
   - You cannot find the maximum value of a vector quantity. Therefore, make sure that the result is a scalar.
   - Before computing the maximum value of a complex quantity, you must find the real part of the quantity using the **Cmplx/Real** or **Cmplx/AtPhase** commands.
2. Load a point, line, or volume into the calculator using one of the **Geometry** commands.
3. Do one of the following:
   - Choose **Max/Value** to compute the maximum field value on the geometry.
   - Choose **Max/Position** to identify the point at which this value occurs.

Use the **Eval** command to display the actual maximum field value or the coordinates of the point where it occurs.

Gradient Command

Takes the gradient (∇) of the scalar quantity in the top register.

Ln Command

Takes the natural logarithm (base e) of the scalar quantity in the top register.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Log Command

Takes the logarithm (base 10) of the scalar quantity in the top register.

Takes the mean of the quantity in the top register.

Std Command

Takes the standard deviation of the quantity in the top register.

Vector Commands

The Vector Column contains calculator operations which can only be performed on vector stack entries. Drop-down menus in this column are Scal? (convert vector to scalar) and Unit Vec (create unit vector). Standard vector algebra operations (Dot, Cross, etc.) are also present.

Use these commands to perform operations on vector quantities.

Scal?	Replaces the vector in the top register with a scalar quantity whose value is a component of the vector, your choice of ScalarX, Scalar Y, or ScalarZ.
Matl	Multiplies or divides the vector field quantity in the top register by a material property, or if you select MassDensity as the material property, produces a scalar that operates like a named variable.
Mag	Takes the magnitude of the vector quantity in the top register. The magnitude of a complex vector is defined to be the length of the real vector resulting from taking the modulus of each component of the original complex vector.
Dot	Takes the dot product of the vector quantities in the top two registers.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cross	Takes the cross product of the vector quantities in the top two registers.
Divg	Takes the divergence of the vector quantity in the top register.
Curl	Takes the curl of the vector quantity in the top register.
Tangent	Computes the tangential component of a vector quantity along a line
Normal	Computes the normal component of a vector quantity on a surface such as a cutplane or object surface.
Unit Vec	Computes the normal or tangent unit vector. The unit vector is a "wild card" entry. The context is specified at the time of plotting, integrating, or report generation. The drop-down lets you select from Tangent, Normal, or CoordSys[X, or Y, or Z]
X Form	Transforms to your choice of Cylindrical or Spherical.

Scal? Command

Replaces the vector in the top register with a scalar quantity whose value is a component of the vector. Choose from the following:

ScalarX	Returns the x-component of the vector.
ScalarY	Returns the y-component of the vector.
ScalarZ	Returns the z-component of the vector.

Matl Command

Operates on the vector field quantity in the top register of the Fields Calculator based on a material property. At each tetrahedron, the field quantity is operated on by the value of the selected material property — taking the different material attributes of each object into account.

To operate on a vector quantity by a material property:

1. Click **Matl**.

   The *Material Operation* window appears.

2. Select a material property. Available properties are

Permittivity (epsi)	The permittivity, ε.
Permeability (mu)	The permeability, μ. Defined as $\mu_0\mu_r$ where μ_0 is the permeability of free space.
Conductivity	The conductivity, σ.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Omega (w)	The angular frequency, ω. The angular frequency is equal to $2\pi f$, where f is the frequency at which the solution was generated.
MassDensity	This is based on the value of the Mass Density material property. MassDensity is treated like a named expression. Selecting MassDensity disables the Operation radio buttons for Multiply or Divide in the *Material Operations* dialog box.

> **Warning:** The Matl command is not compatible with nonlinear material properties.

3. For Permittivity, Permeability, Conductivity or Omega, select an operation — **Multiply** or **Divide**. Selecting MassDensity disables these operations. The MassDensity scalar can be used like any other named expression.
4. Choose **OK** to operate on the field quantity by a material property or **Cancel** to stop the operation. If you selected MassDensity and click **OK**, a scalar named expression MassDensity is pushed onto the stack.

Mag Command

Takes the magnitude of the vector quantity in the top register. The magnitude of a complex vector is defined to be the length of the real vector resulting from taking the modulus of each component of the original complex vector.

With a complex vector on the calculator stack, the **Mag** button returns a non-negative scalar. In previous software versions, this command returned a complex scalar.

Dot Command

Takes the dot product of the vector quantities in the top two registers.

Cross Command

Takes the cross product of the vector quantities in the top two registers.

Divg Command

Takes the divergence of the vector quantity in the top register.

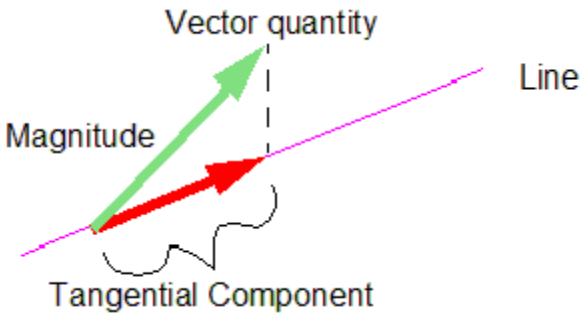
Curl Command

Takes the curl of the vector quantity in the top register.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tangent Command

Computes the tangential component of a vector quantity along a line.



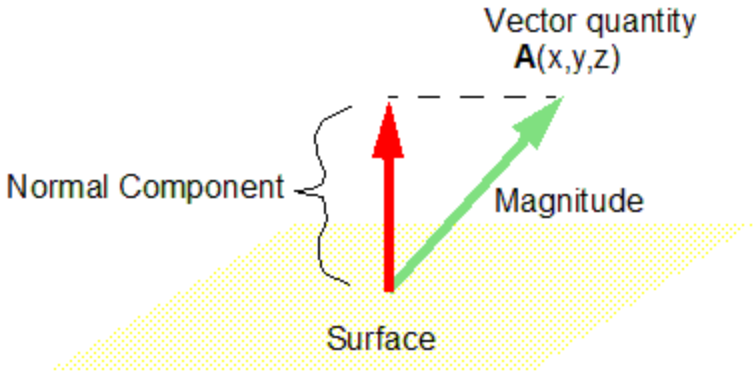
To take the tangent of a vector:

1. Load a vector quantity into the top register.
2. Load a line into the top register using the **Geometry/Line** command.
3. Click **Tangent**.

Normal Command

Computes the normal component of a vector quantity on a surface such as a cutplane or object surface. This is the equivalent of taking the dot product of the quantity with the surface's unit normal vector:

$$Normal = A(x, y, z) \cdot \hat{n}$$



To take the normal of a vector:

1. Load a vector quantity into the top register.
2. Load a surface into the top register using the **Geometry/Surface** command.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **Normal**.

> **Note:**
>
> Because surface normals of sheets are not well defined the fields calculator can produce incorrect results if an expression is evaluated on a sheet. To enforce the correct direction of the surface normal of a sheet, a faceted 3D object (such as a box) can be defined such that one of its planar faces is coincident with the sheet. Because surface normals of a valid object are always defined in an outward direction in Maxwell, the fields calculator uses the surface normal of the face of the 3D object that is coincident with the sheet.

Unit Vec Command

Computes the normal or tangent unit vector. The unit vector is a "wild card" entry. The context is specified at the time of plotting, integrating, or report generation.

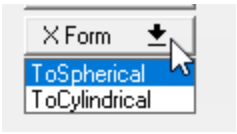
Select from the following:

Tangent	Computes the unit vector tangent to the line specified at the time of plotting, integrating, or report generation based on the context.
Normal	Computes the unit vector normal to the surface specified at the time of plotting, integrating, or report generation based on the context.
CoordSys (X)	Computes the unit vector in the X-dimension of the relative coordinate system in the top register of the calculator stack. Add the relative CS as a geometric object using the **Geometry/Coord** command.
CoordSys (Y)	Computes the unit vector in the Y-dimension of the relative coordinate system in the top register of the calculator stack. Add the relative CS as a geometric object using the **Geometry/Coord** command.
CoordSys (Z)	Computes the unit vector in the Z-dimension of the relative coordinate system in the top register of the calculator stack. Add the relative CS as a geometric object using the **Geometry/Coord** command.

X Form

Fields are defined in the Cartesian coordinate system. In many applications it is much more convenient to use a cylindrical or spherical coordinate system – for example, when the object is of cylindrical or spherical shape. The **XForm** command allows the user to transform vectors into either the cylindrical or spherical coordinate system. Selecting **XForm** displays a selection menu for the coordinate system (**ToCylindrical** or **ToSpherical**):

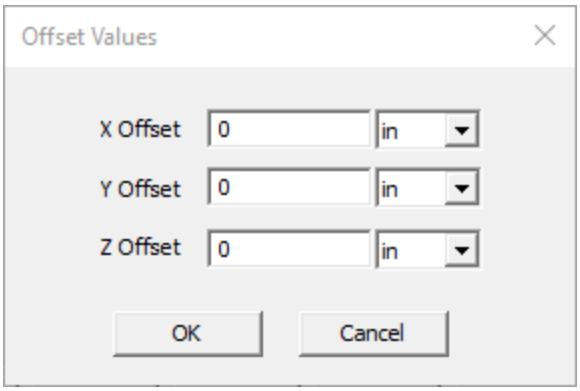
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



ToSpherical – Transforms the vector in the top register of the calculator stack into the spherical coordinate system.

ToCylindrical – Transforms the vector in the top register of the calculator stack into the cylindrical coordinate system.

After you select the coordinate system you want, the **Offset Values** dialog displays.



Here you enter X, Y, and Z offsets to be used as the origin of the new coordinate system.

Click **OK** to compute the offset coordinate transformation to apply to the (complex) vector quantity in the top register on the calculator stack based on your selection of Cylindrical or Spherical coordinates and your input of X, Y, and Z offsets. The new expression will be pushed onto the stack. This provides the origin of the new coordinate system. It will be something like

CVc : ToCylindrical(<Ex,Ey,Ez>,offset<0mil,0mil,0mil>)

or

CVc : ToSpherical(<Ex,Ey,Ez>,offset<0mil,0mil,0mil>)

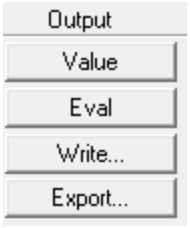
After you transform the quantities, you can then access the vector components in the Fields Calculator.

You will then use the ScalarX/ScalarY/ScalarZ commands under Scal?. For a vector converted to cylindrical coordinates, ScalarX gives the radial component, ScalarY gives the phi (polar) component, and ScalarZ gives the Z component. For a vector in spherical coordinates, ScalarX gives the radial component, ScalarY gives the phi (polar) component and ScalarZ gives the theta (azimuthal) component.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output Commands

The Output Column contains those calculator operations that result in final data outputs from calculations. The Eval button obtains final numerical results from the last stack placeholder (such as integrations).



Use these commands to compute or evaluate expressions and to output the data in the calculator.

Value command	Computes the value of a field quantity on a geometry.
Eval command	Numerically evaluates and displays the results of calculator operations from the last stack placeholder, such as integrations.
Write command	Saves the contents of the top register to a disk file for future re-use.
Export command	Saves field quantities in a format that can be read by other modeling or post-processing software packages.

To plot/tabulate the extracted data using create report or create field overlays, you should use a named expression.

Value Command

This computes the value of a field quantity at a point. Use it to find:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The magnitude of a scalar field quantity at that point.
- The x-, y-, and z-components of a vector field quantity at that point.

The **Value** command applies the value of the next-to-top stack entry (a field quantity) on the geometry entry at the top stack entry. The field quantity entry may be scalars, real or complex vectors. The geometry entry may be of any type (point, surface, line, or volume).

For example, if the top stack entry in the Field Calculator is a surface and the second stack register is for the real part of the E Vector ("Real(<Ex, Ey, Ez>)"), the resulting quantity will be an expression for the real part of the E vector on the surface (a VecSrf quantity). Again, the result of this command is still a placeholder expression, not the final numerical values.

Strictly speaking, the Value command provides access to the field quantity, not data outputting. It simply provides a handle to the (numerical) values of a field quantity on a given geometry, and thus the outcome of the command is not numerical values themselves. If the numerical value of a field quantity is desired, you will need to follow it up with one of the three other output commands, or define a named expression for the quantity and plot it with the Reporter.

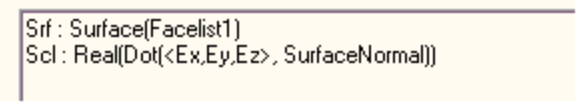
To find the value of a field quantity at a point:

1. Load the field quantity into the top register, and perform any needed operations on it.
2. Load the appropriate point into the calculator using the **Geometry/Point** command.
3. Click **Value**.

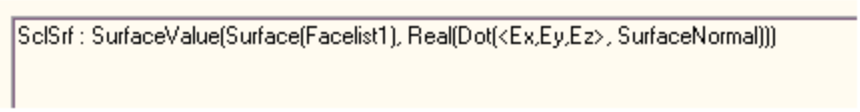
To view the numerical results of this operation, use the **Eval** command.

You can also use the **Value** command to access the intermediate SurfaceValue function.

For example, after inputting an expression for a quantity, such as an E field, and then selecting a surface geometry, the calculator stack displays something like this:



Clicking the **Value** command changes the display to the following, showing the intermediate SurfaceValue function.



In this case, SurfaceValue provides the x, y, z, coordinates of the FEM mesh and Lagrangian points so you can use **Write** to generate an .fld file containing an evaluated scalar quantity at those points.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

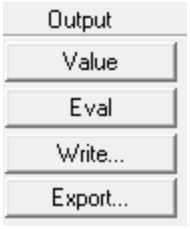
In general for **Value**:

1. Enter any quantity onto the stack.
2. Enter a volume / surface / line / point onto the stack.
3. Press the **Value** button and you will get an appropriate geometry value on the stack.

Now you can perform suitable operations such as Write, Integrate, etc. For PointValue you can also do **Eval**.

Eval Command

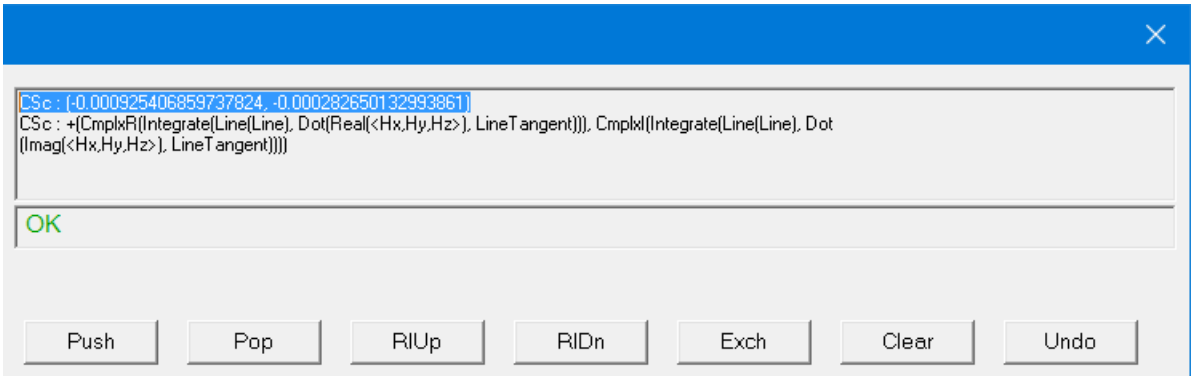
The **Eval** command finalizes computations and converts the text-string indicating the computation being performed to an actual numerical value. This command numerically evaluates and displays the results of calculator operations such as integrations, maximum or minimum field computations, field values at points, and so forth. For this command to work properly, the stack entry must be reduced to a single-value entry which can be a scalar, complex, or vector (real or complex). The quantity to be evaluated must be in the top register. The **Eval** command computes the numerical results of the operation, which replace the contents of the register. Units of the numerical value are not provided.



For a calculator expression that evaluates to a single value, the **Eval** command offers a quicker way to obtaining the final numerical result than defining a named expression and then trying to plot/tabulate it in the reporter. But for many expressions that don't evaluate to single values, such as the wave impedance along a line mentioned earlier, **Eval** command does not apply. As such, use of named expression and the Report Editor will be the only option.

Evaluate the expression for voltage along a line shows the definition of an expression for the voltage between two conductors by integrating the electric field along a line using the recipe given for Calculating the Current along a Wire or Trace. Then, by clicking the Evaluate command in the Output column, we obtain the numerical value for the voltage as shown in the top of the stack. It can then be either written down or copied/pasted to other programs.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



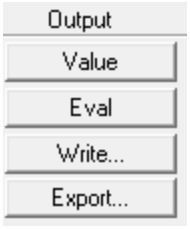
$$I = \oint H \cdot dl$$

Because **H** and **I** are complex quantities, you will need to evaluate the real part of **H** to obtain the real part of **I**, then evaluate the imaginary part of **H** to obtain the imaginary part of **I**. To do this:

1. Load **H** into the calculator using the **Qty** command.
2. Take the real part of **H** using the **Cmplx/Real** command.
3. Load the rectangular loop using the **Geom/Line** command. Create the loop, a closed polyline, to integrate over.
4. Click **Tangent** to get the component of **H** along the line.
5. Take the integral around the loop using the ∫ command.
6. Click **Eval** to evaluate the integral. The real part of **I** appears in the top register.
7. Repeat this process using the imaginary part of **H** (found with the **Cmplx/Imag** command) to obtain the imaginary part of **I**.

Write Command

If the calculator operations performed have obtained a stack entry that is intended for use in still other calculator operations, the stack entry can be saved for this purpose by using the **Write** button in the calculator Output column.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that in this case no name will be requested for the expression; only a save filename location will be required. This function will not work for field values derived upon a specific geometric quantity (those containing either Lin, Srf, or Vol in the stack data type indicator) as the calculator cannot know that these geometric quantities exist in identical forms in other postprocessing sessions.

When you try to read a file written with the **Write** command by clicking the **Read** button in the Input column, you must ensure that the mesh used for generating the current solution is the same as the one with which the saved field data is generated. Since the two results must be generated on the same mesh, not only the geometries of the designs on which the **Write/Read** commands were applied must be the same, they must also have gone through the same adaptive meshing process. In practice, this limits the use of Write/Read command to solutions for different frequency sweeps of the same analysis (adaptive) setup of the same design.

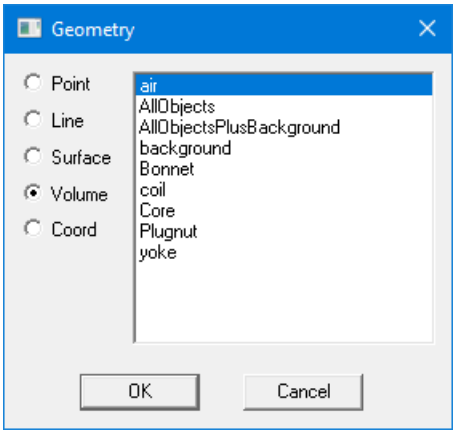
This command saves the contents of the top register to a disk file. Use this command to:

- Save registers for use during a later postprocessing session.
- Save a field quantity for use when postprocessing a different model.

> **Note:**
>
> Use of the Global Coordinate System is assumed. Local coordinate systems are not used.

To save a register:

1. Click **Write**.
2. If the register includes numeric data with a constrained quantity (such as jsurf), you see a dialog box that gives a choice of constraining geometries. For example:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select the geometry of interest, and select **OK**.

   This displays a file browser.

4. Use the file browser to specify the register's file name and directory path. A .reg extension is automatically assigned to register files and a .fld extension is assigned to field files. You can choose to save both .reg and .fld files, or either one.
5. Click **OK**.

   The contents of the register are saved to the file you specified.

Export Command

To output a field quantity or calculation result for use by some third-party post-processors, use the **Export** button in the calculator Output column. You can map the field quantity to either a customized grid of points specified by a points file or a three-dimensional Cartesian, Cylindrical, or Spherical grid specified interactively through a dialog box. In the latter case, you must specify the dimensions and spacing of the grid in the coordinate system with units. For Cylindrical and Spherical coordinate systems, you can also specify an offset from the origin.

The **Export** command opens the **Export Solution** dialog box, from which you can export the field quantity in the top register to a file, mapping it to a grid of points.

Use this command to save field quantities in a format that can be read by other modeling or post-processing software packages. Two options are available for defining the grid points on which to export:

Input grid points from file	Maps the field quantity to a customized grid of points. Before using this command, you must create a file containing the points and units.
Calculate grid points	Maps the field quantity to a three-dimensional Cartesian grid. You specify the dimensions and spacing of the grid in the Cartesian, cylindrical, or spherical coordinates, with units that you specify. The initial units are taken from the model.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Use of the Global Coordinate System is assumed. Local coordinate systems are not used.

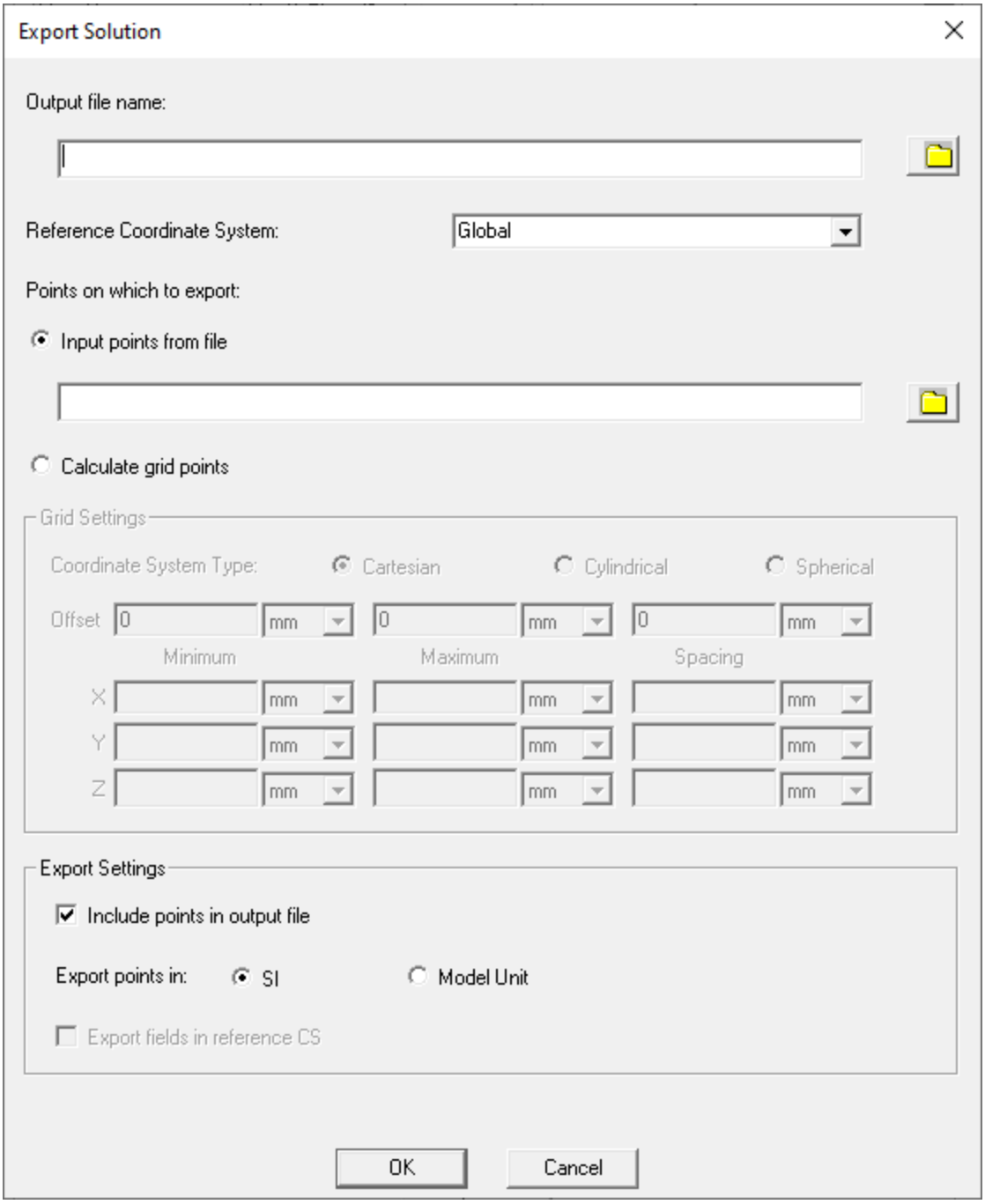
To export a field quantity to a customized grid:

1. Load the quantity into the top register for the fields calculator, and perform any operations on it.
2. If desired, load a volume using the **Geometry** command.

   You can use the Domain command to limit the calculation to the volume you specify. If you export a Domain filtered numeric, points that are filtered out by the domain will not be written out.
3. Click the **Export** button in the Fields Calculator.

   This opens the **Export Solution** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Type or select the name of the file in which the field quantity is to be saved in the **Output File Name** text box. You can use the file icon to open the file browser to specify the file name and directory path. The **.fld** extension is automatically assigned to this file.
5. If the design contains multiple coordinate systems, a drop-down menu lets you select the coordinate system to use. This can be helpful in evaluating the fields output when relative

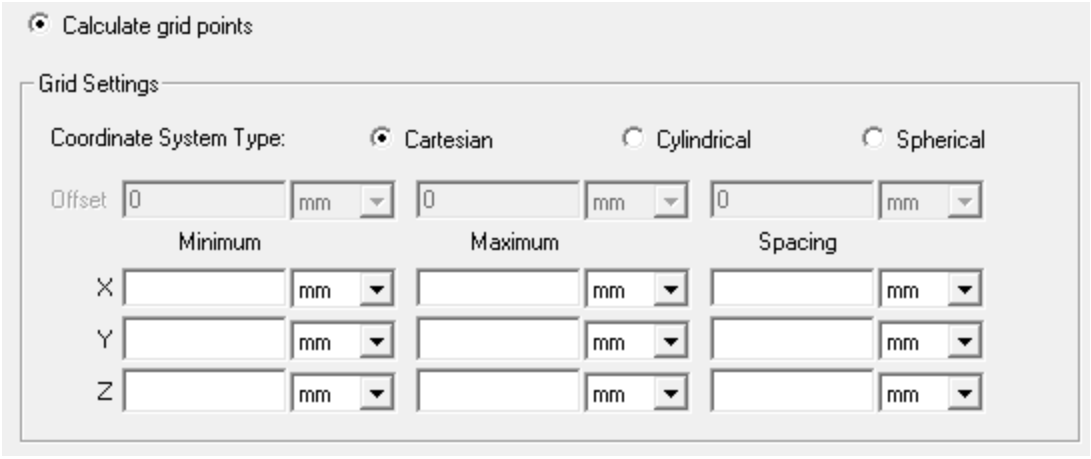
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

coordinate systems used and unitized in the design.

6. Click either the **Input grid points from file** radio button if you have a created a .pts file containing the grid points, or click the **Calculate grid points** radio button.
   - If you select **Input grid points from file**, either type the name and directory of the file containing the points on which the field is to be mapped, or, click on the file icon and use the file browser to locate the point file (**.pts** extension).

     > **Note:**
     >
     > The **.pts** file should contain the units to use for the export as shown in this file stub:
     >
     > Unit=mm
     >
     > -5.5 -5.5 -5.21475
     >
     > -5.5 -5.5 -5.14425
     >
     > -5.5 -5.5 -5.07375
     >
     > -5.5 -5.5 -5.021

   - If you select **Calculate grid points** button, you can specify the coordinate system as Cartesian, Cylindrical, or Spherical.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cartesian: for each grid dimension on **X**, **Y**, and **Z**, enter the Minimum, Maximum, and grid point spacing.

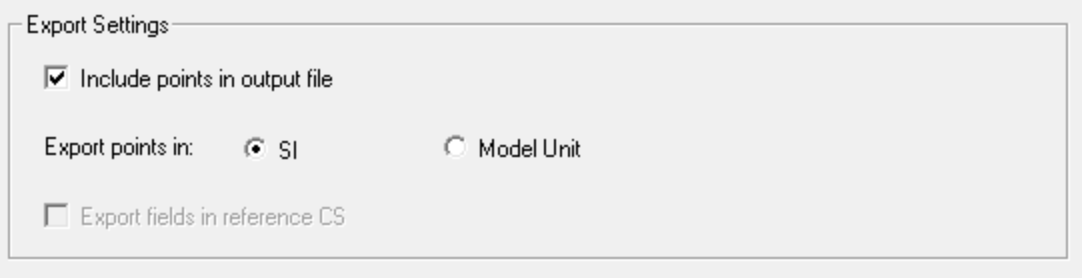
Cylindrical: for each dimension Rho, Phi, and Z, enter the Minimum, Maximum, and grid point Spacing. You can also specify an origin of Offset.

Spherical: for each dimension R, Theta, and Phi, enter the Minimum, Maximum, and grid point Spacing. You can also specify an origin of Offset.

> **Note:** When you export fields on a 1D or 2D line/surface from the field calculator, the start and stop values must be the same for one or two of the coordinate system start/stop ranges. If you specify a zero spacing for a dimension, the export uses only the minimum value.

The default coordinate system will be Cartesian. The default offset will be all zeroes. The length units will default to model unit and default angle unit will be degree. At the start the minimum/maximum/Spacing entries are blank. The user entered values are not remembered when the dialog is closed.

7. For larger files, for the Export Settings, you may want to uncheck the **Include points in output file** box. If you uncheck the box, the file header will include minimum, maximum and spacing information from which you can recalculate the grid points.
8. You can also specify SI or Model Units. Model Units can be helpful to evaluate the fields output when relative coordinate systems are unitized in the design.



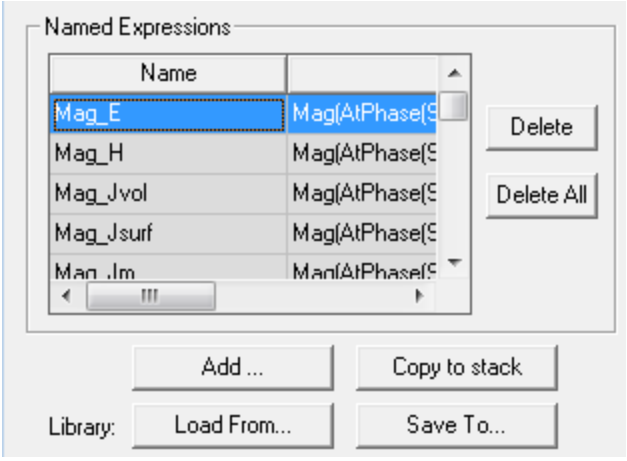
9. If field to be exported has type vector or complex vector, the 'Export Field in reference CS' checkbox is enabled and you can specify that field to be exported in reference CS. It can be helpful to evaluate the fields output when rotated coordinate systems are unitized in the design
10. Click **OK** to export the file.

    The field quantity is mapped to the grid and saved to the file you specified (**.fld** extension).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculating Derived Field Quantities Using Named Expressions

The **Named Expressions** panel displays expressions that can be included in register definitions by name. You can add additional expressions to the Named expression list by creating the expression in the register display area and then clicking the Add button. This lets you add to the Named expression library.



Click on a named expression to select it. When a named expression has been selected, the **Copy to Stack** button is activated. Click **Copy to Stack** to push the expression on the top of the stack.

When a Maxwell design is open and a Solution Setup has been performed, the following predefined named expressions are available.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Expression Name	Expression Definition
Mag_E	Mag(AtPhase(Smooth (<Ex,Ey,Ez>),Phase))
Mag_E_t	Mag(Smooth(<E_tx,E_ty,Etz>)). Used with Transient projects.
Mag_H	Mag(AtPhase(Smooth (<Hx,Hy,Hz>),Phase))
Mag_H_t	Mag(Smooth(<Hx,Hy,Hz>). Used with Transient projects.
Mag_Jvol	Mag(AtPhase(Smooth (<JVx,JVy,JVz>),Phase))
Mag_Jsurf	Mag(AtPhase(Smooth (<Jsurfx,Jsurfy,Jsurfz>),Phase))
Mag_Jsurf_t	Mag(<Jsurf_tx,Jsurf_ty,Jsruf_ tz>). Used with Transient projects.
ComplexMag_E	Mag(CmplxMag(Smooth (<Ex,Ey,Ez>))
ComplexMag_H	Mag(CmplxMag(Smooth (<Hx,Hy,Hz>))
ComplexMag_Jvol	Mag(CmplxMag(Smooth (<JVx,JVy,JVz>))
ComplexMag_Jsurf	Mag(CmplxMag(Smooth (<Jsurfx,Jsurfy,Jsurfz>))
Vector_E	AtPhase(Smooth (<Ex,Ey,Ez>),Phase)
Vector_E_t	(Smooth(<E_tx,E_ty,E_tz>). Used in Transient projects.
Vector_H	AtPhase(Smooth (<Hx,Hy,Hz>),Phase)
Vector_H_t	(Smooth(<H_tx,H_ty,H_tz>). Used in Transient projects.
Vector_Jvol	AtPhase(Smooth (<JVx,JVy,JVz>),Phase)
Vector_Jsurf	AtPhase(Smooth(<Jsurf_ x,Jsurf_y,Jsurf_z>),Phase)
Vector_Jsurf_t	(Smooth(<Jsurf_tx,Jsurf_ ty,Jsurf_tz>). Used in Transient projects.

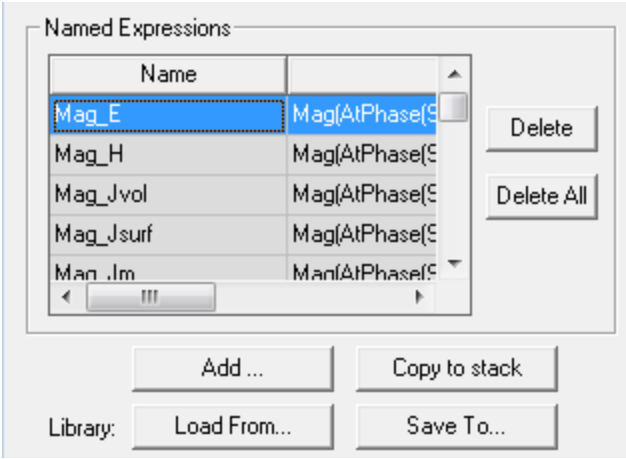
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Vector_RealPoynting	Real(Poynting)
VectorReal_Poynting_t	Poynting_t. Used in Transient projects.
Local_SAR	LocalSAR
Average_SAR	AverageSAR
Surface_Loss_Density	SurfaceLossDensity. See further discussion here.
Volume_Loss_Density	VolumeLossDensity. See further discussion here.
Volume_Loss_Density_t	VolumeLossDensity_t. Used in Transient projects.
Surface_Force_Density	Surface Forces exist when one side is conductor, but the other is not, or finite conductivity and layered impedance boundary. This is mainly for the purpose of mapping surface force density in Maxwell to Workbench Mechanical.
Mag_Displacement	Mag(Smooth(<Ux,Uy,Uz>)). Magnitude of displacement, used with Workbench in projects exploring stress feedback.
Displacement_Vector	Smooth(<Ux,Uy,Uz>). Used with Workbench in projects exploring stress feedback.

Named Expression Library

The named expression library in the Fields Calculator provides a way to conveniently calculate frequently used quantities. Some standard field quantities have been predefined and are always accessible. These field quantities have a gray background color and cannot be deleted. You can assign a name to the top entry of Data Stack and add it in the Named Expression list. The library comes with several predefined expressions. Once these new expressions are added/loaded, they are available through Maxwell's standard post-process capabilities. For example, they can be copied back to the Data Stack in calculator by **Copy to Stack** for further operations, or be plotted using Report Editor and Field Overlay.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can combine calculator Input commands in any legal fashion, including complex quantities, to produce new named expressions.

- Adding named expressions to the Fields Calculator expression library
- Copying named expressions to the Calculator Stack
- Saving named expressions to a Personal Library
- Loading named expressions
- Deleting named expressions that you added

> **Note:** Named expressions can be displayed and entered in stack format or algebraic format depending which option you select in the Fields Calculator window; see Using the Fields Calculator for more details.

To add a named expression of your own to the Fields Calculator list:

1. In the register display area, create the expression by using the calculator Input commands.

   You can combine input commands in any legal fashion, including the use of complex quantities. If you select an input command that is not legal for a current operation, you receive an error message.

2. When you finish creating the expression, click **Add** in the *Named Expressions* panel.

   The *Named Expression* dialog box appears.

3. Type a name for the expression in the **Name** text box.

   The new expression is added to the list of named expressions.

To copy a named expressions to the Calculator Stack:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You can scroll through the list, select any desired named expression, and click **Copy to Stack** to move it to the calculator stack, where you can use it to generate calculated outputs.

To delete named expressions that you added:

When the *Named Expression* list contains one or more user-defined expressions, the **Delete** and **Clear All** buttons are active. (You cannot delete or clear the predefined named expressions.)

- To delete the selected user-defined named expression, click **Delete**. To delete all user-defined named expressions, click **Clear All**.

To save one or more named expressions for the Fields Calculator to a personal Library:

1. Click **Save To** on the Fields Calculator.

   The *Select Expressions for Saving* dialog box displays.

2. If any new named expressions exist, you can select one or more to save to a file.
3. Specify a file name and click **OK** to save the file.

To load named expressions for the Fields Calculator from a personal library:

1. From the Fields Calculator, click **Load From**.

   This displays a file browser that you can use to search for existing .clc files.

2. Select the library to load and click **OK**.

   This loads the expression file you have selected.

Exiting the Fields Calculator

Click **Done** to exit the Fields Calculator.

Plotting the Mesh

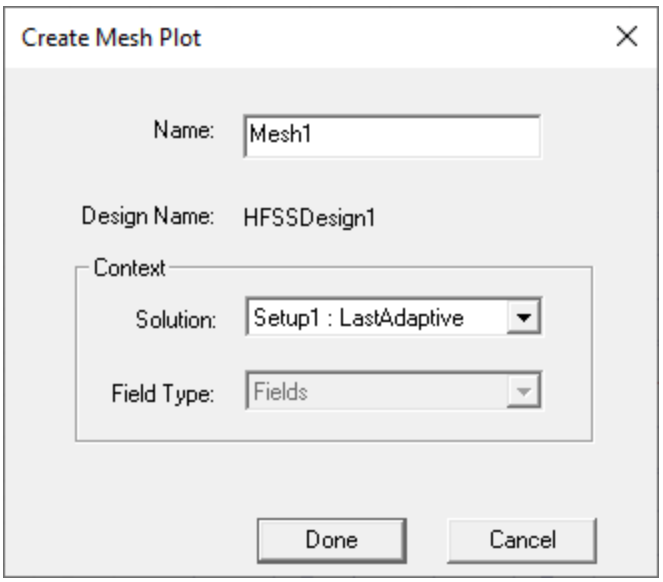
Before or after the solution is complete, you can plot the finite element mesh on surfaces or within 3D objects. For an HFSS or Maxwell design that includes Layout Components, you can also select nets and layers from the layout components to plot.

1. Select one or more surfaces or objects to create the mesh plot on or within. If the design is HFSS and includes Layout Components, you can select the surfaces either before or after selecting **Plot Mesh**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

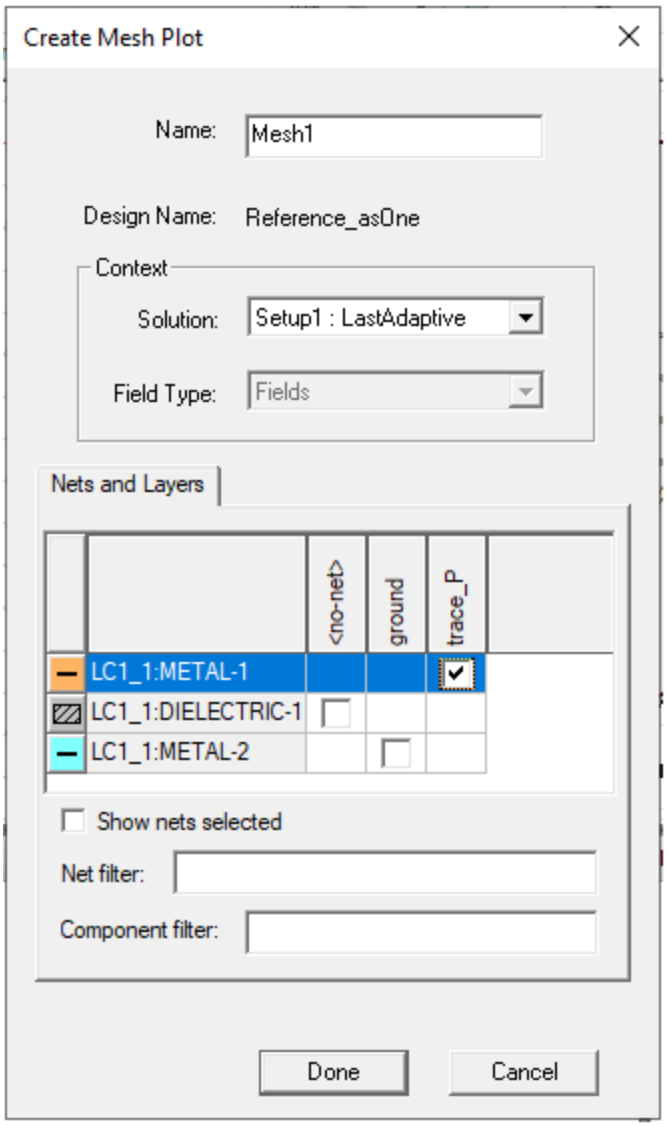
2. Select **Plot Mesh** in one of two ways:
    - Click **Maxwell** > **Fields**> **Plot Mesh**.
    - In the Project Manager, right-click **Field Overlays** and select **Plot Mesh...**

    The *Create Mesh Plot* dialog box appears.



    If the HFSS design includes Layout Components, the **Create Mesh Plot** dialog box includes a table of Nets and Layers from which you can make selections.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This **Nets and Layers** section displays all the layers and nets of every layout component in the design. Upon opening, it adjusts its size to try to show all the rows and columns for the layout components. You can resize the parent dialog. For objects in the component belonging to a specific layer and net, you can select a checkbox. By clicking the column headers, you can select or unselect checkboxes for an entire column. You can toggle the **Show nets selected** checkbox to move all the selected nets to the left side of the table. You can use the **Net filter** and **Component filter** fields to filter out specific nets and components. The net and component filters support '*' and '?' wildcards and you can delimit multiple filters with commas. The Nets and Layers table is refreshed each time you update the filter text.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

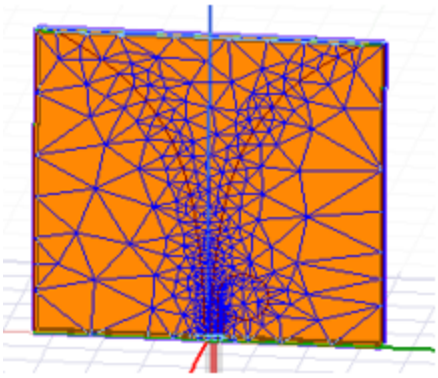
If you right click on the Nets and Layers panel, a short-cut menu appears that lets you Select This Layer Only, Select or Unselect All, Select or Unselect All Signals, or Select and Unselect All Dielectrics.

3. Select the **Solution** to use from the drop-down menu.

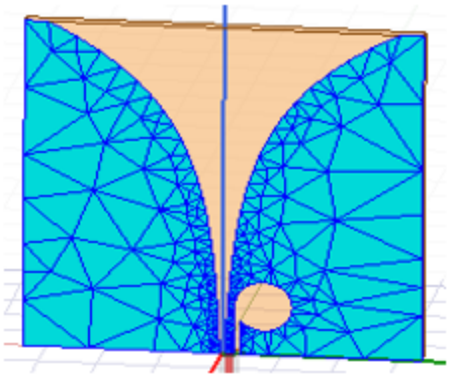
   For HFSS designs that include layout components, see a discussion with example below.

4. If **Field Type** is enabled, you can select from the drop-down menu for a mesh plot.
5. Click **Done**.

   The mesh appears on the surface or object you selected. For example:

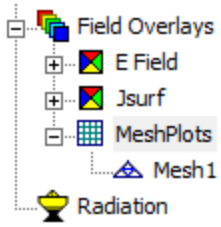


Mesh Plot of All Objects



Mesh Plot of Selected Surfaces

An icon for the mesh also appears in the Project Manager under **Field Overlays** > **Mesh Plots**.



If a solution is ongoing, you can select the **Mesh Plots** icon in the in the Project Tree, right-click to display the shortcut menu, and select **Update Plots**. This updates the mesh plot to include the latest data available. After the last adaptive pass, the mesh plot is automatically updated.

If a mesh includes seeding, these effects appear at the start of the adaptive passes. Any mesh adaptation at the start of a subsequent pass is not plotted until after that pass is completed. This delay ensures that mesh plots and actual solutions remain consistent with each other.

You can modify an existing plot by selecting the plot and changing the properties.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you modify the geometry of an object, the solution is invalidated and the mesh plot is removed.

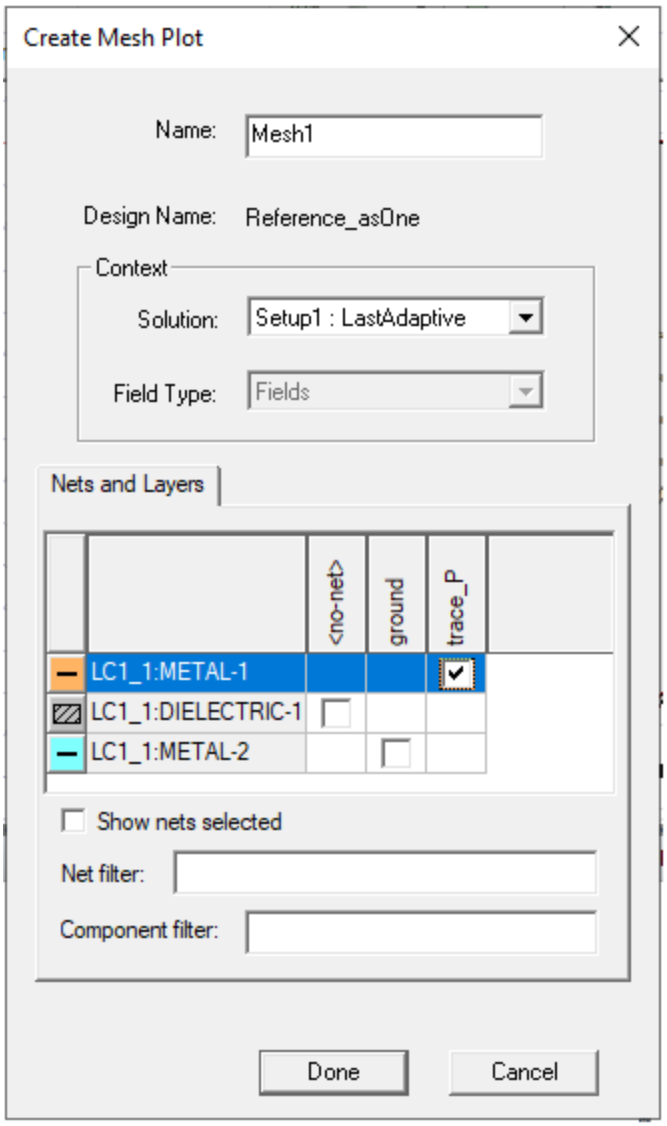
If you have plotted the mesh of a Solve-Inside object, you can view the internal mesh by using either the clip-plane feature or the Model Analysis dialog box.

Also see the discussion here, under **Visualization for Skin Depth Refinement Mesh Results**. The same techniques for visualization for internal mesh plots apply even if seeding is not used.

Example of Mesh Plots for an HFSS Design with Layout Components

First, solve the design. You can either select geometries in the Modeler window before opening the **Create Mesh Plot** dialog, or use the **Nets and Layers** area in the Create Mesh Plot dialog to select objects from the layout components to plot. These objects are represented as combinations of nets and layers and can be selected by checking the corresponding checkboxes.

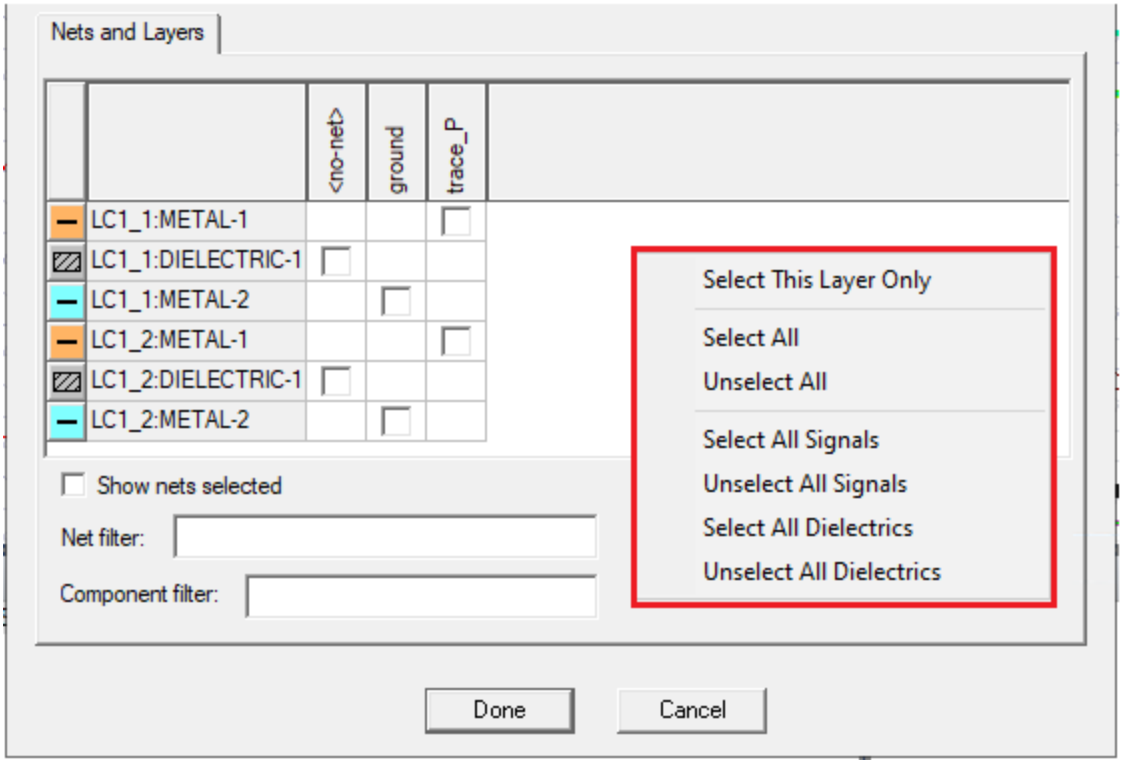
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This **Nets and Layers** section displays all the layers and nets of every layout component in the design. Upon opening, it adjusts its size to try to show all the rows and columns for the layout components. You can resize the parent dialog. For objects in the component belonging to a specific layer and net, you can select a checkbox. By clicking the column headers, you can select or un-select checkboxes for an entire column. You can toggle the “Show nets selected” checkbox to move all the selected nets to the left side of the table. You can use the **Net filter** and **Component filter** fields to filter out specific nets and components. The net and component filters support ‘*’ and ‘?’ wildcards and you can delimit multiple filters with commas. The Nets and Layers table is refreshed each time you update the filter text.

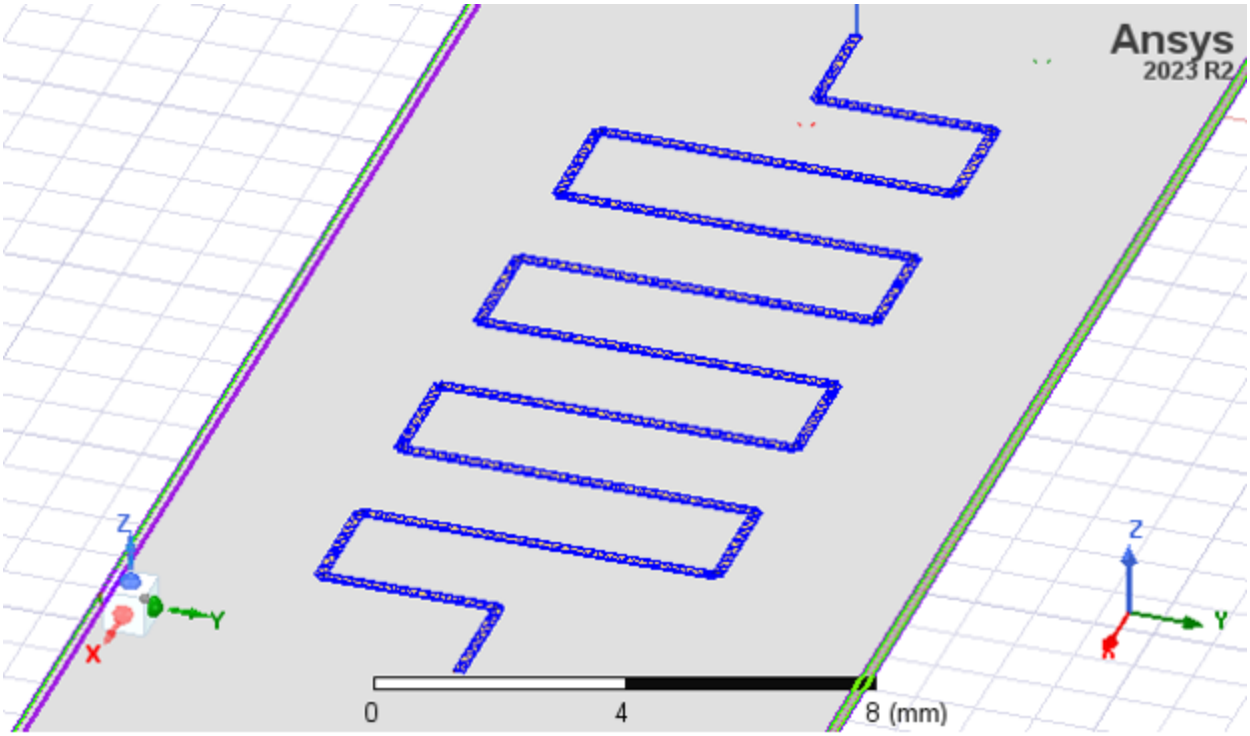
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you right click on the Nets and Layers panel, a short-cut menu appears that lets you Select This Layer Only, Select or Unselect All, Select or Unselect All Signals, or Select and Unselect All Dielectrics.

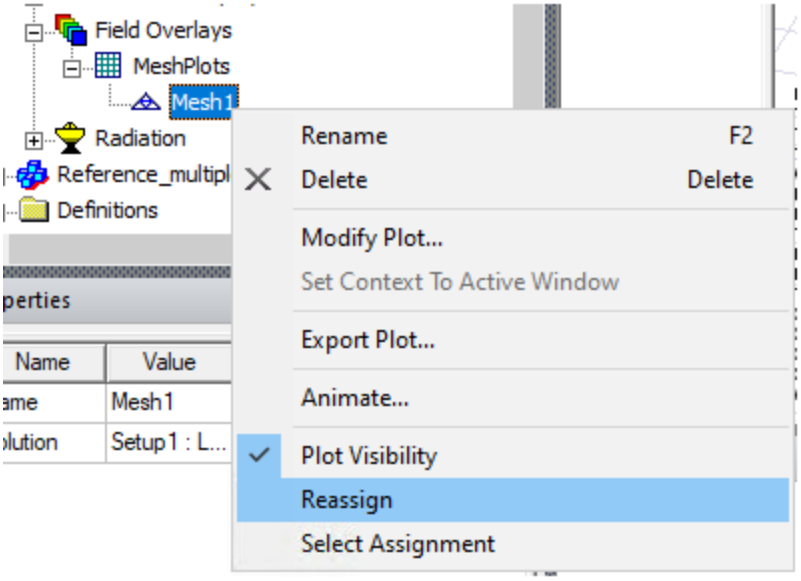


After making selections click **Done**. The mesh plot will be generated on the objects belonging to the selected layer/net combinations.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

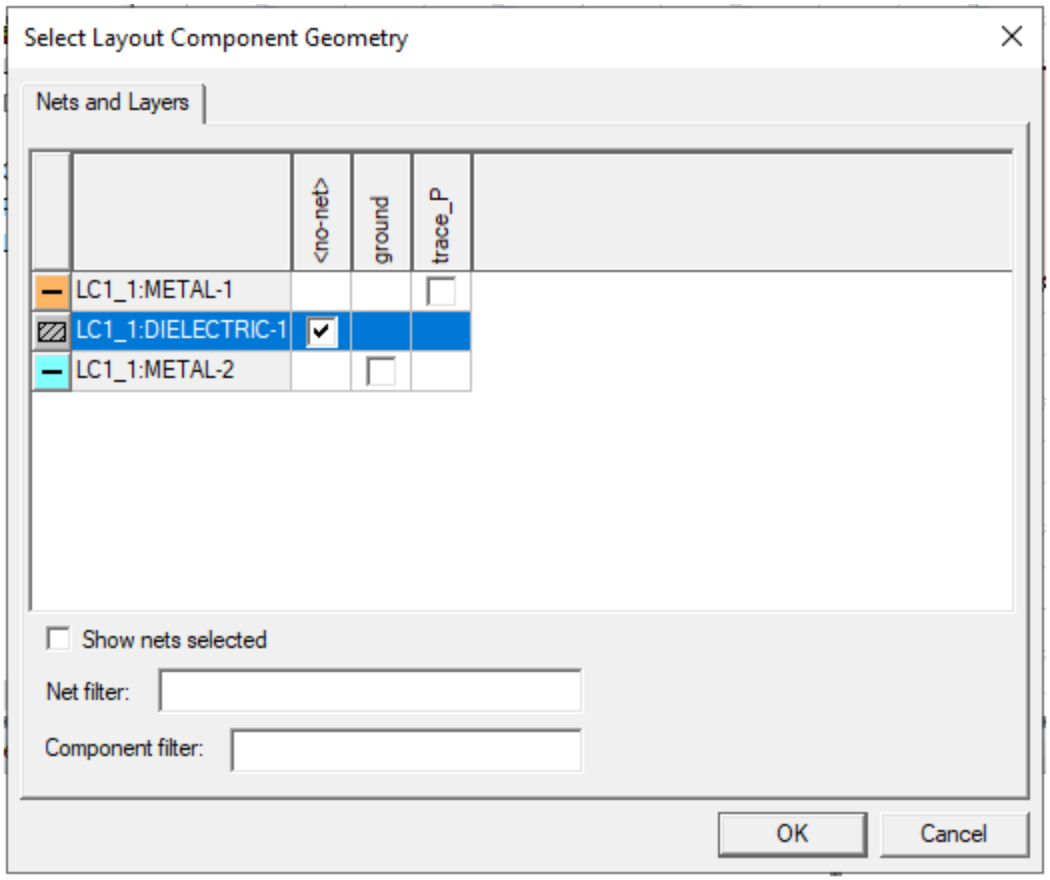


If you want to reassign a mesh or field plot, either select 3D geometries from layout components in the modeler window before selecting **Reassign** in the right-click menu, or select **Reassign**.



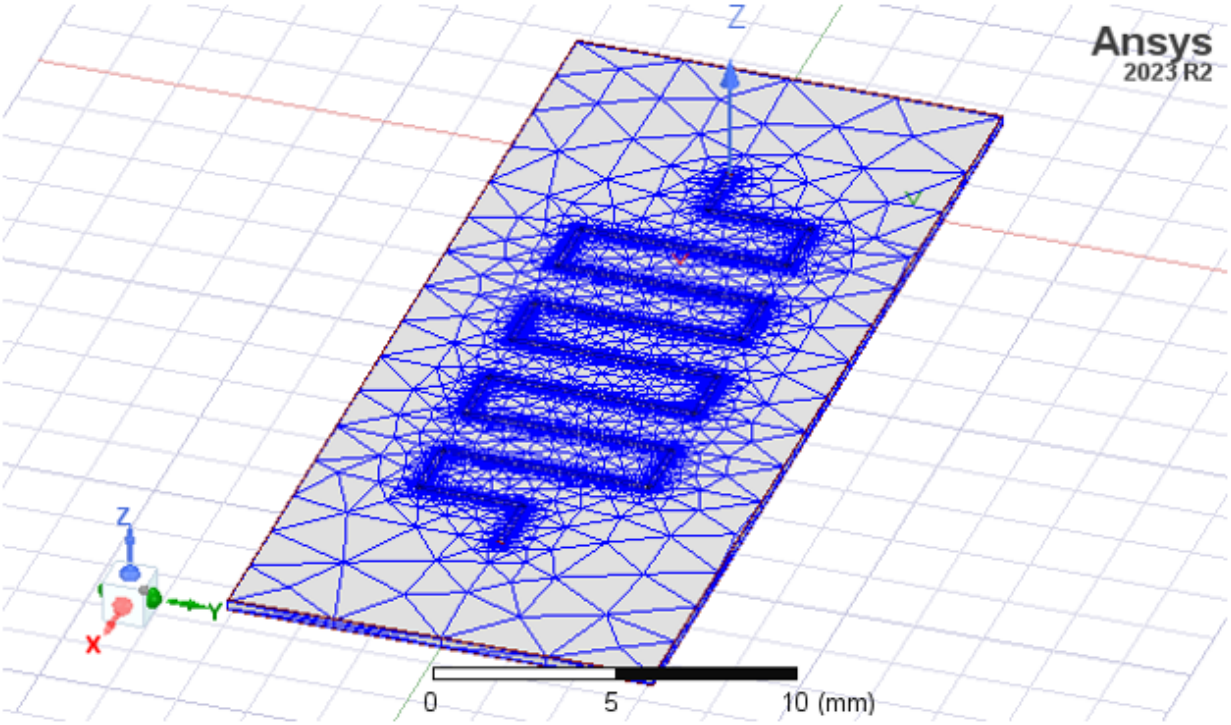
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For HFSS designs with layout components, the **Select Layout Component Geometry** dialog appears. This has the same functionality and menus as the this **Nets and Layers** section of the **Create Mesh Plot** dialog.



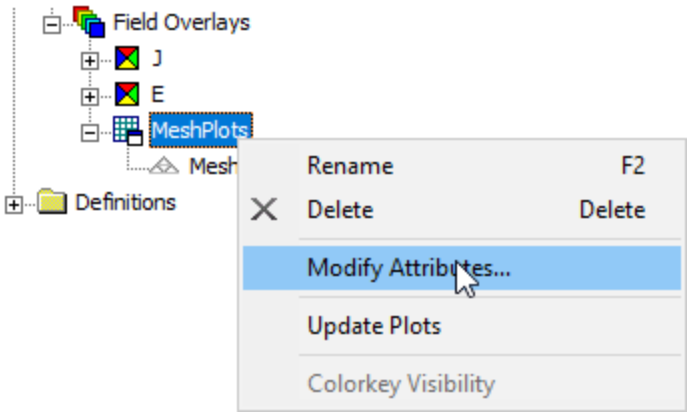
Change the selection in the dialog and click **OK**. The mesh plot will be re-assigned to the newly selected objects, in addition to any 3D geometries previously selected from the modeler window. If you click cancel, the plot is only re-assigned on 3D geometries previously selected. If there are no previously selected 3D geometries, the program issues an error message. Change the selection in the dialog and click **OK** to display the reassigned mesh plot.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



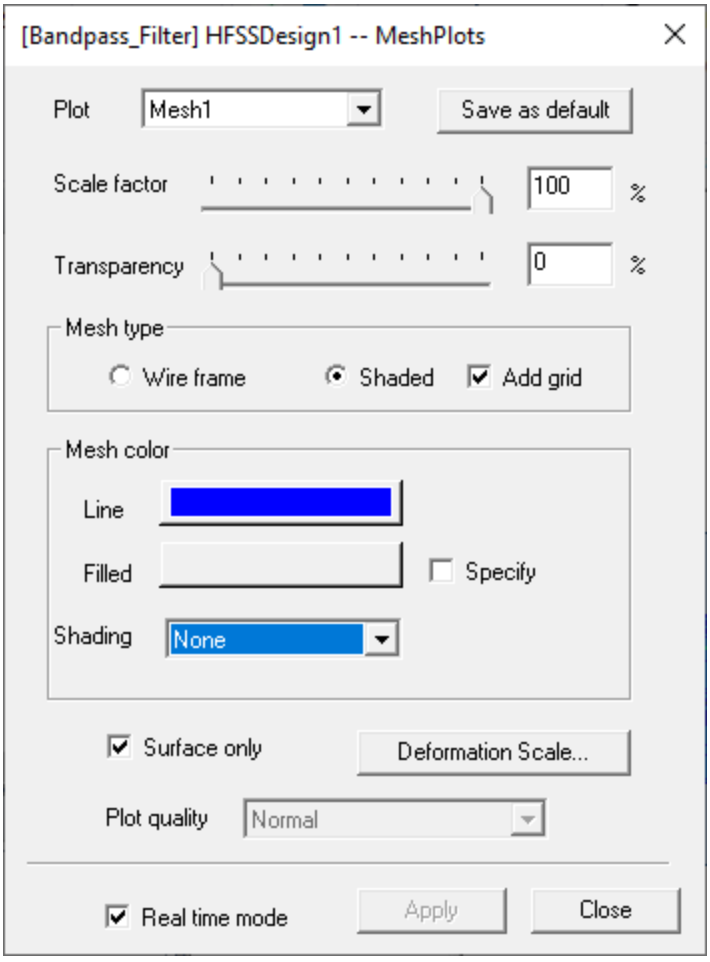
Setting Mesh Plot Attributes

1. In the Project Tree, expand **Field Overlays**.
2. Right-click **Mesh Plots** and select **Modify Attributes**.



The *MeshPlots* window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

You can also open the MeshPlots window from **Maxwell** > **Fields** > **Modify Plot Attributes**. You will be prompted to select the folder containing the mesh you want to modify.

For Mesh plots, the following attributes can be modified:

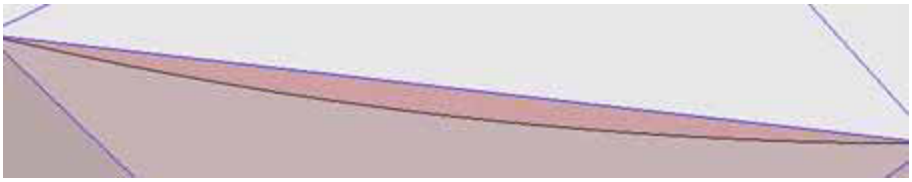
Plot	A drop-down menu of available plots.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scale Factor	The size at which the tets are displayed. Scaling may let you analyze particular situations better. For example, a scale factor of 80% draws the tetrahedra at 80% of their original size. Use the **Scale factor** slider to increase or decrease the size of the tetrahedra.
Transparency	The degree of transparency for the tets. This is useful for viewing objects or plots behind the current plot. Use the **Transparency** slider to increase or decrease plot transparency.
Mesh Type	Whether to display the tets as wire frame or shaded, and whether to Add Grid.
Mesh Color for Line and Fill	The color for the tet edge lines and fill. Clicking the button for each displays a color selection dialog box. You must select **Specify** to enable fill color.
Shading	You can specify Shading as None, Flat, or Smooth, with respect to lighting.
Surface Only	Whether to plot the surface only, or all tets inside selected objects.
Plot Quality	The resolution quality of the plot. Normal by default. Use the drop-down menu to select Coarse, Normal, Fine, or Very Fine. The higher the resolution, the more memory used.
Real Time	Whether to show changes to a mesh immediately in the view window. If this option is cleared, click **Apply** when you want to see the changes.

For **Plot Quality**, higher quality selections permit visualization of curvilinear mesh elements. With the **Coarse** setting, facets do not snap to the midpoint edge:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Normal** setting displays snapping to the midpoint edge:

The **Fine** setting displays more of the curvilinear tets:

The **Very fine** setting gives the most accurate display, but uses the most memory:

3. Click **Save as default** button if you want the tab's settings to apply to mesh plots created after this point.
4. Click **Close**.

Export Layout Field Data

For Maxwell 3D Transient A-Phi designs that include a layout component, you can export field data related to the layout component. Exporting plane surface maximum B-H field data, maximum current density data in each layer, and average current/current density data in all the vias will help users efficiently find where the board or via damage will occur.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

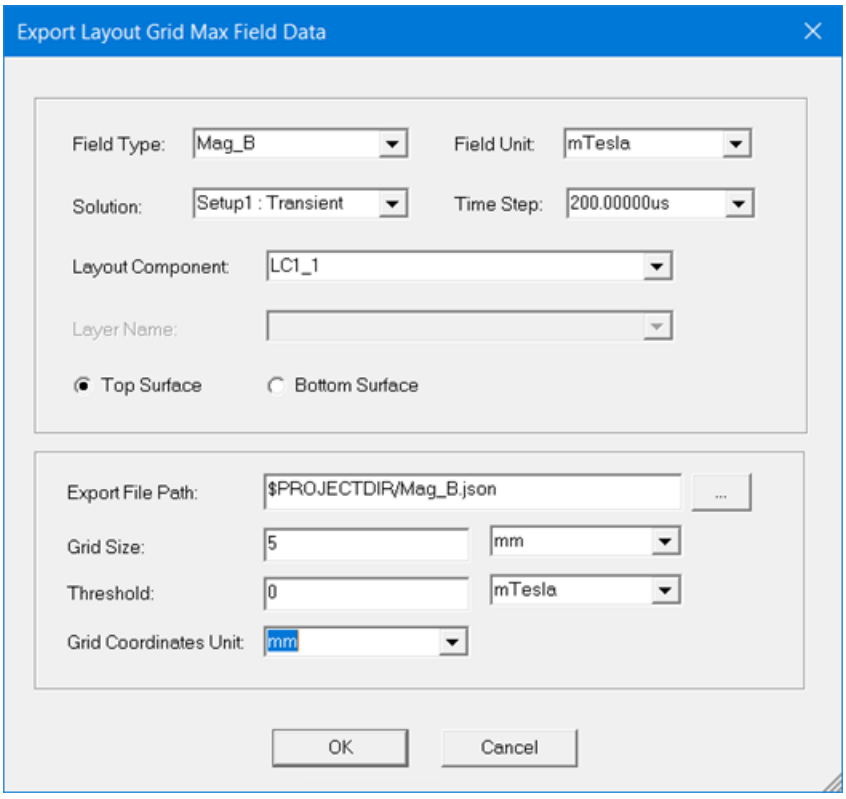
To export layout field data, select **Maxwell 3D** > **Export Layout Data**, or right-click on the **Maxwell3DDesign** entry in the Project Manager tree to select **Export Layout Data**, then select one of the following options:

- Export Grid Max Field Data
- Export Via Average Current Density
- Export Via Current
- Export Layer Induced Eddy Current

> **Note:** The Export Layout Field Data options only support a rigid layout component; they do not support rigid-flex bending PCBs.

Export Grid Max Field Data

The **Maxwell 3D** > **Export Layout Field Data** > **Export Grid Max Field Data** function calculates the maximum field value on the top/ bottom face of layout layers within a user-defined grid in the XY plane. It can calculate the maximum field value for **Mag_B**, **Mag_H**, or **Mag_J**.



After the window is set up and the user clicks **OK**, Maxwell exports the data to the specified file. The output data of each grid cell, layer, and net combination contains

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- the maximum field value of the top/bottom surface in each grid (in user-specified unit),
- the net and layer names,
- the top Y, bottom Y, left X, and right X value (as determined with user-defined length unit).

To set up the Export Grid Max Field Data window, configure the following settings:

1. Click the drop-down arrow beside **Field Type**, and select one of the following: **Mag_B**, **Mag_H**, or **Mag_J**.
2. Click the drop-down arrow beside **Field Unit**, and select the unit for exporting data. Options include **Tesla** and **Gauss**.
3. If there are multiple analysis setups in your design, you must select which setup solution you would like to use for the export. Click the drop-down arrow beside the **Solution** field, and select the desired solution results. If there is only one solution in the design, it will automatically display in the Solution field.
4. Select the **Time Step** value you would like to use for the extraction. The available values are determined by transient setup in the Solve Setup window.
5. From the **Layout Component** drop-down menu, select the layout component instance to export data from. If there is only on layout component in your design, it will be selected by default.
6. For the **Mag_J** field only: select the layer you would like to use for the export: from the **Layer Name** drop-down menu, select the layer to perform the calculations on; use **All_ Layers** for selecting all layers.
7. For **Mag_B** and **Mag_H** only: select **Top Surface** to calculate and export data from the top face of the layer. Select **Bottom Surface** to calculate and export data from the bottom face of the layer.
8. Specify the **Export File Path** and name of the exported file with the .json extension. Note that this field support macros like $PROJECTDIR; for example, $ProjectDir/Mag_B.json.
9. Specify the **Grid Size**, which determines the length of the user-specified grid in the X, Y directions, and the measurement units for the grid size.
10. Specify the **Threshold** of the field value, which is the minimum field value for the data to be exported. In the adjacent field, specify the units for the **Threshold**.
11. Specify the **Grid Coordinate Unit**, which determines how the coordinate values are listed in the data report.
12. Click **OK** to generate the report.

The figure that follows shows partial Mag_B results for all layers of the LC1_1 component:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

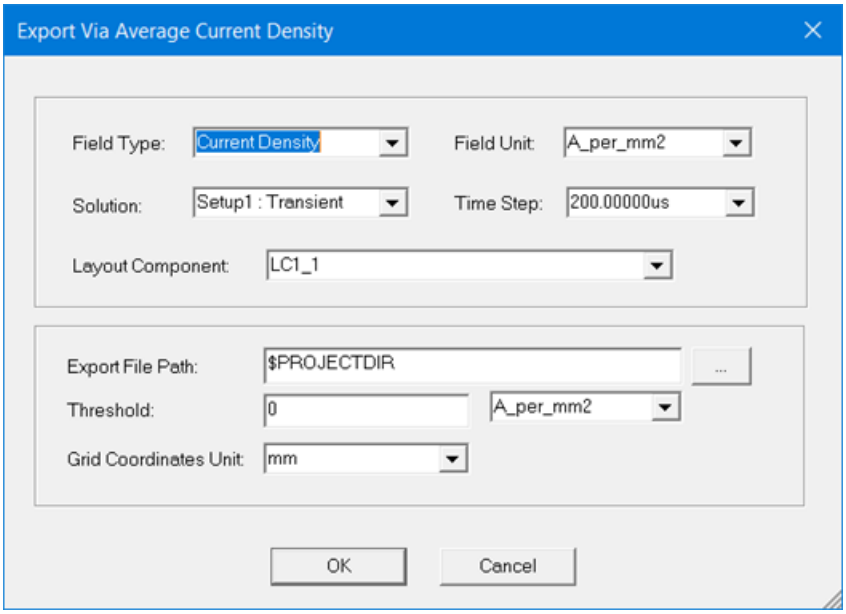
grid_data.json

```
{
    "time_step":"200us",
    "threshold":"0uTesla",
    "units":{
        "length":"mm",
        "magnetic_flux":"uTesla"
    },
    "magnetic_flux":[
        {
            "net":"<no-net>",
            "layer":"SURFACE",
            "left":"-102.35946",
            "top":"-105.23346",
            "right":"-97.35946",
            "bottom":"-110.23346",
            "magnetic_flux":"144.988053949992"
        },
        {
            "net":"<no-net>",
            "layer":"SURFACE",
            "left":"-102.35946",
            "top":"-100.23346",
            "right":"-97.35946",
            "bottom":"-105.23346",
            "magnetic_flux":"166.285586932325"
        },
```

Export Via Average Current Density

The **Maxwell 3D** > **Export Layout Field Data** > **Export Via Average Current Density** exports the average current density on the plane in the middle of two signal layers for all the vias in a layout component. If a via cuts through multiple signal layers, it outputs one value between every two neighboring layers.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After the window is set up and the user clicks **OK**, Maxwell exports the data to the specified file. The output data for each via includes the following:

- Start and end layer of the current is flowing from and to.
- X and Y coordinate for the center of the via.
- Pad stack name.
- Average current density on the plane in the middle of two signal layers.

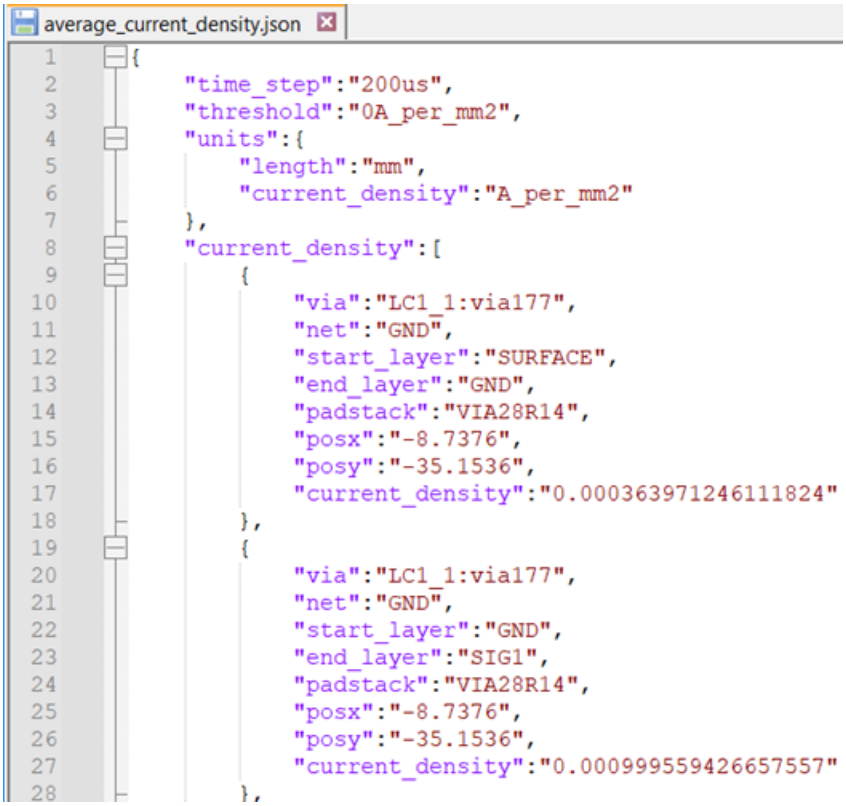
To set up the Export Via Average Current Density window, configure the following settings:

1. For this window, the **Field Type** has only the **Current Density** option.
2. Click the drop-down arrow beside **Field Unit**, and select the unit for exporting data. The base unit is **A_per_mm2**, but the user can select different metric units.
3. If there are multiple analysis setups in your design, you must select which setup solution you would like to use for the export. Click the drop-down arrow beside the **Solution** field, and select the desired solution results. If there is only one solution in the design, it will automatically display in the Solution field.
4. Select the **Time Step** value you would like to use for the extraction. The available values are determined by transient setup in the Solve Setup window.
5. From the **Layout Component** drop-down menu, select the layout component instance to export data from. If there is only on layout component in your design, it will be selected by default.
6. Specify the **Export File Path** and name of the exported file with the .json extension. Note that this field support macros like $PROJECTDIR; for example, $ProjectDir/current_density.json.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Specify the **Threshold** of the field value, which is the minimum field value for the data to be exported. In the adjacent field, specify the units for the **Threshold**.
8. Specify the **Grid Coordinate Unit**, which determines how the coordinate values are listed in the data report.
9. Click **OK** to generate the report.

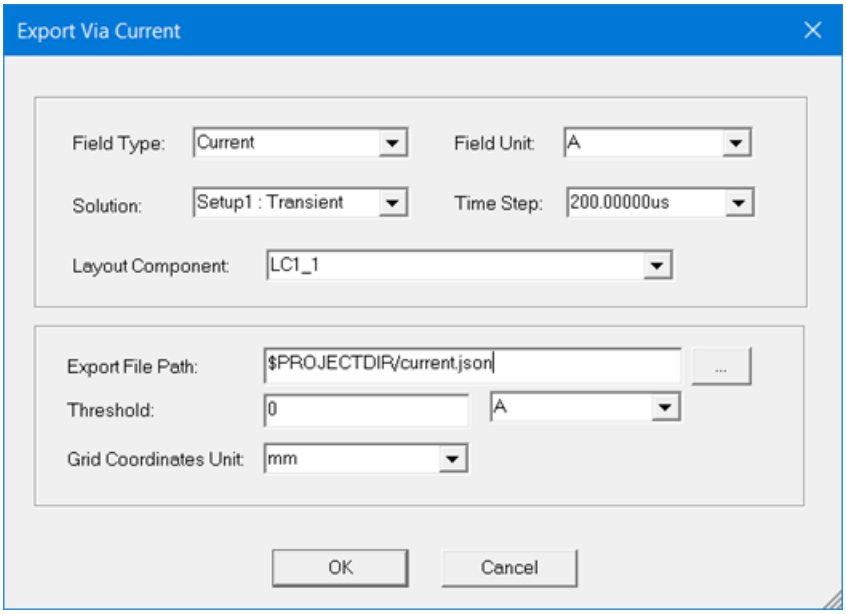
The figure that follows shows partial current density results for the LC1_1 component:



Export Via Current

The **Maxwell 3D** > **Export Layout Field Data** > **Export Via Current** function exports the current on the plane in the middle of two signal layers for all the vias in a layout component. If a via cuts through multiple signal layers, it outputs one value between every two neighboring layers.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After the window is set up and the user clicks **OK**, Maxwell exports the data to the specified file. The output data for each via includes the following:

- Start and end layer of the current is flowing from and to.
- X and Y coordinate for the center of the via.
- Pad stack name.
- Average current on the plane in the middle of two signal layers.

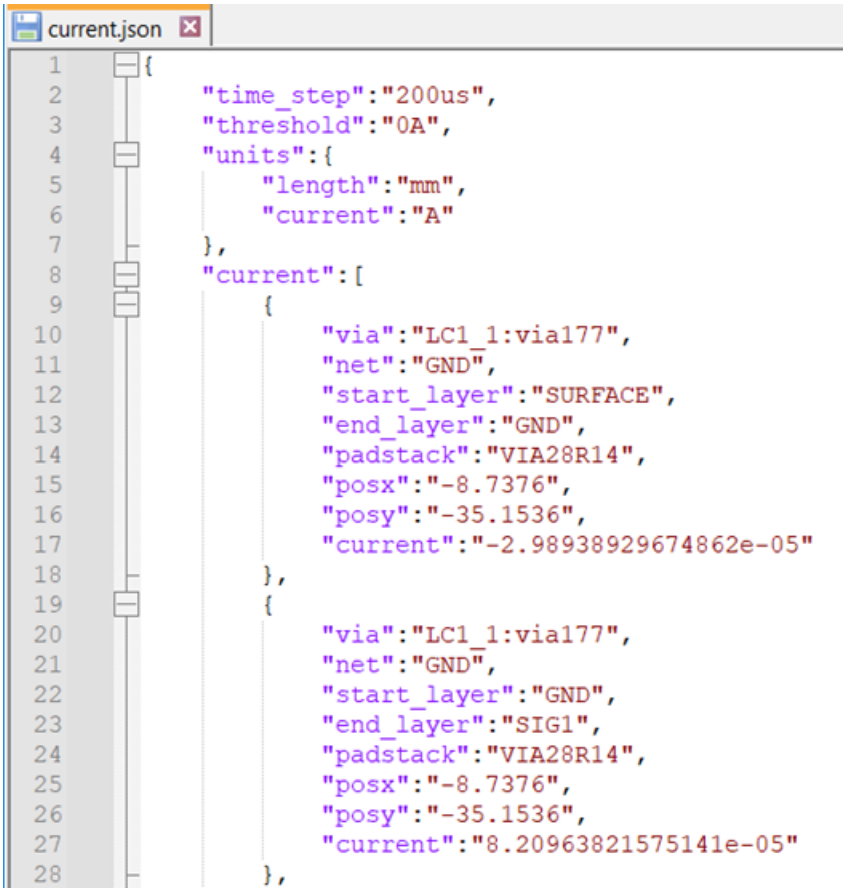
To set up the Export Via Current window, configure the following settings:

1. For this window, the **Field Type** has only the **Current** option.
2. Click the drop-down arrow beside **Field Unit**, and select the unit for exporting data. The base unit is **A**, but the user can select different metric units.
3. If there are multiple analysis setups in your design, you must select which setup solution you would like to use for the export. Click the drop-down arrow beside the **Solution** field, and select the desired solution results. If there is only one solution in the design, it will automatically display in the Solution field.
4. Select the **Time Step** value you would like to use for the extraction. The available values are determined by transient setup in the Solve Setup window.
5. From the **Layout Component** drop-down menu, select the layout component instance to export data from. If there is only on layout component in your design, it will be selected by default.
6. Specify the **Export File Path** and name of the exported file with the .json extension. Note that this field support macros like $PROJECTDIR; for example, $ProjectDir/current.json.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Specify the **Threshold** of the field value, which is the minimum field value for the data to be exported. In the adjacent field, specify the units for the **Threshold**.
8. Specify the **Grid Coordinate Unit**, which determines how the coordinate values are listed in the data report.
9. Click **OK** to generate the report.

The figure that follows shows partial current results for the LC1_1 component:



Export Layer-Induced Eddy Current

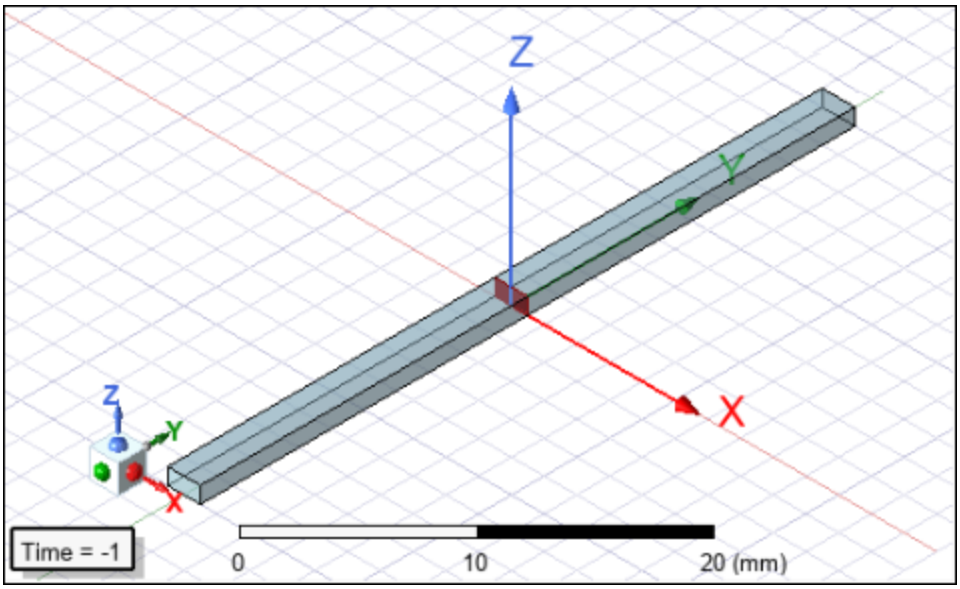
> **Important:** The Export Layer-Induced Eddy Current function is only accessible if it has been activated from **Tools** > **Options** > **General Options** > **Desktop Configuration** > **Beta Options** window. Select the **Maxwell Export ECAD Induced Eddy Current** feature.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important: This feature was introduced in Ansys Maxwell 25r1; you cannot export layer-induced eddy current from field data generated with version earlier than 25r1. To use this export function, rerun your analysis before exporting.

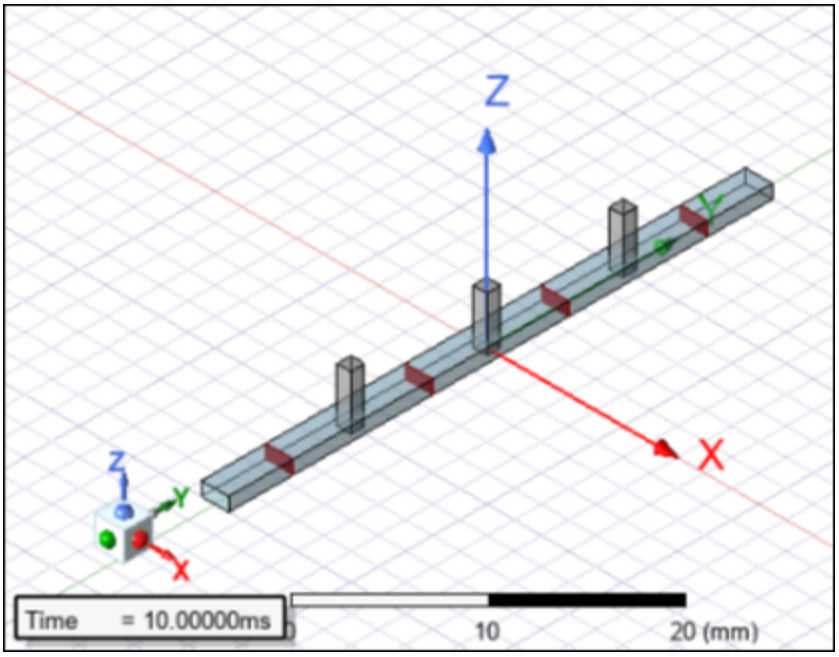
For a Maxwell 3D Transient A-Phi design with a layout component, you can calculate and export the induced current on each trace of the layout. The layout component of a chip design typically consists of vertical and horizontal traces, and the traces on different layers are connected by vias. How the calculation is performed depends on if the trace is connected to a via:

- If trace does not connect to any via, the current following through the trace is considered consistent, so the calculation is performed on the center cross-section of the trace object:



- If the trace is connected to vias, current will flow into or from the vias, but the sections between the vias will have a similar current value. Therefore, in this case, the current on the center of each section will be calculated:

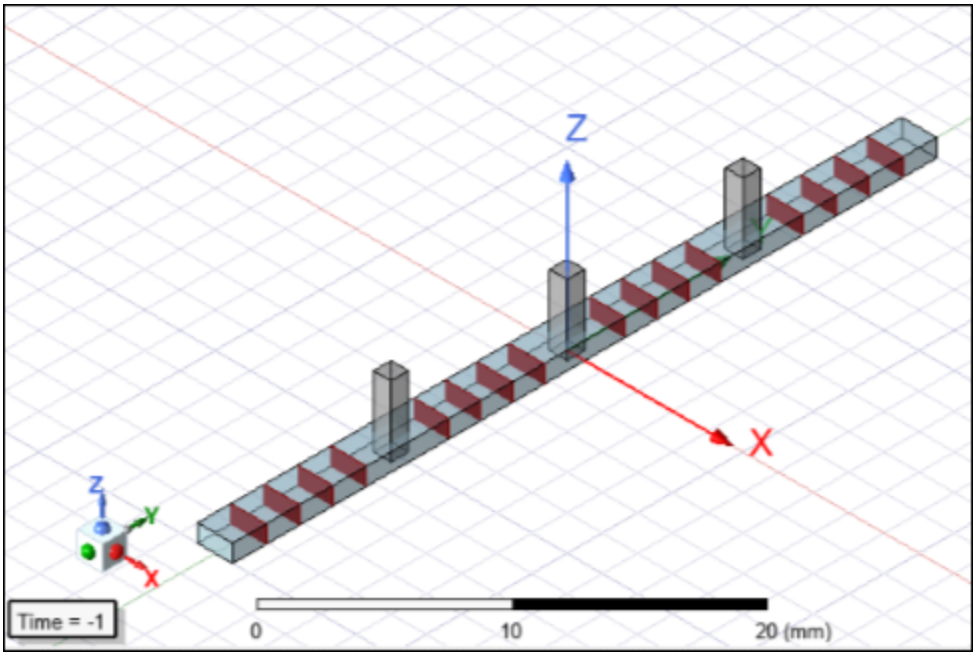
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Warning:** If a trace is touching vias or conductors from other layers, the current on the touching surface area does not flow in the x or y direction. In this case, there will be no current value calculated for this area.

- Induced eddy current may be different within different locations of the same section, so the **Export Layer Induced Eddy Current** function provides a maximum interval length option (see **Max Interval** below). By decreasing the distance between intervals, users can increase the number of samplings on each object to better capture the distribution of induced current.

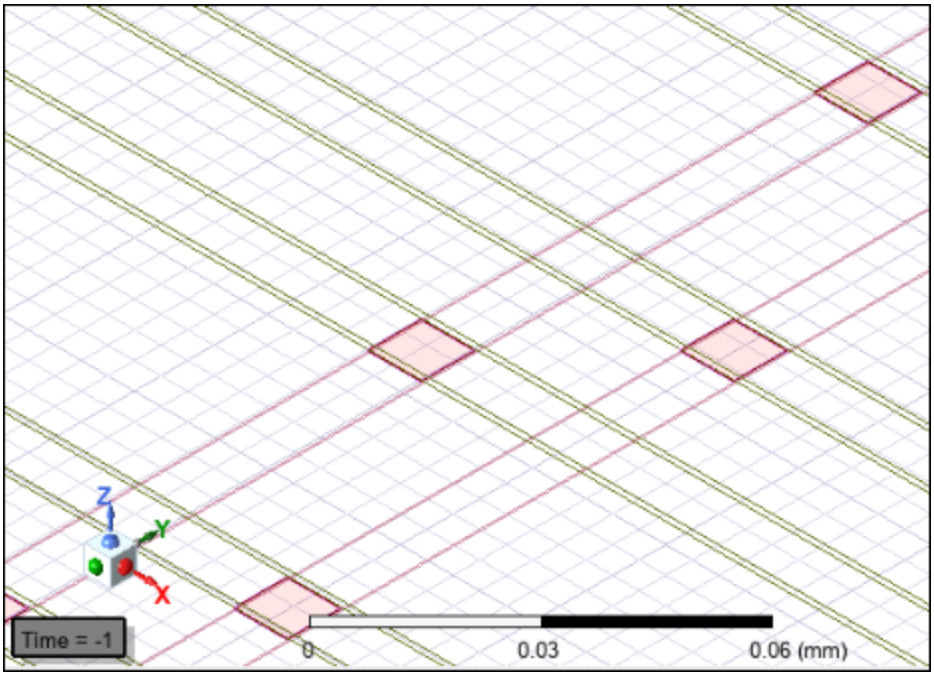
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:** Note that when a trace is touching vias or conductors from other layers, the current on the touching surface area does not flow in the x or y direction. In this case, there will be no current value calculated in this area.

- For some chip designs, the vias between two layers are exported as non-via objects in a layer between the two trace layers. In this case, the current value on the Z direction will be exported.

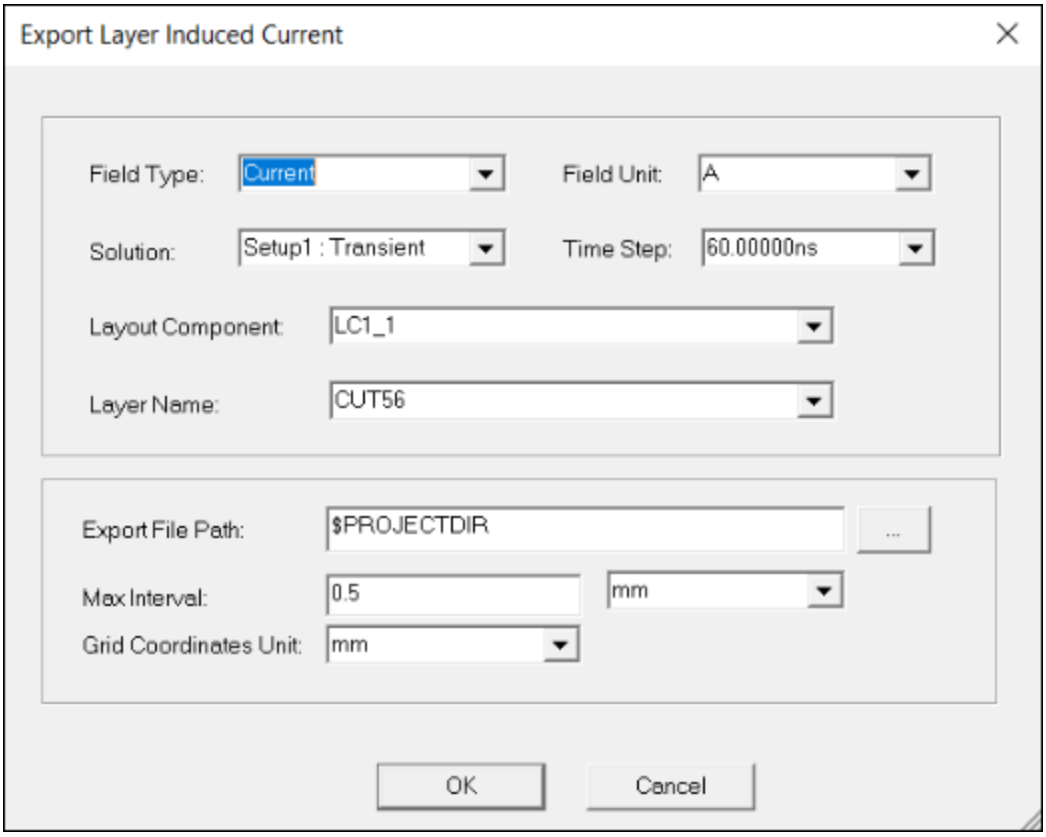
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To set up the **Export Layer Induced Current** function:

1. In the Program Manager tree, right click on the Transient A-Phi design that has been simulated, and select **Export Layout Field Data** > **Export Layer Induced Current**. As an alternative, select **Maxwell 3D** > **Export Layout Field Data** > **Export Layer Induced Current**. The following dialog box appears:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. For this window, the **Field Type** has only the **Current** option.
3. Specify the **Field Unit** for exporting data (mA by default).
4. Specify the **Time Step** to perform the calculation.
5. From the **Layout Component** drop-down menu, select the name of the layout component instance that will be used to export data. If there is only one layout component, it will be selected by default.
6. From the Layer Name drop-down menu, select the layer to perform the calculation on; use **All_layers** to select all the conducting layers; dielectric layers will be excluded.
7. For the **Max Interval** field, specify the maximum length of the cross-section intervals to be used to calculate the current (see illustration above).
8. Specify the **Export File Path**; the file will be save at this location. The path specification supports macros like $PROJECTDIR.
9. Specify the **Grid Coordinates Unit**, which determines how the coordinate values are listed in the data report (mm by default).
10. Click **OK**. Maxwell exports the data to the specified file.

The induced current is generated on each trace. Each data point contains the following data:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The version number of the export data format; this version will be named as 1.0.
- The X, Y, and Z coordinates of the center of the cross-section on the trace where the current is calculated (with respect to the global coordinate system of the layout design)
- The layer that the trace belongs to
- The net name of the trace
- The direction of the current, which can only be x or y
- The value induced current on the cross-section

Here is a sample output:

```
{
  "version":"1.0", // version number
  "time_step":"60ns", // solved time step
  "units":{ // definition for data unit
    "length":"mm",
    "current":"mA"
  },
  "current":[
    {
      "layer":"M6", // layer name
      "net":"VDD", // net name
      "direction":"x", // direction, can be x or y
      "posx":"1.87804255", // the x coordinate for the center of the
      cross section
      "posy":"0.7707", // the y coordinate for the center of the
      cross section
      "posz":"0.0047075", // the z coordinate for the center of the
      cross section
      "current":"0.0659217742139845"// the current value in x or y
      direction
    },
    {
      "layer":"M6",
      "net":"VSS",
      "direction":"x",
      "posx":"1.87804255",
      "posy":"0.8001",
      "posz":"0.0047075",
      "current":"0.0472994327556773"
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
    },
...
```

When using this function, note the following:

- The layout component should be placed without any rotation with respect to the global axis.
- The selected time step should be in the selected calculation time step in the solved setup and have valid field solution.
- The traces in the layout component should be straight lines in the X or Y direction. The calculation for the induced current does not support objects that do not align with the X or Y axis.

> **Tip:** The exported data can be used as input for Ansys RedHawk-SC, Ansys' tool for chip analysis; for more details, see https://www.ansys.com/products/semiconductors/ansys-redhawk-sc

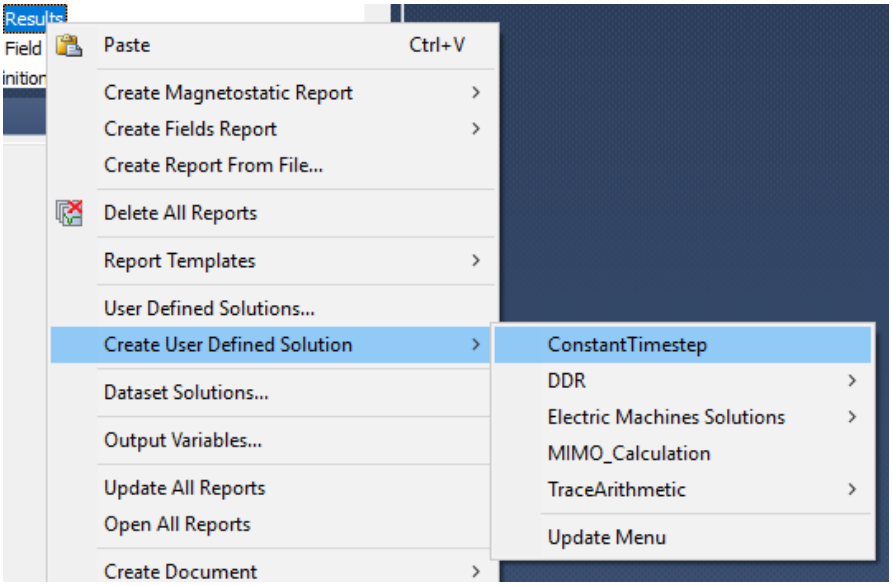
User-Defined Outputs (UDOs)

User-defined outputs (UDOs) allow users to define calculations through IronPython scripts or any .NET language (and used by the IronPython script). The UDO scripts need to be in the **UserDefinedOutputs** directory under either **syslib**, **userlib**, or **Personallib** with any directory structure needed for organization. (The **Lib** directory name is special and its purpose will be explained later on in the document.)

The UDO scripts that are placed in syslib/UserDefinedOutputs, userlib/UserDefinedOutputs, or Personallib/UserDefinedOutputs become available to the user to create "User Defined Solutions" through the **Results** > **Create User Defined Solution** menu.

Use **Results** > **Create User Defined Solution** > **Update Menu** to refresh the menu to include the new UDO scripts that might have been copied to syslib, userlib, or Personallib; or exclude them if they have been deleted, after the launch of desktop. Once the user-defined-solution is created, the solution and the calculations defined by UDO become available in Reporter as any other quantities in a new "User Defined" report type. (**Electric Machines Solutions** submenu is available only on Windows.)

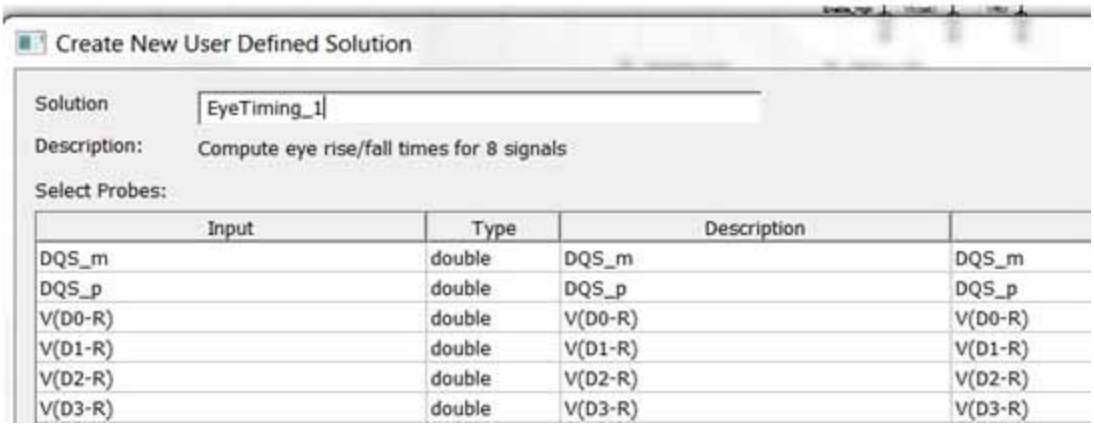
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Named Probes and Properties in User-Defined Outputs

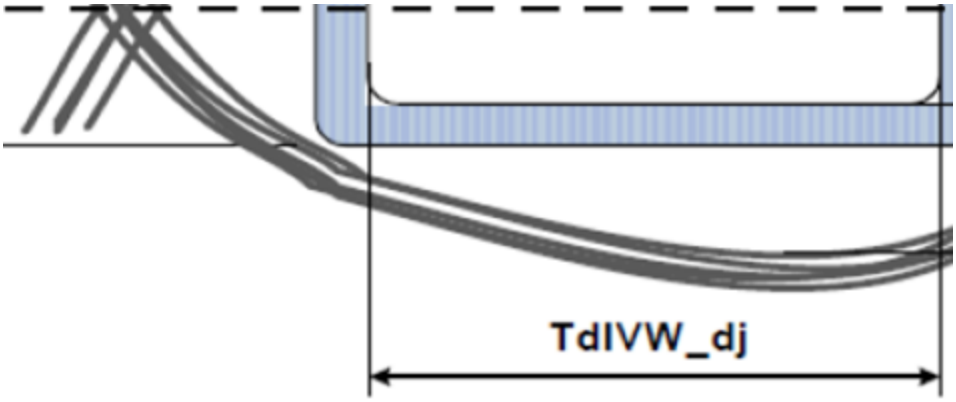
UDOs allow processing data across traces, solutions, and report types. A UDO specifies the named probes and properties for which user selects or enters the values at the time of creation of user-defined solution. Probes are very similar to traces except that the user selects the values of only intrinsic variables for probes. The values of design or project variables are selected when a trace is created based upon the user defined solution in reporter.

For example, you could create a user-defined solution called EyeTiming_1.

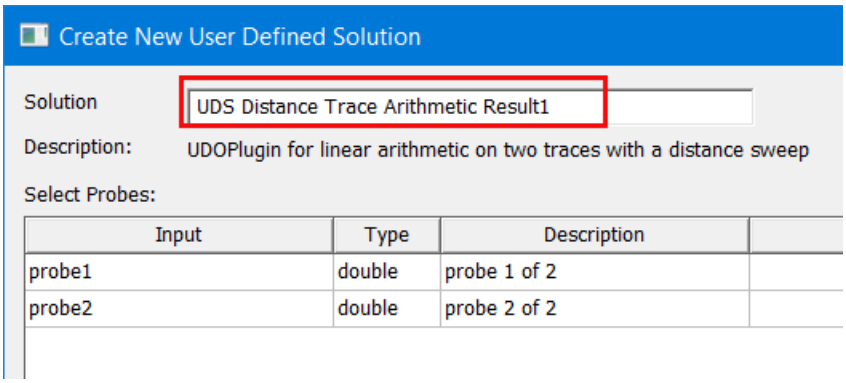


You can then access this solution in the Reporter.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

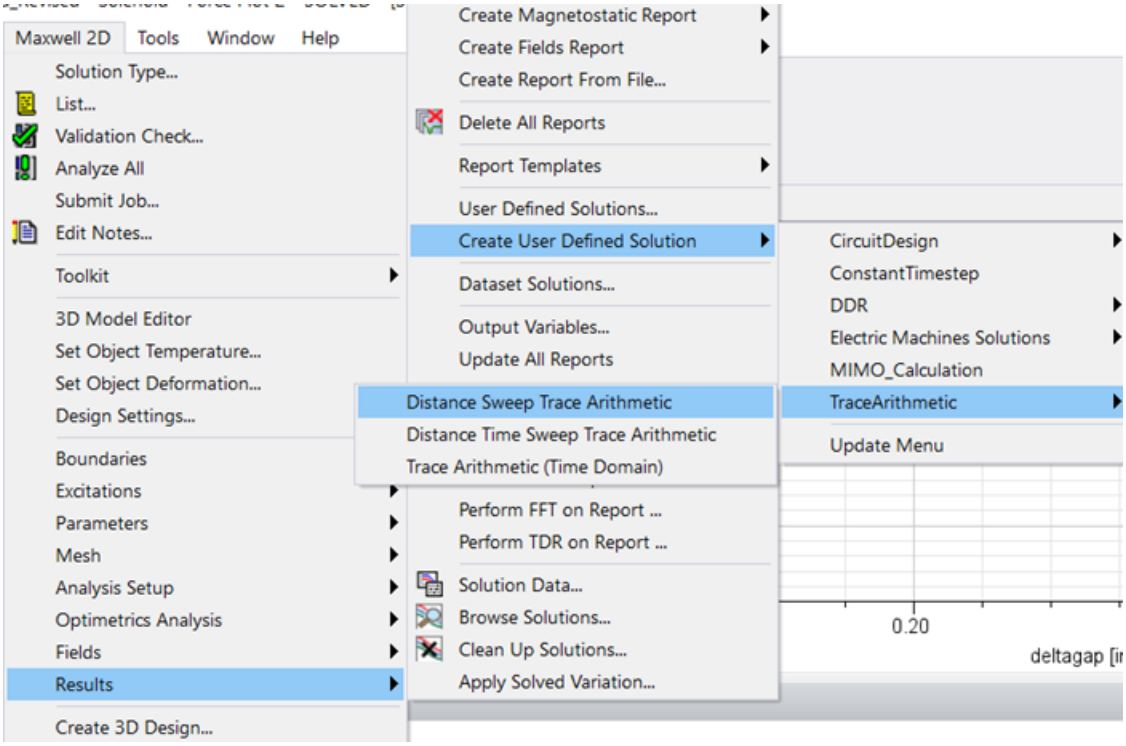


In another example, you could create a user-defined solution called **UDS Distance Trace Arithmetic Result1**:



Then access this solution in the **Create User Defined Solution** menu:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Computation of Traces Based UDO Calculations

When traces that are based upon UDO outputs are computed, the data for probes is computed and passed to the UDO script for each design variation. Along with the probe data, the values of properties entered by user are also passed. The information about the UDO calculations that need to be computed is also made available. The UDO then performs the computation and passes the results to reporter. Note that UDOs can compute and pass back more calculations than have been requested at that point of time. This allows UDOs to compute a set of calculations that take almost same amount of computational resources as any one calculation in that set and cache that with reporter. *When those calculations are subsequently plotted by user, reporter will use the cached results instead of invoking the computation on UDO*.

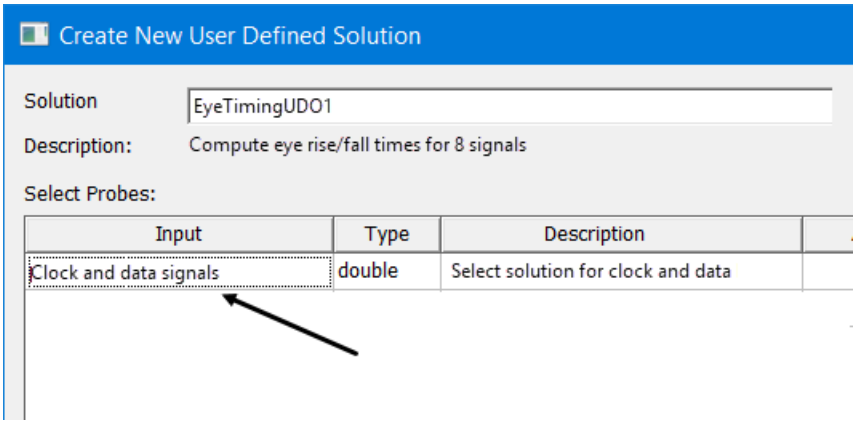
Dimensions Reduction by UDO Calculations

The probes in a UDO can have heterogeneous dimensions of data, for example, one probe in a UDO can have data that is a function of *n* intrinsic variable, while another probe in same UDO can have data that is function of *m* intrinsic variables, with *n* and *m* are potentially different. UDOs allow you to reduce any number of these intrinsic variables. In the above example, UDO calculations can be a function of any number of intrinsic variables including not being function of any intrinsic variable at all. UDO calculations can also be a function of an intrinsic variable that none of the probes is function of. *The only restriction is that Freq cannot be reduced if any of the probes are on a Fields report type.*

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dynamic Probes

In addition to named probes and properties, UDOs can specify named dynamic probes. The difference between probes and dynamic probes is that while the end user of UDO specifies the complete trace definition for probe, the expression for dynamic probe is specified by UDO code itself and not by end user. This allows UDOs to access the data for probes without requiring the end user to enter each individual probe. For example, a UDO can access data for a huge S matrix for 100 port design without having the end user enter the probe information for each of those 10,000 quantities. Each dynamic probe is associated with a named probe that is entered by user. Information about solution, context, and intrinsic variables is used from selected probes; however multiple dynamic probes can be associated with the same user selected probe. The dynamic probes are acquired from UDOs at the time of trace computation and not at the time of creation of user defined solution.



This means that you select solution, context, and values of intrinsic variables just once, and the same information is used (in this case) for all clock and data signals. The expression for those signals comes from the UDO code.

User-Defined Outputs: Python Script API

A User-Defined Output (UDO) extension is implemented as an IronPython script that defines a class with a specific name: **UDOExtension**, which derives from a specific base class **IUDOPluginExtension** and implements its abstract methods.

- UDO Extension Implementation
- Data Types Used in UDO Python Scripts
- Working With Properties for UDO
- Other Application Specific Classes Used in Python Scripts
- User-Defined Outputs: Messaging Methods
- Using .NET Collection Classes and Interfaces in Python Scripts

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UDO Extension Implementation

The purpose, argument list, and expected return types for each of the IUDOPluginExtension abstract methods, which the UDO author is expected to implement are described below.

- Import Statements
- UDOExtension Class
- IUDOPluginExtension Abstract Class

Import Statements

The base class to use and the types it uses in turn are contained in .NET assemblies. The use of these requires that the assemblies be imported into the UDO script. The following import statements should be added to the top of the Python script:

```
from Ansys.Ansoft.ModulePluginDotNet.Common.API import *
 from Ansys.Ansoft.ModulePluginDotNet.Common.API.Interfaces import *
 from Ansys.Ansoft.ModulePluginDotNet.UDO.API.Interfaces import *
 from Ansys.Ansoft.ModulePluginDotNet.UDO.API.Data import *
```

UDOExtension Class

The UDO itself should be implemented as an IronPython class called **UDOExtension** which *must* derive from the **IUDOPluginExtension** abstract base class (from the **Ansys.Ansoft.ModulePluginDotNet.UDO.API.Interfaces** namespace).

Note that power users could derive a class hierarchy tuned toward a specific type of UDOs and that they can derive from their own base classes. The only requirement is that directly or indirectly, the UDO class must derive from **IUDOPluginExtension**.

The UDOExtension abstract class declares the optional Validate abstract method that may be implemented in the UDOExtension class or one of its base classes.

Example:

```
def BaseClassUDO ((IUDOPluginExtension):
 #base class implementation
 …
 def UDOExtension ((BaseClassUDO):
 #UDO class implementation …
```

Validate

This optional method is used to validate the user choices. The values of the properties entered, the probes etc. can be checked for suitability. This function, while a part of the IUDOExtensionabstract class, has a meaningful default implementation and is therefore optional. However, it can be overridden to take advantage of advanced functionality.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UI Access	NA		
Parameters	Name	Type	Description
	<errorStringList>	List<string>	[out] C# list of Python strings. Should be set only if validation failed; ignored if validation is successful. One error string should be set per each validation error.
	<udsProbParams>	List<UDSProbeParams>	[in] C# list of UDSProbeParams objects.
	<propList>	IPropertyList object	[in] list of properties
	< userSelectionForDynamicProbes >	List<UDSProbeParams>	[in] C# list of UDSProbeParams objects.
Return Value	Boolean. True on validation success and false on failure. The default implementation always returns true.		

Python Syntax	Validate (<*errorList*>, <*probeList*>, <*propertiesList*>, <*dynamicProbes*>)

Python Example	`def Validate(self, errorStringList, probeList, propList, dynamicProbes):`
	`  if probeList == None or probeList.Count == 0:`
	`  errorStringList.Add("Empty probe list")`
	`  return False`
	`  return True`

IUDOPluginExtension Abstract Class

The implementation of the IUDOPluginExtension class will be described in this section using a simple UDO example that expects a single probe and reduces its dimension returning as its outputs, the max, min, and average of its input probe data. The script in its entirety will also be listed later on.

Required functions:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The IUDOPluginExtension abstract class declares the following abstract methods that must be implemented in the UDOExtension class or one of its base classes. Not implementing any of these methods will result in a run-time error and a non-functioning UDO. The UDS refers to User Defined Solution parameters.

- GetUDSName
- GetUDSDescription
- GetUDSSweepNames
- GetCategoryNames
- GetQuantityNames
- GetQuantityInfo
- GetInputUDSParams
- GetDynamicProbes
- Compute

GetUDSName

Return a string that is used as a prefix for all solution instances created using this UDO.

UI Access	NA
Parameters	None
Return Value	String

Python Syntax	GetUDSName

Python Example	`def GetUDSName(self):`
	`    return "MinMaxAvg"`
	`udsName = UDOPluginEntention.GetUDSName`

GetUDSDescription

Returns a description for the UDO, its purpose, etc. This is used in multiple UDO related dialogs in the application to describe the UDO.

UI Access	NA
Parameters	None
Return Value	String

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Python Syntax	GetUDSDescription
Python Example	`def GetUDSDescription(self):` `    return "Sample UDO for dimension reducing quantities`

GetUDSSweepNames

Returns a list of sweep names to be used for the solution generated by the UDO. These will appear in the sweeps list displayed in the standard reporter dialog when used to create reports from the solution generated by the UDO.

UI Access	NA
Parameters	None
Return Value	List of strings. If the UDO outputs have no sweeps, return the empty list [].

Python Syntax	GetUDSSweepNames

Python Example	`# Returns list of sweeps names`
	`# We have no sweeps as we reduce them.`
	`def GetUDSSweepNames(self):`
	`    return []`

GetCategoryNames

The outputs that the UDO solution provides or generates can be classified into multiple categories, like how the application is displayed in the report creation dialog box.

UI Access	The output will be listed in the categories box in the dialog when creating reports from the UDO generated solution data.
Parameters	None
Return Value	List of strings

Python	GetCategoryNames

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Syntax	
Python Example	`def GetCategoryNames(self):` `  return ["UDOOutputs"]`

GetQuantityNames

For each of the category names returned from the GetCategoryNames method, this function is called to return a list of quantities to be organized under that category name.

> **Note:**
>
> The quantity names must be unique across the categories; that is, no two categories can have quantities with the same name.

UI Access	NA		
Parameters	Name <*categoryName*>	Type String	Description [in] Name of category.
Return Value	List of strings		

Python Syntax	GetQuantityNames(<*categoryName*>)

Python Example	`# returns a list of quantity names for the supplied category name`
	`def GetQuantityNames(self, catName):`
	`  if catName == "UDOOutputs":`
	`  return ["min_val", "max_val", "avg_val"]`
	`  else:`
	`  return []`

GetQuantityInfo

For each quantity that the UDO creates, it must also describe the quantity (unit and other details). This method is called for each quantity name (across all categories) as returned from an earlier call of the GetQuantityNames method.

UI Access	NA		
Parameters	Name	Type	Description

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

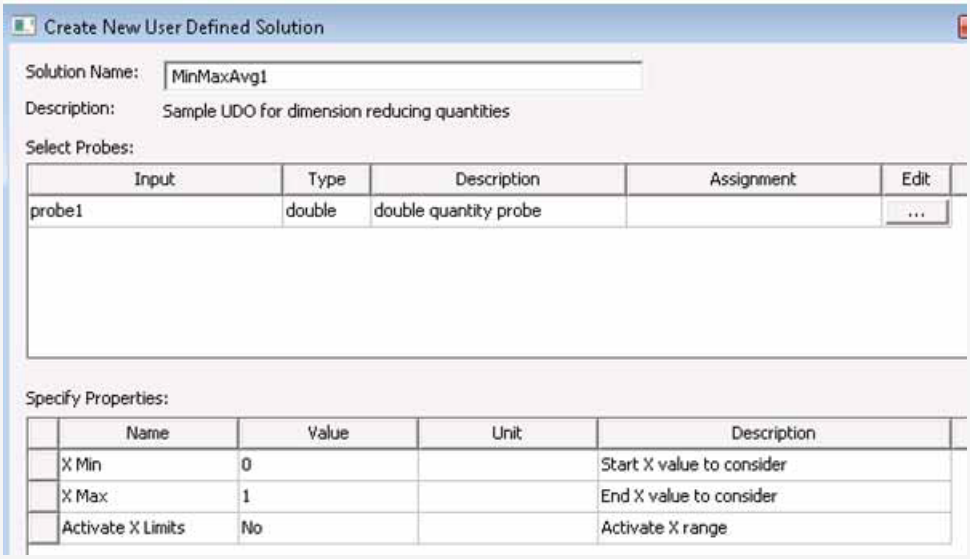
	<quantityName>	String	Name of quantity
Return Value	Object of type QuantityInfo.		

Python Syntax	GetQuantityInfo(*<quantityName>*)

Python Example	`# Returns an instance of QuantityInfo for the qtyName supplied or None if such a`
	`# quantity could not be found`
	`def GetQuantityInfo(self, qtyName):`
	`    # All the quantities we have are simple doubles`
	`    # we can leave them unitless`
	`    return QuantityInfo(Constants.kDoubleParamStr)`

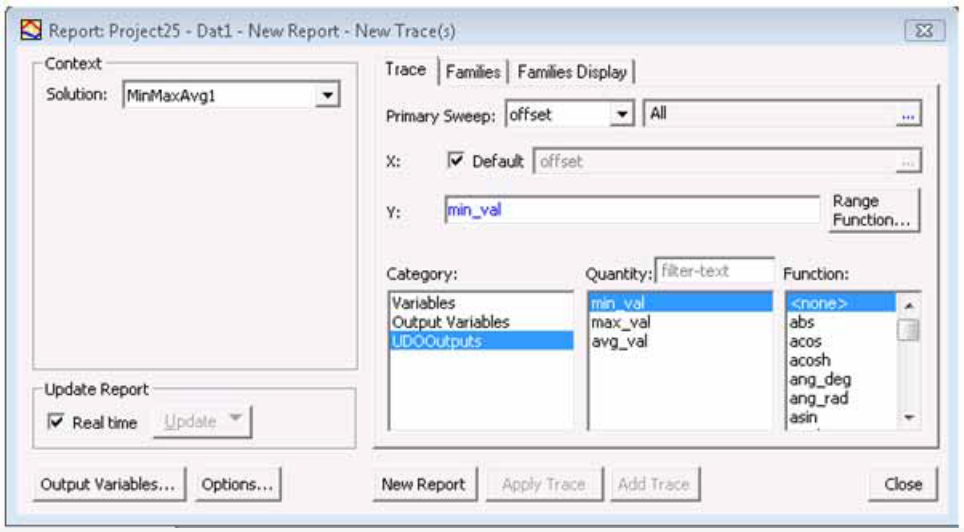
GetInputUDSParams

This is the main definition method of the UDO. The supplied arguments populate details of the parameters to which the UDO user will specify value, specify the probe names and their types as well as the dynamic probe selections.

UI Access	The **GetInputUDSParams** function results in the following dialog when you click **Results**>**Create User Defined Solution**. The mapping from the UDSParams and the properties to the GUI elements should be unambiguous. The name and description of the UDS are also displayed in the **Create New User Defined Solution** window. 

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When a report is created from the UDO dialog box, the category and quantity names specified by the UDO are used (as seen below).



Parameters

Name	Type	Description
<udsParams>	List< UDSProbeParams >	Parameters for the probe. The UDO script must add one instance of UDSProbeParams for each probe definition it will display. When creating the UDO solution, the UDO user must assign a matching quantity to each probe. When user define a UDS in the GUI, the user must select a design-solution-quantity for each UDSProbeParam .
<propList>	IPropertyList	The propList object is used to add properties that should be displayed to the user for data collection. These properties with their user-supplied values are returned to the UDO script by the Compute

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	<*userSelectionForDynamicProbes*>	List< UDSProbeParams >	method. When user defines a UDS in the GUI, the user ONLY needs to specify the Design-Solution. This quantity will be assigned dynamically by matching the component-expression defined in each UDSProbeParam.
Return Value	Boolean. If true, the event was handled successfully. If false, it was not.		

Python Syntax	GetInputUDSParams(<*udsParams*>, <*propList*>, <*userSelectionForDynamicProbes*>)

Python Example	`# Returns list of UDSParams and list of dynamic properties`
	`# Adds setup time properties to the propList`
	`def GetInputUDSParams(self, udsParams, propList, userSelectedDynamicProbes):`
	
	`  # Add the probes. We need only one double quantity`
	`  param1 = UDSProbeParams("probe1", "double quantity probe", Constants.kDoubleParamStr, "", "")`
	`  udsParams.Add(param1)`
	
	`  # Add the properties we want the user to supply`
	`  # In this case, we will ask for a start/end range for`
	`  # X parameters. Since we cannot reasonably provide defaults`
	`  # as we have no idea what the sweep limits will be, we will`
	`  # also ask if the limits are to be activated.`
	`  prop = propList.AddNumberProperty("X Min", "0")`
	`  prop.Description = "Start X value to consider"`
	
	`  prop = propList.AddNumberProperty("X Max", "1")`
	`  prop.Description = "End X value to consider"`
	
	`  # For menus, the first option is the default.`
	`  prop = propList.AddMenuProperty("Activate X Limits", ["No", "Yes"])`
	`  prop.Description = "Activate X range"`

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	`return True`

GetDynamicProbes

This is the primary mechanism by which the UDO script obtains the probe data (as double precision values) for its compute process.

UI Access	NA		
Parameters	Name <*dynamicProbes*>	Type List< UDSDynamicProbes >	Description [out] List of dynamic probes
Return Value	Boolean. If true, the method was successful. If false, it was not.		

Python Syntax	GetDynamicProbes(<*dynamicProbes*>)

Python Example	`# Returns list of UDSParams and list of dynamic properties`
	`# output UDSDynamicProbeCollection probes`
	`def GetDynamicProbes(self, probes):`
	`pass`

Compute

This is the main computation method which generates the data for the quantities that make up the UDO solution.

UI Access	The data is received from the UI using IUDSInputData. It is processed and the result data is sent to the UI using IUDSOutputData.		
Parameters	Name <*inData*> <*outData*> <*propList*> <*progressMonitor*>	Type IUDSInputData IUDSOutputData IPropertyList IProgressMonitor	Description Used to get the input probe data. Used to set the UDO solution quantity and sweep data. Used to get the user entered values for each of the properties defined during the GetInputUDSParams call. This can be used to set progress for long running calculations, check for

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	user initiated abort etc.
Return Value	Boolean. If true, the method was successful. If false, it was not.

Python Syntax	Compute (<*inData*>, <*outData*>, <*propList*>, <*progressMonitor*>)

Python Example	`# IUserDefinedSolutionHandle API implementation.`
	`# Calculates output values and sets them using IUDSInputData/IUDSOutputData API.`
	`def Compute(self, inData, outData, propList, progMon):`
	
	`  # Get the sweeps associated with the probe and validate`
	`  # use the probe name that we had defined earlier`
	`  sweeps = inData.GetSweepNamesForProbe("probe1")`
	`  if (sweeps == None or sweeps.Count > 1):`
	`  AddErrorMessage(self.GetName() + "Unexpected sweep count 0 or > 1 in Compute")`
	`  return False`
	
	`  # Get the data associated with our probe`
	`  probeData = inData.GetDoubleProbeData("probe1")`
	`  sweepData = inData.GetSweepsDataForProbe("probe1", sweeps[0])`
	
	`  # Get the user specified properties.`
	`  # Note that ideally, these "X Min" etc names should be`
	`  # written as constant members and referred to in both`
	`  # the GetInputUDSParams and in Compute to reduce the`
	`  # change of typos.`
	`  useXRangeProp = propList.GetMenuProperty("Activate X Limits").SelectedMenuChoice`
	`  xRangeStart = propList.GetNumberProperty("X Min").ValueSI`
	`  xRangeEnd = propList.GetNumberProperty("X Max").ValueSI`
	
	`  # At this stage, one can look at the`
	`  # RequestedQuantities and create a dictionary to later`
	`  # check against. However, I am simply computing all`
	`  # the quantities.`

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
minVal = 0
maxVal = 0
avgVal = 0

# Check if we need to perform range computation
if useXRangeProp == "Yes":
  seenAny = False
  avgSum = 0
  count = 0

# zip is used since we also need to pull in sweep data
# an index and the array notation could also have been
# used
for probeVal, sweepVal in zip(probeData, sweepData):
  if sweepVal < xRangeStart or sweepVal > xRangeEnd:
    pass

# Note that in a better written script, this code
# would be refactored into its own function to
# avoid code duplication
if not seenAny:
  minVal = probeVal
  maxVal = probeVal
  avgSum = probeVal
  seenAny = True
  count = 1
else:
  if probeVal < minVal:
    minVal = probeVal

  if probeVal > maxVal:
    maxVal = probeVal

  avgSum += probeVal
  count += 1

if seenAny:
  avgVal = avgSum/count

else:
  seenAny = False
  avgSum = 0
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Data Types Used in UDO Python Scripts

There are several types that you must use while authoring a Python script. Some of them are used to pass data from the UI to a Python script and to provide interface for working with this data. Some are used to pass data from a Python script to the UI.

To pass data from a Python script to the UI, the objects must be created in the Python script. Then they can be set as a function's return values or set to the output parameters using their API.

Constants

- **kTraceTypeStr** : string constant used to specify an input of trace type
- **kSolutionTypeStr** : string constant used to specify an input of solution type
- **kNumberTypeStr** : string constant used to specify an input of number type
- **kTextTypeStr** : string constant used to specify an input of text type
- **kBoolTypeStr** : string constant used to specify an input of boolean type
- **kStandardReportStr** : string constant to specify a standard report
- **kEyeDiagramReportStr** : string constant to specify an eye diagram report
- **kUserDefinedReportStr** : string constant to specify a user defined report
- **kSweepDomainStr** : string constant to specify the sweep domain
- **kTimeDomainStr** : string constant to specify the time domain

Abstract Classes

- IProgressMonitor
- IUDSInput
- IUDSOutput

IUDSInputData

The purpose of this class is to get data (probe and sweep) from Desktop. The IUDSInputData abstract class declares the following abstract methods:

- GetDoubleProbeData
- GetSweepsDataForProbe
- GetComplexProbeData
- GetSweepNamesForProbe
- GetRequiredQuantities

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- GetVariableValues
- GetInterpolationOrdersData

Examples in this section are just to show proper syntax of the function calls. For actual usage of the class, see the Compute function's example.

GetDoubleProbeData

This is the primary mechanism by which the UDO script obtains the probe data (as double precision values) for its compute process.

UI Access	NA		
Parameters	Name <*probeName*>	Type String	Description Name of the probe for which data is requested. This is one of the many probes supplied during a call to the UDO's GetInputUDSParams method. Each probeName must be unique within an UDO.
Return Value	Double array of data for the specified probe if the probe exists or null if the probe is unknown.		

Python Syntax	GetDoubleProbeData (<*probeName*>)
Python Example	`# doubleData is a list of floats` `doubleData = inData.GetDoubleProbeData("probe1")`

GetSweepsDataForProbe

All probe data that is supplied is associated with one or more sweep (an intrinsic quantity like Time, Frequency, Theta, Phi, etc. that is swept) quantities.

UI Access	NA		
Parameters	Name <*probeName*> <*sweepName*>	Type String String	Description Name of the probe for which data is requested. This is one of the many probes supplied during a call to the UDO's GetInputUDSParams method. Each probeName must be unique within an UDO. Name of the sweep.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Return Value	Double array of data for the specified probe and sweep.

Python Syntax	GetSweepsDataForProbe(<*probeName*>, <*sweepName*>)
Python Example	`# sweepData is C# Array of doubles (floats in python)` `sweepData = inData.GetSweepsDataForProbe ("FarFieldsProbe","Freq"])`

GetComplexProbeData

The primary mechanism by which the UDO retrieves data for its input probes (if it expects complex data for the probe).

UI Access	NA		
Parameters	Name <*probeName*>	Type String	Description Name of the probe for which data is requested. This is one of the many probes supplied during a call to the UDO's GetInputUDSParams method. Each probeName must be unique within an UDO.
Return Value	Double array (float in Python) of data for the specified probe. Each pair of floats represents one complex number. The first value is for real part and the second value it the imaginary part. For instance, array [10.0, 0, 5.1, 2.1] represents 2 complex numbers: (10.0, 0) and (5.1, 2.1).		

Python Syntax	GetComplexProbeData(<*probeName*>)

Python Example	`# complexDataAsDouble is C# Array of doubles (floats in Python)`
	`# each pair of floats represents one complex number`
	`complexDataAsDouble = inData.GetComplexProbeData ("FarFieldsProbe")`
	`# creating a list of complex numbers from complexDataAsDouble array`
	`complexData = []`
	`if complexDataAsDouble != None:`
	`  for i in xrange(0,complexDataAsDouble.Count, 2):`

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	`complexData.append(complex(complexDataAsDouble [i],complexDataAsDouble[i+1]))`

GetSweepNamesForProbe

Retrieves a list of sweep quantity names associated with a given probe. This also indicates the dimensionality of the data. One name implies that the probe-data is 2D (probe-quantity vs Sweep Quantity) and two names implies 3D data (probe-quantity vs Sweep 1 X Sweep 2).

UI Access	NA		
Parameters	Name <*probeName*>	Type String	Description Name of the probe for which data is requested. This is one of the many probes supplied during a call to the UDO's GetInputUDSParams method. Each probeName must be unique within an UDO.
Return Value	IList<string> - list of sweep names for the current probe name.		

Python Syntax	GetSweepNamesForProbe(<*probeName*>)
Python Example	`# sweepNames is C# Array of strings` `sweepNames = inData.GetSweepNamesForProbe("FarFieldsProbe")`

GetRequiredQuantities

A given UDO can specify that it provides one or more computed quantities. The user might choose to create a report from only a few among the various available UDO outputs. This function returns the list of UDO output quantities that the user requested. Only these need be computed in the UDO's Compute method.

UI Access	NA
Parameters	None
Return Value	IList<string>

Python Syntax	GetRequiredQuantities()

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Python Example	`# quantities is C# Array of strings` `quantities = inData.GetRequiredQuantities()`

GetVariableValues

This allows the UDO to obtain the names and values of all the design variables for which the UDO quantities are being requested.

UI Access	NA
Parameters	None
Return Value	IDictionary<string,string> of key-value pairs for variables. Both key and value are strings.

Python Syntax	GetVariableValues()

Python Example	`# theDict is C# Dictionary<string, string>`
	`theDict = inData.GetVariableValues()`
	`if theDict != None:`
	`  #varPair is of .Net KeyValuePair type`
	`  for varPair in theDict:`
	`  varName = varPair.Key #string`
	`  varValue = varPair.Value #string`

GetInterpolationOrdersData

Returns the interpolation orders that are associated with the probe-data. The probe data is specified at each value of the various sweeps. Any value in between the sweep data points, can use the interpolation data to get a possibly more accurate (compared to linear interpolation) inter-sweep value.

UI Access	NA		
Parameters	Name *<probeName>*	Type String	Description Name of the probe for which data is requested. This is one of the many probes supplied during a call to the UDO's GetInputUDSParams method. Each probeName must be unique within an UDO.
Return Value	Byte array of interpolation order for the specified probe. These are to be		

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	treated as 8-bit signed integers, that is, their values range from 0-127.

Python Syntax	GetInterpolationOrdersData(<*probeName*>)

Python Example	`# interData is C# Array of bytes (integers in Python)`
	`interData = inData.GetInterpolationOrdersData(kProbeNames[0])`
	`for interValue in theDict:`
	`  # interValue and order are integers`
	`  order = interValue`

IUDSOutputData

This type is a twin of the IUDSInputData in that it is used to store the values computed by the UDO's Compute method. The IUDSOutputData abstract class declares the following abstract methods:

- SetSweepsData
- SetDoubleQuantityData
- SetComplexQuantityData

Examples in this section are just to show proper syntax is function calls. For actual usage of the class, see the Compute function example.

SetSweepsData

Each quantity that is computed by the UDO can be associated with a sweep. If it is, the values that make up the sweep's data points must be specified using this call.

UI Access	NA		
Parameters	Name <*sweepName* > <*sweepData*>	Type String List<floats>	Description Name of the sweep Sweep data for the specified sweep.
Return Value	Boolean. If true, the event was handled successfully. If false, it was not.		

Python Syntax	SetSweepsData (<*sweepName*>,<*sweepData*>)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Python Example	`sweepList = [12.3, 14.5, 16.7]` `outData.SetSweepsData("Freq", sweepList)`

SetDoubleQuantityData

This method is used to record the computed quantity data for each output that is computed. Please note that unless all the sweeps are reduced, this should be used in conjunction with SetSweepsData.

UI Access	NA		
Parameters	Name <*qtyName*> <*qtyData*>	Type String List<Floats>	Description Name of the quantity. Quantity data for the specified quantity.
Return Value	Boolean. If true, the event was handled successfully. If false, it was not.		

Python Syntax	SetDoubleQuantityData (<*qtyName*>,<*qtyData*>)
Python Example	`doubleList = [12.3, 14.5, 16.7]` `outData.SetDoubleQuantityData("V1PlusV2", doubleList)`

SetComplexQuantityData

If the quantity computed is a complex quantity, use this method to set the quantity values. Any sweep values must be set separately via the SetSweepsData method.

UI Access	NA		
Parameters	Name <*qtyName* > <*qtyData* >	Type String List<floats>	Description Name of the quantity. Quantity data for the specified quantity. Complex numbers are passed as pairs of floats in a list.
Return Value	Boolean. If true, the method was successful. If false, it was not.		

Python Syntax	SetComplexQuantityData (<*qtyName*>,<*qtyData*>)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Python Example	doubleFromComplexList=[]
	complexList = [(1+1j), (2+4j), (9.1+3.2j)]
	for aComplex in complexList:
	    doubleFromComplexList.append(aComplex.imag)
	    doubleFromComplexList.append(aComplex.real)
	    outData. SetComplexQuantityData ("V1PlusV2", doubleFromComplexList)

Working With Properties for UDO

A property is the unit for collecting and using input from the user that is used to influence the UDO's Compute. These are initially set up when the UDOs **GetInputUDSParams** method is called and are retrieved in the UDO's Compute method.

There are three supported property types that could be used in the UDO script:

- INumberProperty to specify number properties (with unit support).
- IMenuProperty to allow the user to select from a list of options.
- ITextProperty to allow the user to enter text.

The IPropertyList type implements a collection for these properties:

- IPropertyList Abstract class
- IProperty Abstract class
- INumberProperty Abstract class
- ITextProperty Abstract class
- IMenuProperty Abstract class

IPropertyList Abstract Class

Attributes:

- AllProperties (IEnumerable<IProperty> - see IProperty)
- NumProperties (int)

Functions:

- GetProperty(string propName): Returns a named property as an IProperty.
- GetMenuProperty (string propName): Returns the named property as an IMenuProperty.
- GetTextProperty (string propName): Returns the named property as an ITextProperty.
- GetNumberProperty (string propName): Returns the named property as an INumberProperty.
- DeleteProperty (string propName): Deletes an already added named property.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- AddNumberProperty(string name, string numberWithUnits): Adds a new number property. If a property with the same name already exists, it is overwritten.
- AddTextProperty(string name, string textValue): Adds a new named text property with the supplied value. Any existing property with the same name is overwritten.
- AddMenuProperty(string name, IList<string> menuChoices): Creates a new named menu property with the supplied list of choices. The default selection is set to item 0 (the first item). Any property with the same name is overwritten.

IProperty Abstract Class

Attributes:

- Name (string)
- Description (string)
- PropType (read-only EPropType - see Constants)

Constructor:

- IProperty(string name, EPropType type)

The class is used as base class for INumberProperty, IMenuProperty, and ITextProperty.

INumberProperty Abstract Class

Base class:

- abstract class IProperty

Attributes:

- ValueSI (read-only double)
- ValueInUnits (read-only double)
- Units (read-only string)
- HasUnits (read-only bool)

Constructor:

- INumberProperty(string name)

Functions:

- Set(string numberWithUnits)
- SetDouble(double number, string unitString)

ITextProperty Abstract Class

Base class:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- abstract class IProperty

Attributes:

- Text (string)

Constructor:

- ITextProperty(string name)

IMenuProperty Abstract Class

Base class:

- abstract class IProperty

Attributes:

- MenuSelection (int): This represents the index into the MenuChoices list.
- SelectedMenuChoice (string): This is the item in the MenuChoices list corresponding to the MenuSelection index.
- MenuChoices (IList<string>)

Constructor:

- IMenuProperty (string name)

Example:

```
# adding data to IPropertyList propList; used in Compute function
prop = propList.AddNumberProperty('Offset 1', '0')
prop.Description = 'Trace 1 Offset'
prop = propList.AddNumberProperty("TRATE", "800 MHz")
prop.Description = "Frequency"
prop = propList.AddTextProperty("Text", "The Text")
prop.Description = "Text Property"
prop = propList.AddMenuProperty('Operation', ['Add', 'Subtract',
'Max' ,'Min','Mean'])
prop.Description = 'Operation menu'

# reading data from IPropertyList propList; used in Validate function
numOfNumberProperties = 0
if propList != None and propList.AllProperties != None:
        for prop in propList.AllProperties:
                if prop.PropType == Constants.EPropType.PT_NUMBER:
                        numOfNumberProperties ++
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Other Application-Specific Classes Used in Python Scripts

This section describes other classes used in Python scripts:

Constants Class

UDSProbeParams Class

UDSDynamicProbes Class

QuantityInfo Class

IProgressMonitor Abstract Class

Constants Class

The constants used in a Python script are defined in the Constants class.

Attributes:

- kDoubleParamStr : string constant used to specify *double* as the type of a quantity
- kComplexParamStr: string constant used to specify *complex* as the type of a quantity
- Enum EPropType: (used to set property type)
  - EPropType.PT_NUMBER
  - EPropType.PT_TEXT
  - EPropType.PT_MENU

Example:

```
paramType = Constants.kDoubleParamStr
propType = Constants.EPropType.PT_NUMBER
```

UDSProbeParams Class

This class defines which data quantity that a UDO will be pulled from the design to compute the UDO output (results). The objects of this class must be created in a Python script with the GetInputUDSParams function. They are supplied to the Validate function if implemented.

Attributes:

- ProbeName (read-only string)
- ProbeDescription (read-only string)
- ParamType (read-only string)
- ReportTypeName (read-only string)
- ComponentExpression (read-only string)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Constructor: UDSProbeParams(string probeName, string probeDescription, string paramType, string reportTypeName, string componentExpression);

- probeName - required.
- probeDescription - optional (can be empty string).
- paramType - required; can be one of the Constants:
  - kDoubleParamStr
  - kComplexParamStr
- reportTypeName - optional (can be empty string)
- ComponentExpression - optional (can be empty string)

Example:

```
udsProbParam = UDSProbeParams("probe1","", Constants.kDoubleParamStr,
"", "",)
```

UDSDynamicProbes Class

Attributes:

- UDSParam (read-only UDSProbeParams)
- UserSelectedProbeName (read-only string)

Constructor: UDSDynamicProbes (UDSProbeParams udsParam, string userSelectedProbeName)

- udsParam - required
- userSelectedProbeName - required

Example:

```
udsProbParam = UDSProbeParams("probe1","", Constants.kDoubleParamStr,
"", "",)
selectedName = "probe1"
udsDynamicProbParam = UDSDynamicProbes(udsProbParam, selectedName )
```

QuantityInfo Class

Attributes:

- ParamType (read-only string)
- FullUnitType (read-only string)

Constructors:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- QuantityInfo(string paramType)
- QuantityInfo(string paramType, string fullUnitType)

Parameters:

- paramType can be one of the Constants:
  - kDoubleParamStr
  - kComplexParamStr
- fullUnitType is a case insensitive string representing full unit type. It is not defined in Constants. Instead you can use any of the units in string representation - for example, "mm" or"ghz".

Example:

```
quantityInfo1 = QuantityInfo(Constants.kDoubleParamStr)
quantityInfo2 = QuantityInfo(Constants.kDoubleParamStr,"ghz")
```

IProgressMonitor Abstract Class

The object of this class is a progress monitor. It displays a task's calculated progress in the UI and checks if the user has requested to abort the computation. Use this object for computations that take a long time to complete or if the UI might freeze during the computation. When displayed in the application, each progress message has four items:

- A task name
- A sub-task name
- The progress amount
- A button to abort the task in progress

This class provides the following functionality and abort interaction:

- SetTaskName (string taskName)
- SetSubTaskName (string subTaskName)
- BeginTask (string name)
- SetTaskProgressPercentage(int progressPercent)
- CheckForAbort()
- EndTask (bool passFail)

Example:

```
progMon.BeginTask("Process DQS")
progMon.SetSubTaskName("Compute UI segments")
progMon.SetTaskProgressPercentage(33)
progMon.SetSubTaskName("Compute the rest")
progMon.SetTaskProgressPercentage(100)
progMon.EndTask(True)
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> There should be the same number of calls to BeginTask as to EndTask.

SetTaskName

Sets the name of the task whose progress is being monitored.

UI Access	NA		
Parameters	Name <*TaskName*>	Type String	Description Name of the task
Return Value	None		

Python Syntax	SetTaskName (<*TaskName*>)
Python Example	`progMon.SetTaskName("Compute UI")`

SetSubTaskName

Sets the name of the subtask whose progress is being monitored.

UI Access	NA		
Parameters	Name <*TaskName*>	Type String	Description Name of the task
Return Value	None		

Python Syntax	SetSubTaskName (<*TaskName*>)
Python Example	`progMon.SetSubTaskName("Compute UI segments")`

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BeginTask

Sets the name of the task whose progress will be monitored. Call this method only once for each task and subtask.

UI Access	NA		
Parameters	Name <*Name*>	Type String	Description Name of the task
Return Value	None		

Python Syntax	BeginTask (<*Name*>)
Python Example	`progMon.BeginTask("Process DQS")`

SetTaskProgressPercentage

Sets the progress percentage of a task to a specific value.

UI Access	NA		
Parameters	Name <Percent>	Type Integer	Description Percent of progress
Return Value	None		

Python Syntax	SetTaskProgressPercentage (<*Percent*>)
Python Example	`progMon.SetTaskProgressPercentage(33)`

CheckForAbort

Checks if a user has aborted the task. Call this method if the quantities are computationally expensive. This method can be called multiple times.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UI Access	Abort button on the progress dialog.
Parameters	None
Return Value	Boolean flag that if true indicated that the user requested an abort and you should call EndTask. If false, allow the task to continue.

Python Syntax	CheckForAbort
Python Example	`should_abort = progMon.CheckForAbort`

EndTask

Stops a task from running. Call this method only once for each task and subtask. In a script, there should be the same number of BeginTask calls as **EndTask** calls.

UI Access	NA		
Parameters	Name <PassFail>	Type Boolean	Description Flag that indicates whether to continue the main task or not. If true, the task will continue to process the next subtask. If false, the main task stops and an error icon appears in the UI.
Return Value	None		

Python Syntax	EndTask (<*PassFail*>)
Python Example	`progMon.EndTask(True)`

Using .NET Collection Classes and Interfaces in Python Scripts

Some of the API functions specified above use .Net collection classes and interfaces, that is, Array class, IList interface, IEnumerable interface, and IDictionary interface. The following section describes how to work with the .Net collection objects in Python scripts.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

.NET Array, IEnumerable, and IList objects can be indexed and iterated over as if they were Python lists. You can also check for membership using 'in'. To get .Net Array and IList sizes you can use Python's 'len' or .Net 'Count'.

Example:

Getting size:

```
arraySize = doubleDataArray.Count
   arraySize = len(doubleDataArray)
listSize = sweepsNamesList.Count
 listSize = len(sweepsNamesList)
```

Iterating:

```
for sweep in sweepsNamesList:
   print sweep
for in in xrange(listSize)
      print sweepsNamesList[i]
```

Checking for membership:

```
if 'Time' in sweepsNamesList:
   doThis()
  else:
   doThat()
```

For .NET IDictionary, the same as for Array and IList, you can get size with 'len' or 'Count' and check for membership of the keys using 'in'. Getting values for the keys also works the same way as in Python 'dict'.

Example:

Getting size:

```
varValuesSize = varValues.Count
 varValuesSize = len(varValues)
```

Checking for membership:

```
if 'offset' in varValues:
   print varValues['offset']
```

Getting value:

```
if 'offset' in varValues:
   offsetValue = varValues['offset']
```

As for iteration .NET Dictionary is different from Python dict. While iterating, python dict will return keys, .Net Dictionary will return .Net KeyValuePair.

Example:

Iterating:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for .NET IDictionary:

```
for varPair in varValues: #varPair is of .Net KeyValuePair type
     varName = varPair.Key
    varValue = varPair.Value
```

for Python dict:

```
for varName in varValues:
   varValue = varValues[varName]
```

You can use Python types instead of .NET types if you prefer. For this, you need to cast .NET array and .Net iList to Python list type and .NET Dictionary to Python dict type.

Casting should not be used for data arrays - it can be extremely costly for the memory usage as well as time consuming.

Example:

```
aPythonList = list(dotNetArray)
  aPythonList = list(dotNetList)
   aPythonDict = dict(dotNetDictionary)
```

User-Defined Outputs: Messaging Methods

Messaging methods are provided to convey additional information to the user from any of the UDOs methods. The Compute function is the one typically location where such use is anticipated. Any message sent via these functions are displayed in the application's message window using the appropriate icon.

These functions can also be used for debugging purposes.

- AddErrorMessage
- AddInfoMessage
- AddWarningMessage

Example script:

```
############################################################
# Imports
############################################################
from Ansys.Ansoft.ModulePluginDotNet.Common.API import *
from Ansys.Ansoft.ModulePluginDotNet.Common.API.Interfaces import *
from Ansys.Ansoft.ModulePluginDotNet.UDO.API.Interfaces import *
from Ansys.Ansoft.ModulePluginDotNet.UDO.API.Data import *
class UDOExtension(IUDOPluginExtension):
  def __init__(self):
    pass

  #--- IDA IUDOPluginExtension -----------------------
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
def GetUDSName(self):
  return "MinMaxAvg"

#--- ISA IUDOPluginExtension ------------------------
def GetUDSDescription(self):
  return "Sample UDO for dimension reducing quantities"

#--- ISA IUDOPluginExtension ------------------------
# Returns list of category names
 def GetCategoryNames(self):
  return ["UDOOutputs"]

#--- ISA IUDOPluginExtension ------------------------
# returns a list of quantity names for the supplied category name
def GetQuantityNames(self, catName):
  if catName == "UDOOutputs":
    return ["min_val", "max_val", "avg_val"]
  else:
    return []

#--- ISA IUDOPluginExtension ------------------------
# Returns an instance of QuantityInfo for the qtyName supplied or
# None if such a quantity could not be found
def GetQuantityInfo(self, qtyName):
  # All the quantities we have are simple doubles
  # we can leave them unitless
  return QuantityInfo(Constants.kDoubleParamStr)

#--- ISA IUDOPluginExtension ------------------------
# Returns list of UDSParams and list of dynamic properties
# Adds setup time properties to the propList
def GetInputUDSParams(self, udsParams, propList,
userSelectedDynamicProbes):
  # Add the probes. We need only one double quantity
  param1 = UDSProbeParams("probe1",
    "double quantity probe",
    Constants.kDoubleParamStr,
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
      "", "")
    udsParams.Add(param1)


    # Add the properties we want the user to supply
    # In this case, we will ask for a start/end range for
    # X parameters. Since we cannot reasonably provide defaults
    # as we have no idea what the sweep limits will be, we will
    # also ask if the limits are to be activated.
    prop = propList.AddNumberProperty("X Min", "0")
    prop.Description = "Start X value to consider"

    prop = propList.AddNumberProperty("X Max", "1")
    prop.Description = "End X value to consider"

    # For menus, the first option is the default.
    prop = propList.AddMenuProperty("Activate X Limits", ["No",
    "Yes"])
    prop.Description = "Activate X range"

    return True

#--- ISA IUDOPluginExtension ------------------------
# Returns list of UDSParams and list of dynamic properties
# output UDSDynamicProbeCollection probes
def GetDynamicProbes(self, probes):
    pass

#--- ISA IUDOPluginExtension ------------------------
# Returns list of sweeps names
# We have no sweeps as we reduce them.
def GetUDSSweepNames(self):
    return []

#-----------------------------------------------------------------
# IUserDefinedSolutionHandle API implementation.
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
# Calculates output values and sets them using
IUDSInputData/IUDSOutputData API.
def Compute(self, inData, outData, propList, progMon):

  # Get the sweeps associated with the probe and validate
  # use the probe name that we had defined earlier
  sweeps = inData.GetSweepNamesForProbe("probe1")
  if( sweeps == None or sweeps.Count > 1):
    AddErrorMessage(self.GetName() + "Unexpected sweep count 0 or >
    1 in Compute")
    return False

  # Get the data associated with our probe
  probeData = inData.GetDoubleProbeData("probe1")
  sweepData = inData.GetSweepsDataForProbe("probe1", sweeps[0])

  # Get the user specified properties.
  # Note that ideally, these "X Min" etc names should be
  # written as constant members and referred to in both
  # the GetInputUDSParams and in Compute to reduce the change
  # of typos.
  useXRangeProp = propList.GetMenuProperty("Activate X
  Limits").SelectedMenuChoice
  xRangeStart = propList.GetNumberProperty("X Min").ValueSI
  xRangeEnd = propList.GetNumberProperty("X Max").ValueSI

  # At this stage, one can look at the RequestedQuantities and
  # create a dictionary to later check against. However, I am
  # simply computing all the quantities.
  minVal = 0
  maxVal = 0
  avgVal = 0

  # Check if we need to perform range computation
  if useXRangeProp == "Yes":
    seenAny = False
    avgSum = 0
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
  count = 0


  # zip is used since we also need to pull in sweep data
  # an index and the array notation could also have been used
  for probeVal, sweepVal in zip(probeData, sweepData):
            if sweepVal < xRangeStart or sweepVal > xRangeEnd:
      pass


            # Note that in a better written script, this code
    would be
            # refactored into its own function to avoid code
            # duplication
            if not seenAny:
      minVal = probeVal
      maxVal = probeVal
      avgSum = probeVal
      seenAny = True
      count = 1
            else:
      if probeVal < minVal:
        minVal = probeVal
      if probeVal > maxVal:
        maxVal = probeVal
      avgSum += probeVal
      count += 1
  if seenAny:
            avgVal = avgSum/count
else:
  seenAny = False
  avgSum = 0
  for probeVal in probeData:
            if not seenAny:
      minVal = probeVal
      maxVal = probeVal
      avgSum = probeVal
      seenAny = True
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
            else:
        if probeVal < minVal:
          minVal = probeVal
        if probeVal > maxVal:
          maxVal = probeVal
        avgSum += probeVal
    if seenAny:
            avgVal = avgSum/probeData.Count


# Finally set the output values. Note that these are always set
# as lists even if we have just one item.
outData.SetDoubleQuantityData("min_val", [minVal])
outData.SetDoubleQuantityData("max_val", [maxVal])
outData.SetDoubleQuantityData("avg_val", [avgVal])


# And we are done.
return True
```

AddErrorMessage

Call this method to convey an error condition to the user.

UI Access	NA		
Parameters	Name <message>	Type String	Description Error message
Return Value	None		

Python Syntax	AddErrorMessage (<*message*>)
Python Example	`AddErrorMessage(self.GetName() + "Unexpected sweep count 0 or > 1 in Compute"))`

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AddInfoMessage

Call this method to convey an informational message to the user. This is the call to use when outputting messages for debugging purposes.

UI Access	NA		
Parameters	Name <message>	Type String	Description Warning message
Return Value	None		

Python Syntax	AddInfoMessage (<*message*>)
Python Example	`AddInfoMessage("Enter starting X value.")`

AddWarningMessage

Call this method to convey a warning message, typically used for conditions that are not ideal but can be tolerated by the script.

UI Access	NA		
Parameters	Name <message>	Type String	Description Warning message
Return Value	None		

Python Syntax	AddWarningMessage (<*message*>)
Python Example	`AddWarningMessage("Unexpected sweep count in Compute")`

User-Defined Outputs: Script Organization

As described in the Introduction section, the UDO scripts should all reside under the **UserDefinedOutputs** folder under either of the three library locations (system, user, or personal).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using Script Libraries

If you decide that you need base classes, additional data files, and etc., to organize your UDOs better, you can do so. This type of library organization allows code reuse between similar UDOs and can be very helpful. There is special support provided for this type of script-library organization:

- **All script-library and other support files need to be in a *Lib* sub-directory under the *UserDefinedOutputs* directory**. Any *.py* files found in such **Lib** directories are ignored and not displayed in the GUI as a valid UDO choice.
- For a UDO script at any given directory depth, all Lib directories in its parent directories will be automatically added to the system include path (and so, any support script files from any Lib directory through the top level *UserDefinedOutputs* directory can be imported).

Using Additional .NET Assemblies

Because the UDO functionality uses IronPython, we have access to the full .NET eco system. If needed, any subset of the UDO functionality can be implemented in any .NET language and used by the UDO script. There are simple rules to follow to achieve this.

1. Build your .NET assembly for .NET 2.0 runtime.
2. Drop the built assembly in any **Lib** directory upstream of the UDO script location: that is, if you have your UDO script in *C:\Users\x\PersonalLib\UserDefinedOutputs\a\b\c\myudo.py* and have a .NET assembly called *com.Acme.UDOLib* You can keep the .NET assembly under:
    - UserDefinedOutputs\Lib
    - UserDefinedOutputs \a\Lib
    - UserDefinedOutputs \a\b\Lib
    - UserDefinedOutputs\a\b\c\Lib
3. Add the following lines to your Python script:
    - `Import clr`
    - `clr.AddReference("com.Acme.UDOLib")`
    - `import com.Acme.UDOLib` -or- `from com.Acme.UDOLib import * etc`

If for some reason you cannot place the .NET assemblies into a Lib directory under *UserDefinedOutputs*, you need to do a couple more steps before step 3 listed above.

```
Import sys
sys.path.append("full path to your .NET assembly location")
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Toolkit

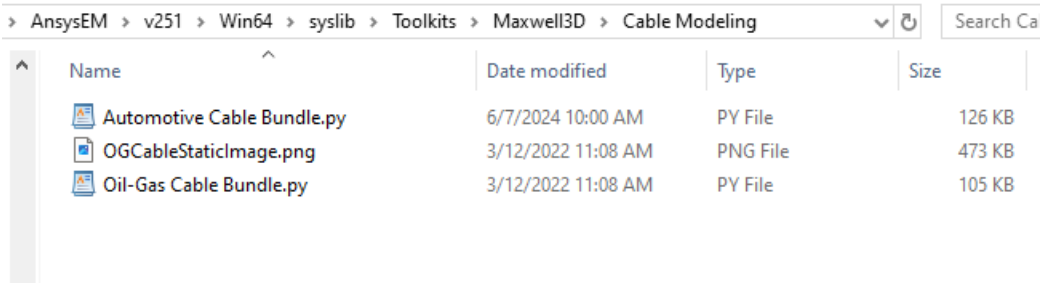
The **Toolkit** entry in the **Maxwell** menu provides access to a design type-specific IronPython script or a module-specific task. The Python script provides a default but customizable UI and canned automations. The default UI is a modal dialog box.

The UDD format can be adjusted/extended as needed. This allows the Maxwell application the flexibility to support changing standards or multiple standards. Similarly, you can create new UDDs to customize the test report.

UDDs are saved in the project result directory. This type of report can be created prior to simulation, but will only be populated when there is a solution.

Undo/Redo/Scripting is supported. If a script includes multiple commands, it takes multiple undo (s) to revert all the changes made by the Python script. Also, note that the script playback might not work if the baseline of the design has changed.

A directory called *Toolkits* appears in the *syslib*, *userlib*, and *personallib* of the Maxwell installation. You can add product specific-directories (Maxwell 2D, Maxwell 3D, RMxprt) in *Toolkits*, and each directory contains toolkits (scripts) that are specific for that product.



Use the **Maxwell** > **Toolkit** > **Update Menu** command to add newly added toolkit scripts to the **Toolkit** menu. Menu items for files found in *<installation>/syslib* will be inserted at the first level of the menu.

Toolkit Examples

The Maxwell application includes the following toolkits for generating 2D and 3D cable bundle models:

- AutomotiveCableBundle
- Oil-GasCableBundle

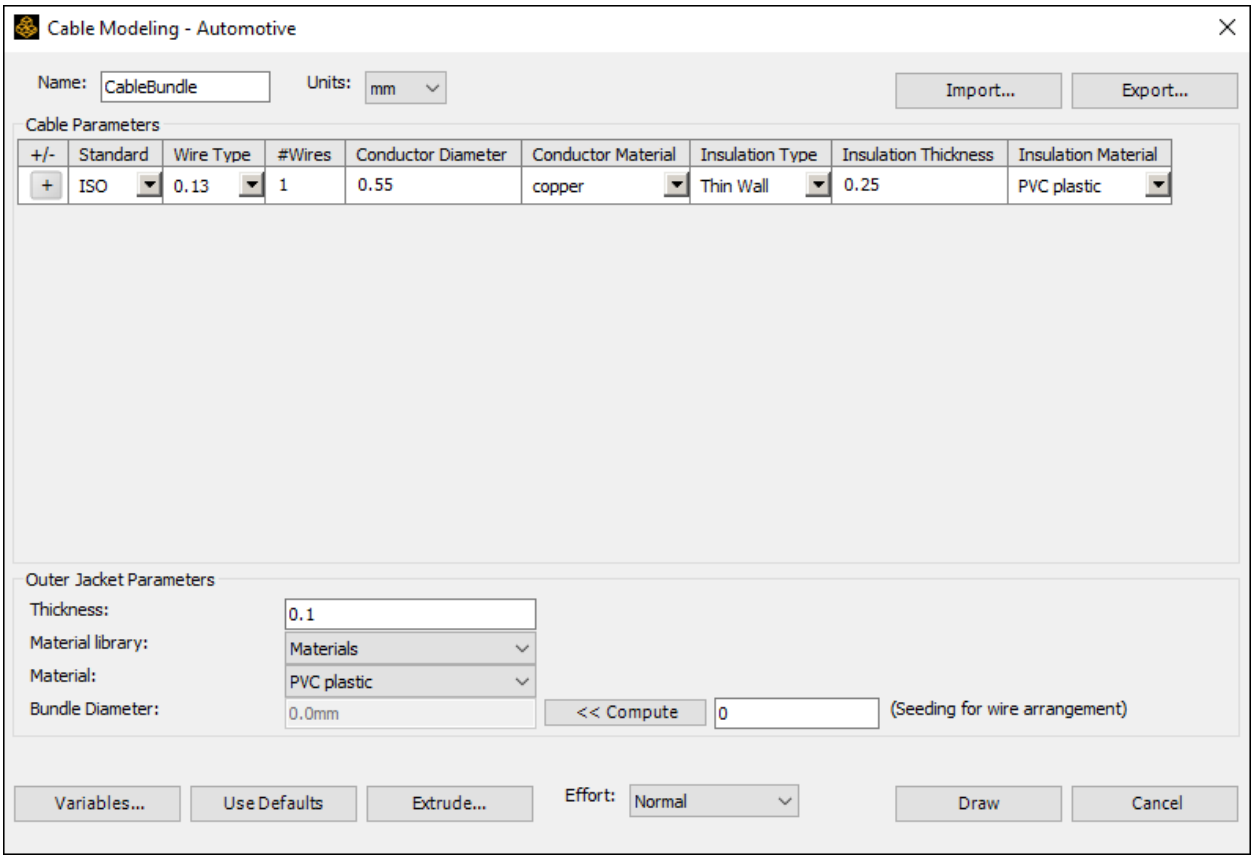
Automotive Cable Bundle Toolkit

To work with the Automotive Cable Bundle toolkit:

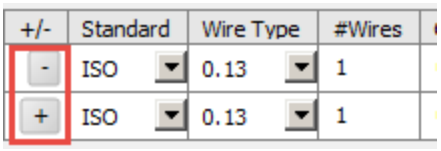
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Maxwell 3D** (or **Maxwell 2D**) > **Toolkit** > **Cable Modeling** > **Automotive Cable Bundle**.

   The **Cable Modeling - Automotive** window appears with a default set of **Cable Parameters** listed.



2. Enter a **Name** for the toolkit, and select **Units**.
3. If you have a *.csv or *.tab file containing cable and outer jacket parameters, you can **Import** it.
4. To manually modify the **Cable Parameters**:
   - Using the +/- column, add or remove wire types to create a bundle. Clicking the **+** button adds a new row. All rows but the initial row will have a **-** button to remove that row.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- In the **Standard** field, choose between the **ISO** or **AWG** standard types.
- In the **Wire Type** field, use the drop-down menu to select the wire type.
- In the **#Wires** field, adjust the number of wires as necessary.
- In the **Conductor Diameter** field, specify the diameter in the unit you selected.
- In the **Conductor Material** field, use the drop-down menu to select a material.
- In the **Insulation Type** field, use the drop-down menu to select either **Thin Wall** or **Ultra Thin Wall**.
- In the **Insulation Thickness** field, specify a value in the unit you selected.
- In the **Insulation Material** field, use the drop-down menu to select a material.

5. To manually modify the **Outer Jacket Parameters**:
   - In the **Thickness** field, specify a value for the inner diameter in the unit you selected.
   - Use the **Material Library** drop-down menu to select the library containing the material you wish you specify, then use the **Material** drop-down menu to select the material.
   - Populate the **Bundle Diameter** field. After specifying all wires to be included in the bundle, click **Compute** to calculate the minimal bundle diameter of the bundle. The value automatically populates.

     > **Note:** The wire radii are adjusted downward very slightly (on the order of 0.1%) at the end of the packing process to eliminate any residual overlaps between the wires and to make the model easier to mesh.

     > **Note:** The Circle Packing algorithm is used to compute the values. It tries to automatically pack the conductors in the bundle tightly with a minimum of empty space between them. It generates and tests a large number of random variations in the conductor positions and attempts to minimize the overall diameter of the bundle.

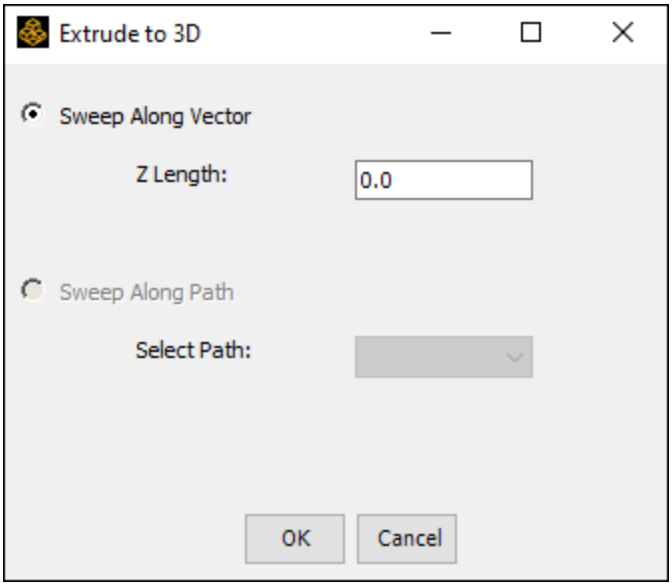
   - You can adjust the **Bundle Diameter** generated by specifying a **Seeding for wire arrangement** value. The value entered here is the seed value for the pseudo-random number generator used in the conductor packing process. Changing this value results in different arrangements of the conductors. This can be useful for performing statistical analysis of the cables.
6. To save your cable and outer jacket parameters for future use, click **Export** to create a *.csv or *.tab file for later import.
7. If desired, click **Use Defaults** to restore the default values. This removes any rows previously added and clears all custom values.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. To add new variables, click **Variables**. This opens the **Edit Variables** dialog box, listing all variables that are present at the project and design levels. Click **Add** to create new variables. See: Working with Variables.
9. Click **Draw** to draw the 2D cross-section geometry. A validation check is run. This catches errors such as any variable or parameter missing a value.
10. For 3D designs, you can create a 3D geometry:

    The **Effort** drop-down menu contains four options: **Low**, **Normal**, **High**, and **Very High**. This controls the trade-off between packing quality (how tightly the conductors are packed together) and the time spent generating the model. Low effort runs relatively fast and may be good enough for small numbers of conductors, but may leave significant space in the model when there are many conductors. High effort may be required for models with large numbers of conductors (100+), but could take several minutes to run.

    Click **Extrude**. The **Extrude to 3D** window appears.



- Click **Sweep Along Vector** to extrude along the z axis. Z length is the input parameter.
- Click **Sweep Along Path** to extrude along a selected path.

> **Note:** 3D models of cables are hard to solve if the ratio of extruded length to cross-section diameter is high.

Oil-Gas Cable Bundle Toolkit

To work with toolkits in the oil and gas industries:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Maxwell 2D** (or **Maxwell 3D**) > **Toolkit** > **CableModeling** > **Oil-GasCableBundle**.

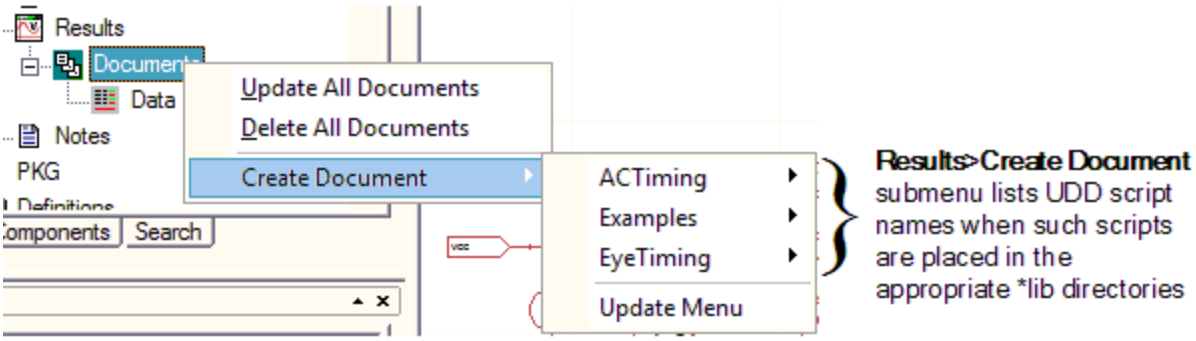
   The **Cable Modeling - Oil & Gas** dialog box appears.
2. Type the **Name** of the toolkit, and select the **Units** to be used for displayed values.
3. Modify the various **Cable Parameters** as needed.
4. If you have previously imported a customized set of parameters, you can click **Use Defaults** to use the default values.
5. To export the cross-section parameters into a .csv or .tab file, click **Export**. The **Export Cable Parameters** dialog box appears. Specify the **File name** and **type** (either comma delimited **.csv** or tab delimited **.tab**), and click **Save**.
6. To read in either a comma delimited **.csv** or tab delimited **.tab** file and populate parameters in the toolkit panel, click **Import**. The **Import Cable Parameters** dialog box appears. Specify the **File name** and **type**, then click **Open**.
7. To add new variables, click **Variables**. This opens the **Edit Variables** dialog box. All variables that are already present in the project and design levels are listed. Click **Add** to add a new row to create new variables.
8. Optionally, for 3D designs only, click **Extrude**. (The **Extrude** button is not present for 2D designs.) The **Extrude to 3D** dialog box appears.
   - Click **Sweep Along Vector** to extrude along the z axis. Z length is the input parameter.
   - Click **Sweep Along Path** to extrude along a selected path.

   > **Note:** 3D models of cables are hard to solve if the ratio of extruded length to cross-section diameter is high.
9. Click **Draw** to draw the 2D or 3D cross-section geometry. A validation check is run. This catches errors like any variable missing a value, or any parameter missing a value. For 3D designs, if you chose to **Extrude**, the 3D geometry is drawn.

User-Defined Documents (UDDs)

User-defined documents (UDDs) are custom reports that you define through IronPython scripts. Once placed in a Lib directory, you can access the scripts via the **Create Document** command. The scripts describe a *Create User Defined Document* dialog box that lets you specify trace and solution inputs. After you confirm your input selections, XML, HTML, and PDF documents are generated. A web browser window opens to display the generated HTML file. The created document appears in the Project Manager, under Results in the Documents folder.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The general UDD process flow is as follows:

The UDD Python scripts must be placed in the **UserDefinedDocuments** directory under either of **syslib**, **userlib**, or **Personallib** with any subdirectory structure needed. The Lib directory can contain Python scripts that have common code that other scripts can use.

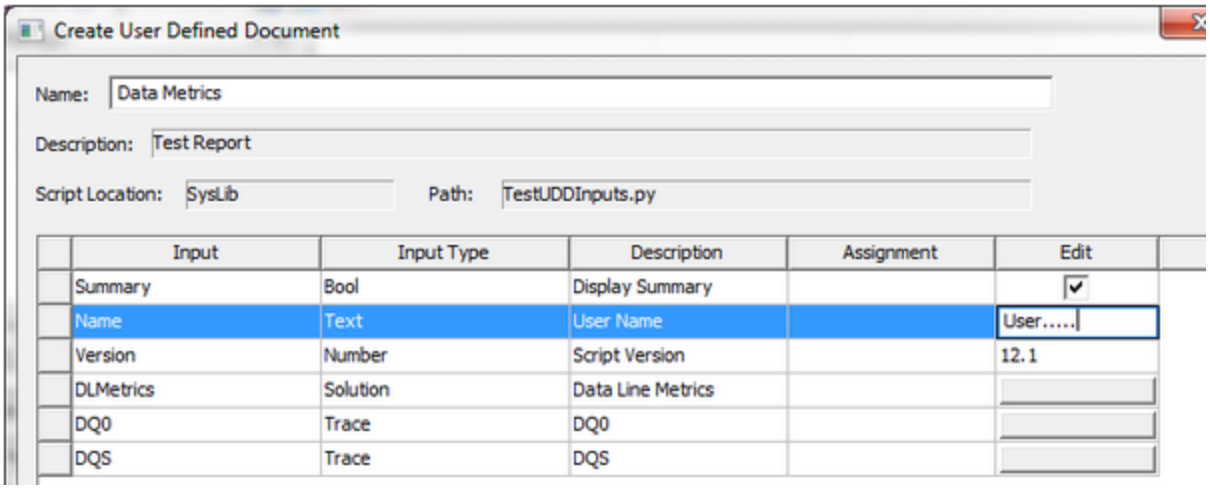
Use **Maxwell** > **Results** > **Create Document**> **Update Menu** to refresh the menu to include the new UDD scripts that have been copied to syslib, userlib, or Personallib, or to exclude them if they have been deleted after the launch of desktop.

The UDD scripts that are in syslib/UserDefinedDocuments, userlib/UserDefinedDocuments, or Personallib/UserDefinedDocuments are available through the **Maxwell** > **Results** > **Create Document** menu.

Create User-Defined Document Inputs

User-defined documents allow data from traces, solutions, and report types as inputs. A UDD can specify the named inputs, which you select or enter the values of in the *Create User Defined Document* dialog box that displays when you run **Maxwell** > **Results** > **Create Document**> <*scriptName*>.

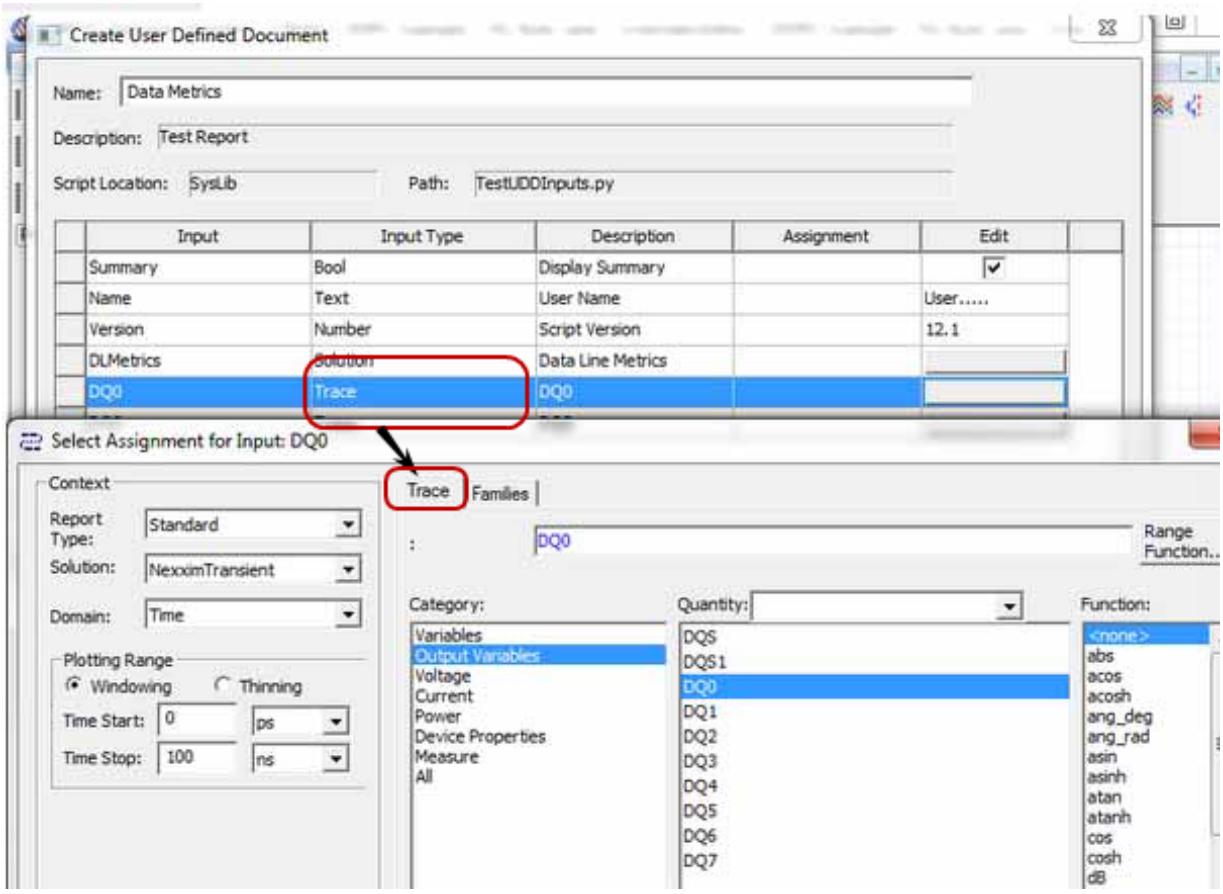
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Input Types can be of Boolean, number, text, trace, or solution type. The boolean, number, and text type can be given a default value that you can interactively override when the document is created or modified. For example, you can select a trace when you create or modify a UDD document. The trace data is available to the user and can be accessed from the Python script.

At the time of selection, you can choose from the *Reporter* dialog box the report type (Standard, Eye Diagram, User Defined), solution name, context, and the quantity for which you want the trace data.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

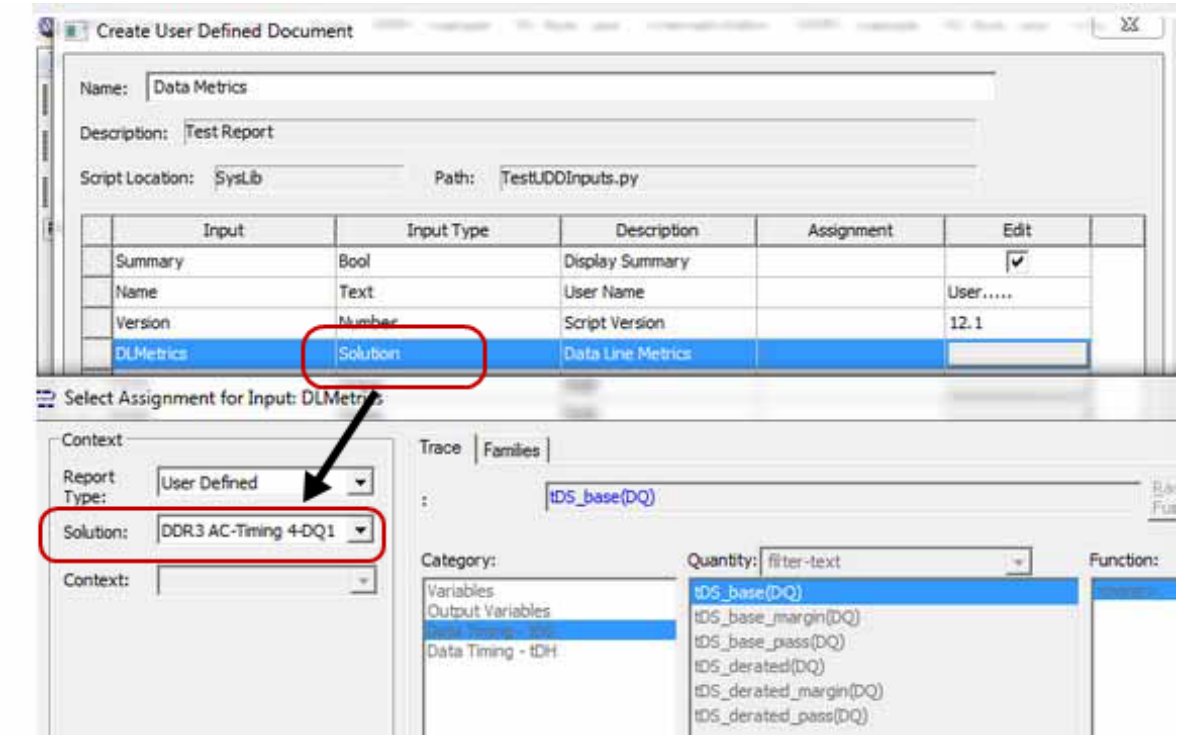


Input Type can also be Solution. You can select an entire solution when the document is created or modified. The solution data in its entirety is now available to the user and can be accessed from the Python script.

At the time of selection, you can choose from the reporter dialog the report type (Standard, Eye Diagram, User Defined), solution name, and context. A specific quantity cannot be selected since data for all quantities in the solution are available.

> **Note:** The Category/Quantity/Function portion of the dialog is disabled for user input.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

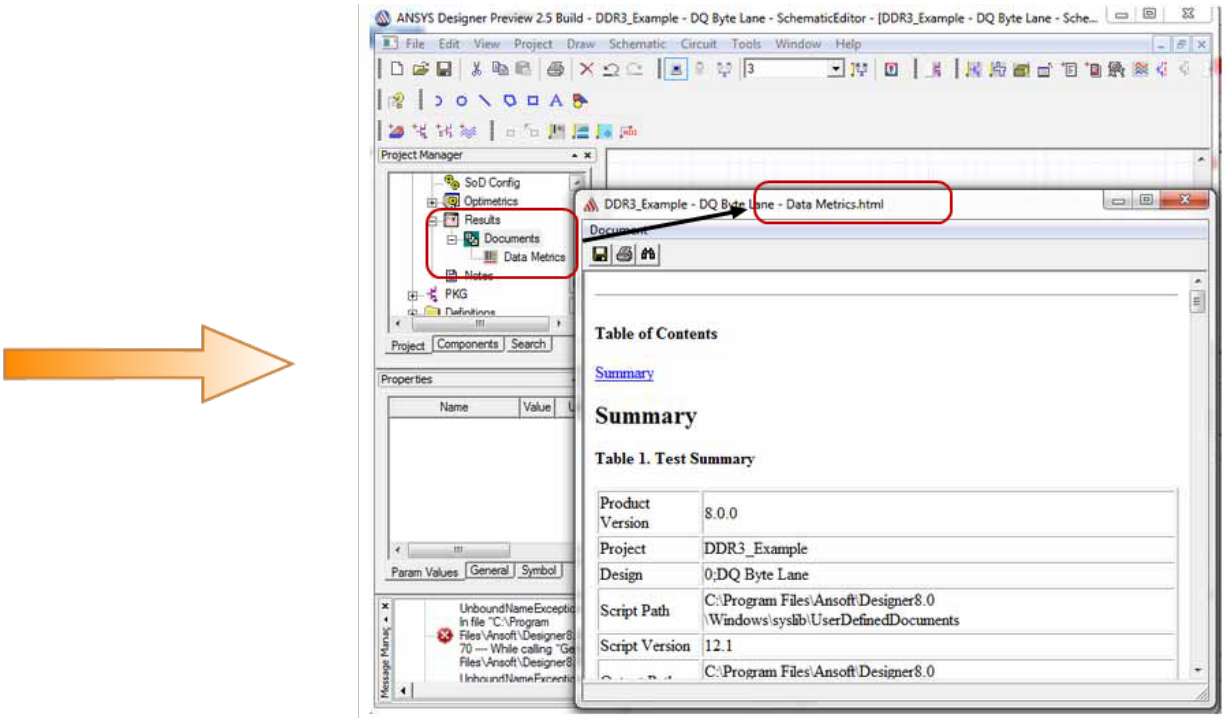


UDD Document Creation and Display

After all the input selections for a UDD are confirmed, based on the script, XML, HTML, and PDF documents are generated based on the inputs provided by the user. (The XML, HTML, and PDF generation is based on specific calls in the Python script, which are explained in a following section.) A web browser window also opens to display the generated HTMLfile.

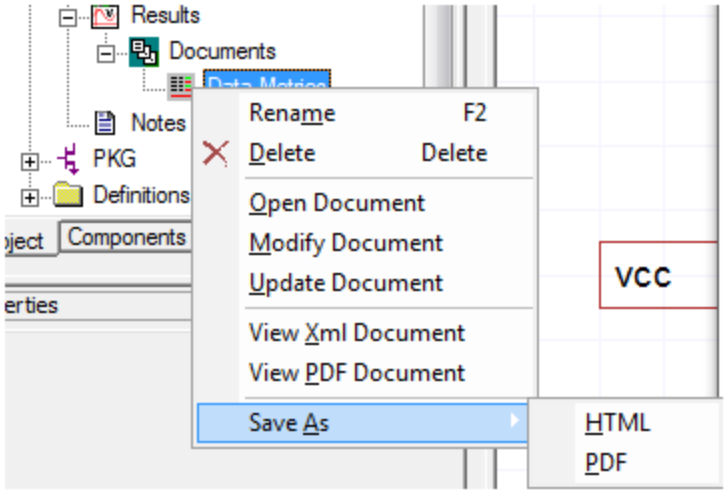
The created document will be placed under a new folder named **Documents** under the **Results** folder. All documents that are created by the user for the design will be placed under this folder.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Managing Documents Listed in the Project Window Under Results

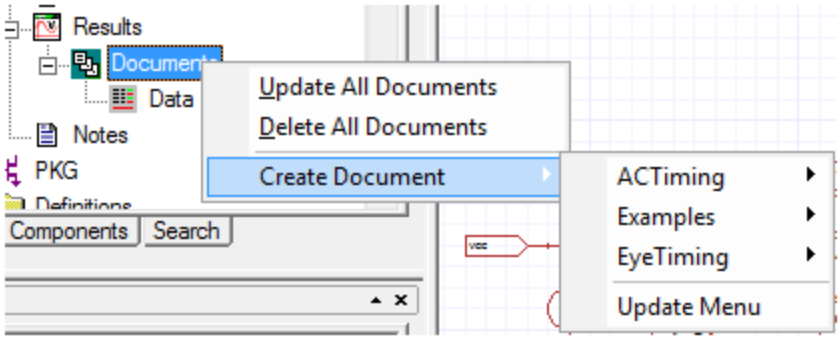
Right-click a user-defined document displayed in the Project Manager tree to bring up a menu where you can rename, delete the document. **Open document** opens the web browser with the html document. **Modify document** opens the setup dialog box, where you can change the selections for the input. To view the XML and the PDF document simply choose the appropriate menu items. There is also a menu item to save the document in a different location.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Documents Folder's Context Menu

Right-click the documents folder has the menu options to **Update All Documents** or **Delete All Documents**. It also provides the option of creating a document from here.

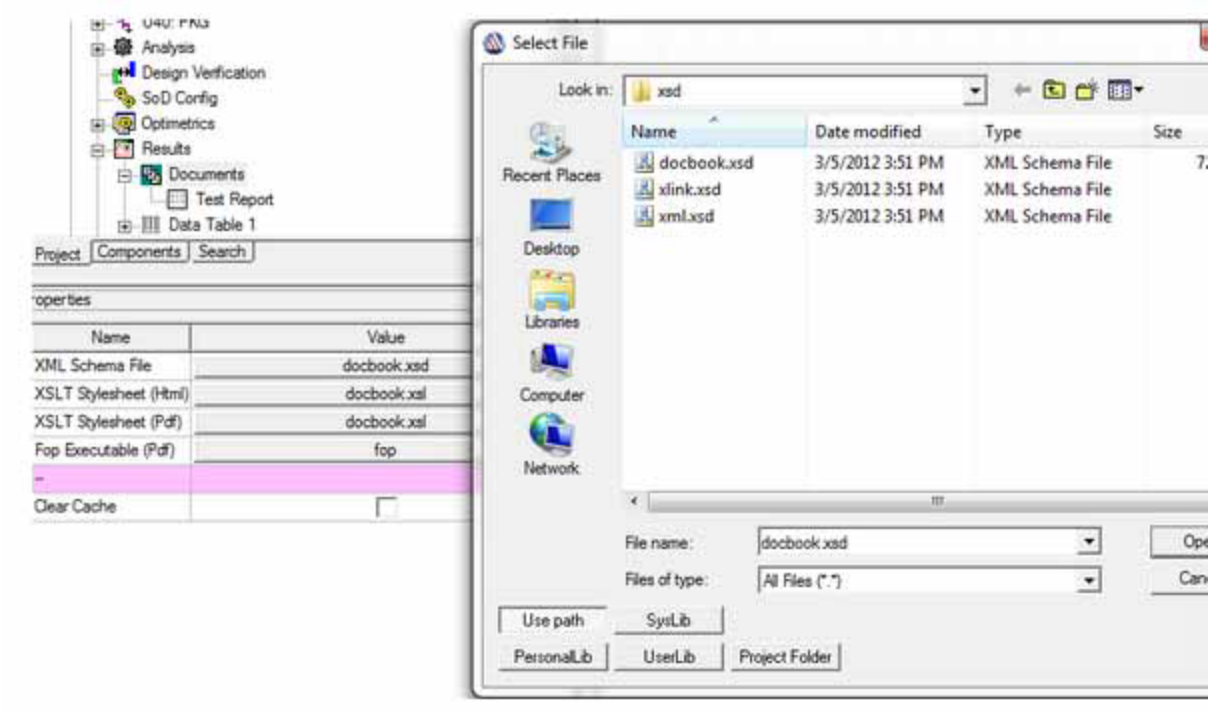


Document Folder's Property Window

When the documents folder is selected, the *Property* window shows the following properties:

- **XML Schema File** - File path to the XML schema file.
- **XSLT StyleSheet (HTML)** - File path to the XSLT stylesheet file used for HTML generation.
- **XSLT StyleSheet (Fo)** - File path to the XSLT stylesheet file used for PDF generation.
- **Fop Executable (PDF)** - File path the Fop executable used for PDF generation.
- **Clear Cache** - Clears the cached XSL transform object and forces creation of a new one. (The caching is done to save time during document generation, so subsequent generation or update of the document can use the cached transform object. But sometimes you may want to force a recompile of the document if you change the stylesheet).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The XML, HTML, and PDF generation require the XML schema file and XSLT stylesheets to generate proper output. In addition, the PDF generation requires a FOP executable. You can use the defaults provided in the installation or provide the file paths of your own preferred stylesheets and fop executable installed in his machine.

Viewing UDDs with an HTML Web Browser

The XML and HTML documents can be viewed in a web browser with some basic functionality like printing the document, searching the document for a phrase or sentence, and saving the document.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

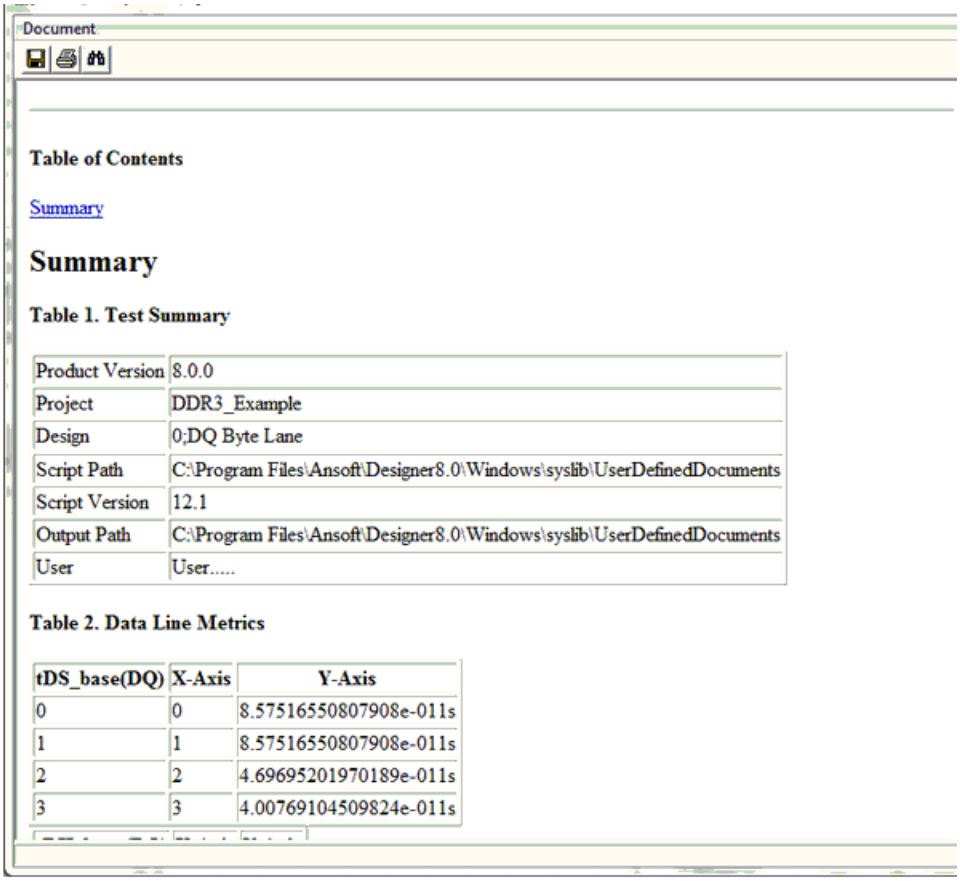

Document

Table of Contents

Summary

Summary

Table 1. Test Summary

Product Version	8.0.0
Project	DDR3_Example
Design	0;DQ Byte Lane
Script Path	C:\Program Files\Ansoft\Designer8.0\Windows\syslib\UserDefinedDocuments
Script Version	12.1
Output Path	C:\Program Files\Ansoft\Designer8.0\Windows\syslib\UserDefinedDocuments
User	User.....

Table 2. Data Line Metrics

tDS_base(DQ)	X-Axis	Y-Axis
0	0	8.57516550807908e-011s
1	1	8.57516550807908e-011s
2	2	4.69695201970189e-011s
3	3	4.00769104509824e-011s

UDD Script Libraries

Base classes and data files shared between similar UDDs can be organized to reuse the code in a better way. All script-library and other support files need to be in a Lib sub-directory under the UserDefinedDefinitions directory. Any .py files found in such Lib directories are ignored and not displayed in the GUI as a valid UDD choice. For a UDD script at any given directory depth, all Lib directories in its parent directories will be automatically added to the system include path (and so, any support script files from any Lib directory through the top level UserDefinedDefintions directory can be imported).

The UDD functionality uses IronPython so we have access to all the .NET assembles. If needed, any subset of the UDD functionality can be implemented in any .NET language and used by the UDD script. There are simple rules to follow to achieve this.

1. Build your .NET assembly for .NET 2.0 runtime.
2. Drop the built assembly in any Lib directory upstream of the UDD script location: that is, if you have your UDD script in C:\Users\x\PersonalLib\UserDefinedDefintions\a\b\c\myudd.py and have a .NET

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

assembly called com.Acme.UDDLib You can keep the .NET assembly under:

- UserDefinedDefintions\Lib
- UserDefinedDefintions\a\Lib
- UserDefinedDefintions\a\b\Lib
- UserDefinedDefintions\a\b\c\Lib

3. Add the following lines to your Python script:

```
Import clr

clr.AddReference("com.Acme.UDDLib")

import com.Acme.UDDLib -or- from com.Acme.UDDLib import * etc
```

If for some reason you cannot place the .NET assemblies into a Lib directory under UserDefinedDefintions, you need to do a couple more steps before step 3 listed above:

```
Import sys

sys.path.append("full path to your .NET assembly location")
```

User-Defined Documents: Python Script API

A User-Defined Documents (UDD) extension is implemented as an IronPython script that defines a class with a specific name: UDDExtension which derives from a specific base class IUDDPluginExtension and implements its abstract methods.

This API supports multiple data types in the forms of constants and classes. It also has several input interfaces.

User-Defined Document scripting commands are provided in this product's scripting guide. An complete example with a line by line explanation of how to use these methods is available.

Import Statements

The base class to be used and the types it uses in turn are contained in .NET assemblies. The use of these requires that the assemblies be imported into the UDD script: the following import statements should be added to the top of the python script:

```
from Ansys.Ansoft.DocGeneratorPluginDotNet.DocGenerator.API.Data
import *
from
Ansys.Ansoft.DocGeneratorPluginDotNet.DocGenerator.API.Interfaces
import *
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Data Types Used in UDD Python Scripts

There are several types that you must use while authoring a Python script. Some of them are used to pass data from the UI to a Python script and to provide interfaces for working with this data. Some are used to pass data from a Python script to the UI.

To pass data from a Python script to the UI, the objects of the C# class must be created in the Python script using their C# constructors. Then they can be set as a function's return values or set to the output parameters using their API.

Constants

- **kTraceTypeStr** : string constant used to specify an input of trace type
- **kSolutionTypeStr** : string constant used to specify an input of solution type
- **kNumberTypeStr** : string constant used to specify an input of number type
- **kTextTypeStr** : string constant used to specify an input of text type
- **kBoolTypeStr** : string constant used to specify an input of boolean type
- **kStandardReportStr** : string constant to specify a standard report
- **kEyeDiagramReportStr** : string constant to specify an eye diagram report
- **kUserDefinedReportStr** : string constant to specify a user defined report
- **kSweepDomainStr** : string constant to specify the sweep domain
- **kTimeDomainStr** : string constant to specify the time domain

Abstract Classes

- IProgressMonitor Abstract Class
- UDDExtension Class
- UDDInputData class
- UDDInputParams class

IProgressMonitor Abstract Class

The object of this class is a progress monitor. It displays a task's calculated progress in the UI and checks if the user has requested to abort the computation. Use this object for computations that take a long time to complete or if the UI might freeze during the computation. When displayed in the application, each progress message has four items:

- A task name
- A sub-task name
- The progress amount
- A button to abort the task in progress

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This class provides the following functionality and abort interaction:

- SetTaskName (string taskName)
- SetSubTaskName (string subTaskName)
- BeginTask (string name)
- SetTaskProgressPercentage(int progressPercent)
- CheckForAbort()
- EndTask (bool passFail)

Example:

```
progMon.BeginTask("Process DQS")
progMon.SetSubTaskName("Compute UI segments")
progMon.SetTaskProgressPercentage(33)
progMon.SetSubTaskName("Compute the rest")
progMon.SetTaskProgressPercentage(100)
progMon.EndTask(True)
```

> **Note:**
>
> There should be the same number of calls to BeginTask as to EndTask.

SetTaskName

Sets the name of the task whose progress is being monitored.

UI Access	NA		
Parameters	Name <*TaskName*>	Type String	Description Name of the task
Return Value	None		

Python Syntax	SetTaskName (<*TaskName*>)
Python Example	`progMon.SetTaskName("Compute UI")`

SetSubTaskName

Sets the name of the subtask whose progress is being monitored.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UI Access	NA		
Parameters	Name <*TaskName*>	Type String	Description Name of the task
Return Value	None		

Python Syntax	SetSubTaskName (<*TaskName*>)
Python Example	`progMon.SetSubTaskName("Compute UI segments")`

BeginTask

Sets the name of the task whose progress will be monitored. Call this method only once for each task and subtask.

UI Access	NA		
Parameters	Name <*Name*>	Type String	Description Name of the task
Return Value	None		

Python Syntax	BeginTask (<*Name*>)
Python Example	`progMon.BeginTask("Process DQS")`

SetTaskProgressPercentage

Sets the progress percentage of a task to a specific value.

UI Access	NA		
Parameters	Name <Percent>	Type Integer	Description Percent of progress
Return Value	None		

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Python Syntax	SetTaskProgressPercentage (<*Percent*>)
Python Example	`progMon.SetTaskProgressPercentage(33)`

CheckForAbort

Checks if a user has aborted the task. Call this method if the quantities are computationally expensive. This method can be called multiple times.

UI Access	Abort button on the progress dialog.
Parameters	None
Return Value	Boolean flag that if true indicated that the user requested an abort and you should call EndTask. If false, allow the task to continue.

Python Syntax	CheckForAbort
Python Example	`should_abort = progMon.CheckForAbort`

EndTask

Stops a task from running. Call this method only once for each task and subtask. In a script, there should be the same number of BeginTask calls as **EndTask** calls.

UI Access	NA		
Parameters	Name <PassFail>	Type Boolean	Description Flag that indicates whether to continue the main task or not. If true, the task will continue to process the next subtask. If false, the main task stops and an error icon appears in the UI.
Return Value	None		

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Python Syntax	EndTask (<*PassFail*>)
Python Example	`progMon.EndTask(True)`

UDDExtension Class

The UDD itself should be implemented as an IronPython class called **UDDExtension** which must derive from the IUDDPluginExtension abstract base class (from the **Ansys.Ansoft.DocGeneratorPluginDotNet.DocGenerator.API.Interfaces** namespace).

Note that power users could derive a class hierarchy tuned toward a specific type of UDDs and that they can derive from their own base classes. The only requirement is that directly or indirectly, the UDD class must derive from **IUDDPluginExtension**.

Example:

```
def BaseClassUDD ((IUDDPluginExtension):
#base class implementation
…
def UDDExtension ((BaseClassUDD):
#UDD class implementation
…
```

> **Note:**
>
> This class is modeled after the UDO class UDOExtension, therefore the usage is similar.

UDDInputData Class

This class contains user input data and can be used in the generated document. UDDInputData has the constructor UDDInputData(string name) and the following properties accessors:

Property accessor	Purpose	Form
Name	Get/Set the name of an input.	string Name();
Type	Get/Set the type of an input	string Type();

The data can be any of the following types, based on the UDDInputParams type specified in the script.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- UDDInputBool
- UDDInputDouble
- UDDInputSolution
- UDDInputText
- UDDInputTrace

UDDInputBool

Method purpose	Return Value	Data type of the returned value in C#
Retrieves boolean value of the input data.	a boolean value of input data	string Data();

UDDInputDouble

Method purpose	Return Value	Data type of the returned value in C#
Retrieves double data.	a double value of input data	double Data();

UDDInputSolution

Method purpose	Return Value	Data type of the returned value in C#
Retrieves x and y double data for a probe	Data collection	IDictionary<double, double> DoubleData (string name);
Retrieves x and y double data for a probe for a particular variation	Data collection	IDictionary<double, double> DoubleData (string name, IDictionary<string, string> variation);
Retrieves x and y complex data for a probe	Data collection	IDictionary<double, double[]> ComplexData (string name);
Retrieves x and y complex data for a probe for a particular variation	Data collection	IDictionary<double, double[]> ComplexData (string name, IDictionary<string, string> variation);
Retrieves x and y text data for a probe	Data collection	IDictionary<string, string> TextData(string name, IDictionary<string, string> variation);
Retrieves x and y text data for a probe for a particular variation	Data collection	IDictionary<string, string> TextData(string name);

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Retrieves a list of category names	A string collection	IList<string> CategoryNames();
Retrieves a list of quantity names given a category name	A string collection	IList<string> QuantityNames(string category);
Retrieves a list of variable values	A string collection	IList<Dictionary<string, string>> VariableValues();

UDDInputText

Method purpose	Return Value	Data type of the returned value in C#
Retrieves string data	a string value of input data	string Data();

UDDInputTrace

Method purpose	Return Value	Data type of the returned value in C#
Retrieves x and y double data	Double	IDictionary<double, double> DoubleData();
Retrieves x and y double data for a particular variation	Double	IDictionary<double, double> DoubleData (IDictionary<string, string> variation);
Retrieves x and y complex data	Double	IDictionary<double, double[]> ComplexData();
Retrieves x and y complex data for a particular variation	Double	IDictionary<double, double[]> ComplexData (IDictionary<string, string> variation);
Retrieves x and y text data	Text	IDictionary<string, string> TextData();
Retrieves x and y text data for a particular variation	Text	IDictionary<string, string> TextData (IDictionary<string, string> variation);
Retrieves a list of variable values	A string collection	IList<Dictionary<string, string>> VariableValues();
Retrieves an image of the trace data in a plot	An image	string Image();

UDDInputParams Class

The objects of this class must be created in Python script in the GetUDDInputParams and SetUDDInputParams functions.

Attributes :

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Input Name (string)
- Input Description (string)
- Input Type (string) (This can be boolean, number, text, trace, or solution.)
- BoolData (boolean)
- DoubleData (double)
- TextData (string)
- ReportType (string)
- SolutionName (string)
- DomainName (string)

Constructors:

- UDDInputParams(string name, string description, string type)
- UDDInputParams(string name, string description, string type, bool data)
- UDDInputParams(string name, string description, string type, double data)
- UDDInputParams(string name, string description, string type, string data)
- UDDInputParams(string name, string description, string type, string reportType, string solutionName, string domainName)

Property Accessors :

- Name : Get/Set the name of an input
- Description : Get/Set the description of an input
- Type : Get/Set the type of an input
- BoolData : Get/Set the data of a boolean input
- DoubleData : Get/Set the data of a number input
- TextData : Get/Set the data of a text input
- ReportType : Get/Set the report type
- SolutionName : Get/Set the name of the solution
- DomainName : Get/Set the name of the domain

UDD Input Interfaces

The Generate function takes in a list of inputs which allows the user to access data from the design.

IUDDInputBool : This interface exposes 3 methods.

- Name() : Gets the inputs name.
- Type() : Gets the input type.
- Data() : Gets the boolean data, set by the user in the setup dialog.

IUDDInputDouble : This interface exposes 3 methods.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Name() : Gets the inputs name.
- Type() : Gets the input type.
- Data() : Gets the double data, set by the user in the setup dialog.

IUDDInputText : This interface exposes 3 methods

- Name() : Gets the inputs name.
- Type() : Gets the input type.
- Data() : Gets the text data, set by the user in the setup dialog.

IUDDInputTrace : This interface exposes 9 methods

- Name() : Gets the inputs name.
- Type() : Gets the input type.
- DoubleData() : Method used to return x and y double data as a IDictionary<double, double>
- DoubleData(IDictionary<string, string> variation) : Method used to return x and y double data as a IDictionary<double, double>, given a variation.
- ComplexData() : Method used to return x data and y complex data as a IDictionary<double, double[]>
- ComplexData(IDictionary<string, string> variation) : Method used to return x data and y complex data as a IDictionary<double, double[]>, given a variation.
- TextData() : Method used to return x data and y data as a IDictionary<string, string>
- TextData(IDictionary<string, string> variation) : Method used to return x data and y data as a IDictionary<string, string>, given a variation.
- VariableValues() : Method used to get a list of variations as a IList<Dictionary<string, string>>

IUDDInputSolution : This interface exposes 11 methods

- Name() : Gets the inputs name.
- Type() : Gets the input type.
- DoubleData(string name): Method used to return x and y double data as a IDictionary<double, double>, given a quantity name.
- DoubleData(string name, IDictionary<string, string> variation) : Method used to return x and y double data as a IDictionary<double, double>, given a quantity name and a variation.
- ComplexData(string name) : Method used to return x data and y complex data as a IDictionary<double, double[]>, given a quantity name.
- ComplexData(string name, IDictionary<string, string> variation) : Method used to return x data and y complex data as a IDictionary<double, double[]>, given a quantity name and a variation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- TextData(string name) : Method used to return x data and y data as a IDictionary<string, string> given a quantity name.
- TextData(string name, IDictionary<string, string> variation) : Method used to return x data and y data as a IDictionary<string, string>, given a quantity name and a variation.
- CategoryNames() : Method to return a list of category names in the solution as an IList<string>
- QuantityNames(string category) : Method to return a list of quantity names in the solution as an IList<string>, given a category.
- VariableValues() : Method used to get a list of variations as a IList<Dictionary<string, string>>

Examples

```
def Generate(self, input, docgen, progMon):
# Getting the boolean data set by the user
boolinput = input[0].Data()
# Getting the double data set by the user
dblinput = input[1].Data()
# Getting the text data set by the user
textinput = input[2].Data()
# Getting the category names in a solution
categories = input[3].CategoryNames()
# Getting the quantity names based on a category
quantities = input[3].QuantityNames(categories[0])
# Getting the XY data from the trace
xydata = input[4].DoubleData()
```

Python Script to Define Document

To define a document in Python script:

```
Set oModule = oDesign.GetModule("UserDefinedDocuments")
% Add a UDD
oModule.AddDocument(
[ "NAME:Test Report", "Test Report", "SysLib",
"Examples/TestUDDInputs",
  [ "NAME:Inputs",
    [
      "NAME:DLMetrics","Solution", "Data Line Metrics", -1, -1
    ],
    [
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
      "NAME:DQ0", "Trace", "DQ0", -1, -1
    ],
    [
      "NAME:DQS", "Trace", "DQS", -1, -1
    ],
    [
      "NAME:Name", "Text", "User Name", ["User Name"]
    ],
    [
      "NAME:Summary", "Bool", "Display Summary", [True]
    ],
    [
      "NAME:Version", "Number", "Script Version" [1021]
    ]
  ]
],
[
  "NAME:DocTraces",
  [
    "NAME:DLMetrics",
    [
      "User Defined", "", "DDR3 AC-Timing 4-DQ1",
      [
                "Context:=" , ""
      ],
      [
                "Index:=" , ["All"], "Trise:=" , ["Nominal"],
        "Tfall:=" , ["Nominal"], "Pulse_Width:=" ,
                ["Nominal"], "Data_Rate:=" , ["Nominal"],
        "Length:=" , ["Nominal"]
      ],
      [
                "Probe Component:=" , [""]
      ],
      []
    ]
  ],
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
  [
    "NAME:DQ0",
    [
      "Standard", "DQ0", "NexximTransient",
      [
                  "NAME:Context", "SimValueContext:=" ,
                  [1,0,2,0,False,False,-
        1,1,0,1,1,"",0,0,"DE",False,"0","DP",False,"20000000","DT",F
        alse,"0.001","WE",False,"100ns","WM",False,"100ns","WN",Fals
        e,"0ps","WS",False,"0ps"]
      ],
    [
      "Time:=" , ["All"],"Trise:=" , ["Nominal"],"Tfall:=" ,
      ["Nominal"],"Pulse_Width:=" ,["Nominal"],
      "Data_Rate:=" , ["Nominal"], "Length:=" , ["Nominal"]
    ],
    [
      "Probe Component:=" , ["DQ0"]
    ],
    []
  ]
  ],
])
```

Sample Script

This script demonstrates how to add, edit, rename and delete a document.

```
Set oModule = oDesign.GetModule("UserDefinedDocuments")
```

Add a UDD

```
oModule.AddDocument(
["NAME:Test Report1", "Test Report",
"SysLib","Examples/TestUDDInputs",
["NAME:Inputs", ["NAME:DLMetrics", "Solution","Data Line Metrics", -
1, -1], ["NAME:DQ0", "Trace", "DQ0", -1, -1], ["NAME:DQS", "Trace",
"DQS", -1, -1],
["NAME:Name", "Text", "User Name", ["Suzy Q Public"]],
["NAME:Summary", "Bool", "Display Summary", [true]],
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
["NAME:Version", "Number", "Script Version"]]],
["NAME:DocTraces", ["NAME:DLMetrics", ["User Defined", "", "DDR3 AC-
Timing 4-DQ1", ["Context:=", ""], ["Index:=", ["All"], "Trise:=",
["Nominal"], "Tfall:=", ["Nominal"], "Pulse_Width:=", ["Nominal"],
"Data_Rate:=", ["Nominal"], "Length:=",["Nominal"]],
["Probe Component:=", [""]], []
]
],
["NAME:DQ0", ["Standard", "DQ0", "NexximTransient", ["NAME:Context",
"SimValueContext:=", [1, 0, 2, 0, false, false, -1, 1, 0, 1, 1, "",
0, 0, "DE", false, "0", "DP", false, "20000000", "DT", false,
"0.001", "WE", false, "100ns", "WM", false, "100ns", "WN", false,
"0ps", "WS", false, "0ps"]],
["Time:=", ["All"], "Trise:=", ["Nominal"], "Tfall:=", ["Nominal"],
"Pulse_Width:=", ["Nominal"], "Data_Rate:=", ["Nominal"], "Length:=",
["Nominal"]],
["Probe Component:=", ["DQ0"]], []
]
])
```

Edit Document

```
oModule.EditDocument("Test Report1",
["NAME:Test Report", "Test Report","SysLib",
"Examples/TestUDDInputs",
["NAME:Inputs",
["NAME:DLMetrics","Solution", "Data Line Metrics", 1000001, 0],
["NAME:DQ0", "Trace", "DQ0", 32, 2],
["NAME:DQS", "Trace", "DQS", 32, 4],
["NAME:Name", "Text", "User Name", ["Suzy Q Public"]],
["NAME:Summary", "Bool", "Display Summary", [true]],
["NAME:Version", "Number", "Script Version"]
]
],
["NAME:DocTraces",
["NAME:DLMetrics", ["User Defined", "Solution", "DDR3 AC-Timing 4-
DQ1", ["Context:=", ""], ["Index:=", ["All"], "Trise:=", ["Nominal"],
"Tfall:=", ["Nominal"], "Pulse_Width:=", ["Nominal"], "Data_Rate:=",
["Nominal"], "Length:=", ["Nominal"]],
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
["Probe Component:=", [""]], []
]
],
["NAME:DQ0", ["Standard", "DQ1", "NexximTransient", ["NAME:Context",
"SimValueContext:=", [1, 0, 2, 0, false, false, -1, 1, 0, 1, 1, "",
0, 0, "DE", false, "0", "DP",false, "20000000", "DT", false, "0.001",
"WE", false, "100ns", "WM", false, "100ns", "WN", false, "0ps", "WS",
false, "0ps"]], ["Time:=", ["All"], "Trise:=", ["Nominal"],
"Tfall:=", ["Nominal"], "Pulse_Width:=", ["Nominal"], "Data_Rate:=",
["Nominal"], "Length:=", ["Nominal"]], ["Probe Component:=",
["DQ1"]], []
]
])
```

Rename a UDD

```
oModule.RenameDocument("Test Report", "Test UDD Report")
```

Update UDD

```
oModule.UpdateDocument ("Test UDD Report"}
```

View Html

```
oModule.ViewHtmlDocument("Test UDD Report")
```

View Pdf

```
oModule.ViewPdfDocument(Test UDD Report")
```

Save Html

```
oModule.SaveHtmlDocumentAs("Test UDD Report",
"c:/AnsysProjects/Test.html")
```

Save pdf

```
oModule.SavePdfDocumentAs "Test UDD Report",
"c:/AnsysProjects/Test.pdf"
```

Delete UDD

```
oModule.DeleteDocument("Test UDD Report")
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** The product has to implement the `GetModule` call to create the `UserDefinedDocument` scripting object. As an example, see AltraSimDesign.cpp (function `GetMgrIDispatch()`).

Document Generator Interfaces

This document briefly describes the API interfaces available in the document generator plugin.

(Ansys.Ansoft.DocGeneratorPluginDotNet.dll)

Scripting objects available in the script for the Generate function:

- oApp = self.GetUDDAppContext()

  Gets the application context. Use this interface to get the active project ad the version of the product.

  ```
  oDesktop = oApp.GetAppDesktop()
    if oDesktop != None:
    vr = oDesktop.GetVersion()
    oProject = oDesktop.GetActiveProject()
  ```

- oDesign = self.GetUDDDesignContext()

  Gets the design context. Use this interface to get the design name.

  ```
  oDesign = self.GetUDDDesignContext()
    if oDesign != None:
      nm = oDesign.GetName()
  ```

- IUDDGenerator interface

  This interface available in the Generate method of the UDDPluginExtension. This interface can be used to:

  1. Set the document output file path.
     ```
     docgen.SetOutput("C:\\Examples\\DocumentOutput.xml")
     ```
  2. Get the document root.
     ```
     docroot = docgen.GetDocumentRoot()
     ```
  3. Write out to the output file.
     ```
     docgen.Write()
     ```
  4. Write HTML document
     ```
     void WriteHTML();
     ```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Write PDF document
   ```
   void WritePDF();
   ```
6. Load the HTML transform object
   ```
   void LoadHTMLTransform();
   ```
7. Load the cached PDF transform object
   ```
   void LoadPDFTransform();
   ```
8. Get script path
   ```
   string GetScriptPath();
   ```
9. Get output file path
   ```
   string GetOutputFilePath();
   ```

- IUDDRoot interface

  Calling GetDocumentRoot() on the IUDDGenerator interface provides you with the this interface. This interface can be used to:

  1. Add a new section to the document. Provide a section title.
     ```
     section1 = docroot.AddSection("Section title")
     ```
  2. Add a new section to the document. Provide a section title and subtitle.
     ```
     section1 = docroot.AddSection("Section title", "Section subtitle")
     ```
  3. Add a new title
     ```
     section1 = docroot.AddTitle("Title")
     ```
  4. Add a new subtitle
     ```
     section1 = docroot.AddSubtitle("Subtitle")
     ```

- IUDDSection interface

  Calling AddSection() on the IUDDRoot interface provides you with this interface. This interface can be used to:

  1. Set an ID for the section for internal links.
     ```
     section1.SetID("id")
     ```
  2. Add a new table to the document. Provide a table title.
     ```
     table1 = section1.AddTable("Table title")
     ```
  3. Add a new image to the document. Provide an image title and a file path to the image file.
     ```
     image1 = section1.AddImage("Image title")
     ```
  4. Add text to the document.
     ```
     text1 = section1.AddText("Random text........")
     ```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- IUDDImage interface

  Calling AddImage() on the IUDDSection interface provides you with this interface. In this interface, you can call the following methods:

  1. Set an ID for the image for internal links.
     ```
     image1.SetID("id")
     ```
  2. Set alignment information; can be "center", "left", and "right".
     ```
     image1.SetAligment("center")
     ```
  3. Set the file path of the image file; not necessary if image file path is set through the AddImage() method.
     ```
     image1.SetFileRef("Image path")
     ```
  4. Set the format of the image file; can be "BMP", "PNG", "JPEG", "JPG", "DVI", etc.
     ```
     image1.SetFormat("format")
     ```

- IUDDText interface

  Calling AddText() on the IUDDSection interface provides you with the this interface. In this interface, you can call the following methods:

  1. Set an ID for the text for internal links.
     ```
     text1.SetID("id")
     ```
  2. Set the emphasis attribute on the text.
     ```
     text1.SetEmphasis()
     ```
  3. Set the quotes attribute on the text.
     ```
     Text1.SetQuotes()
     ```
  4. Set the block quotes attribute on the text.
     ```
     text1.SetBlockquotes()
     ```
  5. Set quotes on the text.
     ```
     Text1.SetQuotes()
     ```
  6. Set the wordsize attribute on the text
     ```
     text1.SetSize(size as an integer)
     ```
  7. Set a link to an ID of any element to provide internal links
     ```
     text1.SetLink("linkname")
     ```
  8. Set an event link to handle an event. The HandleUDDEvents method should be implemented in the script to handle the event.
     ```
     text1.SetEventLink("linkname")
     ```

- IUDDTable interface

  Calling AddTable() on the IUDDSection interface provides you with the this interface. In this interface, you can call the following methods:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Set an ID for the table for internal links.
   ```
   table1.SetID("id")
   ```
2. Set alignment information; can be "center", "left", and "right".
   ```
   table1.SetAlignment("center")
   ```
3. Set the background color of the table
   ```
   table1.SetBgColor(string bgcolor)
   ```
4. Set the frame type; can be "all", "bottom", "top", "sides", and "topbot".
   ```
   table1.SetFrame(string frame)
   ```
5. Add a table group and specify the number of columns. A table can have multiple table groups.
   ```
   IUDDTableGroup table1.SetTableGroup(int columns)
   ```

- IUDDTableGroup interface

  Calling AddTableGroup() on the IUDDTable interface provides you with the this interface. In this interface, you can call the following methods:

  1. Set an ID for the table group for internal links.
     ```
     tgroup1.SetID("id")
     ```
  2. Set alignment information; can be "center", "left", and "right".
     ```
     tgroup1.SetAlignment("center")
     ```
  3. Set the column width of a column given the index of the column and the required width. Width can be set in 2 ways.
     - Width can be set relative to 1. E.g Setting it to "2*" makes the column width double the width of the others.
     - If the entire table width is considered to be 99.99 units. Width can be a number relative to this.

     ```
     tgroup1.SetColumnWidth(int index, string width)
     ```
  4. Add a header to the table group
     ```
     IUDDTableRow tgroup1.AddHeader()
     ```
  5. Add a header with multiple rows to the table group; it takes number of sub rows.
     ```
     IUDDTableRow tgroup1.AddHeader(int rows)
     ```
  6. Add a row of content to the table group
     ```
     IUDDTableRow tgroup1.AddContent()
     ```
  7. Add content with multiple rows to the table group; it takes number of sub rows.
     ```
     IUDDTableRow tgroup1.AddContent(int rows)
     ```

- IUDDTableRow interface

  Calling AddHeader() & AddContent() on the IUDDTableGroup interface provides you with the this interface. In this interface, you can call the following methods:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Set an ID for the table row for internal links.
   ```
   trow1.SetID("id")
   ```
2. Set alignment information; can be "center", "left", and "right".
   ```
   trow1.SetAlignment("center")
   ```
3. Set cell text; can be cell content or header text. It takes a column index and a text string. It is added to the first row.
   ```
   IUDDTextElement trow1.Add(int column, string text)
   ```
4. Set cell text; can be cell content or header text. It takes a column index, row index and a text string and takes in a row number because a table row can have multiple sub rows.
   ```
   IUDDTextElement trow1.Add(int column, int subrow, string
   text)
   ```
5. Set cell content; takes a column index and an int value. It is added to the first row.
   ```
   IUDDTextElement trow1.Add(int column, int value)
   ```
6. Set cell content; takes a column index, row index and a int value.
   ```
   IUDDTextElement trow1.Add(int column, int subrow, string
   text)
   ```
7. Set cell text; takes a column index and a double value. It is added to the first row.
   ```
   IUDDTextElement trow1.Add(int column, double value)
   ```
8. Set cell text; takes a column index, row index and a double value.
   ```
   IUDDTextElement trow1.Add(int column, int subrow, double
   value)
   ```
9. Set cell text spanning two columns; can be cell content or header text. It takes a sub-row index , starting column index, ending column index, and a text string.
   ```
   IUDDTextElement trow1.AddSpanningcolumnst(int subrow, int
   columnstart, int columnend, string text)
   ```
10. Set cell text; can be cell content or header text. It takes a column index, starting sub-row index, ending sub-row index, and a text string.
    ```
    IUDDTextElement trow1.AddpanningRows(int column, int
    subrowstart, int subrowend, string text)
    ```

Includes all the methods exposed by the IUDDText interface.

Chip Model Analyzer (CMA)

Ansys Chip Model Analyzer (CMA) is an advanced Chip Power Model (CPM) creation tool for performance checking and failure diagnosis of the Power Delivery Network (PDN). CMA can use either HSpice or Nexxim simulators. HSpice and external Nexxim require licenses, but the Nexxim Internal simulator does not.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CMA does not currently interact with Electronics Desktop. However, you can launch CMA from within Electronics Desktop by clicking **Tools** > **Chip Model Analyzer (CMA)**.

From there, you can import CPM files for analysis or create a pseudo CPM.

For more information about using CMA, consult the CMA training documentation by launching CMA and selecting **Help** > **Content**.

PinToPinUtility

The PinToPin utility can quickly extract HFSS or SIwave models for specified nets of package and PCB geometries. It establishes a repeatable process for robust geometry extraction, port configuration, and passive model assignment. Process execution can be done via the user interface (Windows only) or in non-graphical mode (Windows or Linux).

For more information, consult the SIwave and HFSS 3D Layout help.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

25 - Optimetrics

Optimetrics enables you to determine the best design variation among a model's possible variations. You create the original model, the *nominal design*, and then define the design parameters that vary, which can be nearly any design parameter assigned a numeric value in Maxwell. For example, you can parameterize the model geometry or material properties. You can then perform the following types of analyses on your nominal design:

optiSLang	optiSLang simulations can be integrated with Ansys Electronics Desktop, such that you can create a setup very much like an Optimetrics setup to run through an optiSLang installation.
Parametric	In a parametric analysis, you define one or more *variable sweep definitions,* each specifying a series of variable values within a range. For example, you can parameterize component values. (See Variables for more information.) Optimetrics solves the design at each variation. You can then compare the results to determine how each design variation affects the performance of the design. Parametric analyses are often used as precursors to optimization solutions because they help to determine a reasonable range of variable values for the optimization analysis.
Design of Experiments	Design of Experiments (DOE) is a technique used to scientifically determine the location of sampling points and is included as part of the Response Surface, Goal Driven Optimization, and Analysis systems.
DesignXplorer	An optimization tool for studying a range of design variations, used with Design of Experiments. This permits you to manage an Optimetrics simulation from the Ansys Workbench.
Optimization	For an optimization analysis, you identify the cost function and the optimization goal. Optimetrics changes the design parameter values to meet that goal. The cost function can be based on any solution quantity that can be computed.
Sensitivity	In a sensitivity analysis, you use Optimetrics to explore the vicinity of the design point to determine the sensitivity of the design to small changes in variables.
Tuning	Tuning allows you to change variable values interactively while monitoring the performance of the design. If you want to ensure that tuning does not resolve variations already solved by parametric setup, you must check **Save Fields Mesh** in the **Options** tab of the Optimetrics setup.
Statistical	In a statistical analysis, you use Optimetrics to determine the distribution of a design's performance, which is caused by a statistical distribution of variable values.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Sensitivity, Statistical, Design Xplorer, and some Optimizers have been placed in legacy mode because they are no longer under active development. Go to **Tools** > **Options** > **General Options** > **Miscellaneous** and toggle the **Enable Legacy Optimetrics Tools** check box to show or hide these legacy tools.

Note:

Sweeping or using a complex variable is not allowed in any optimetrics setup, including optimization, statistical, sensitivity, and tuning setups.

The HPC and Analysis Options dialog can be accessed from the setup dialog for each type of Optimetrics analysis.

Parametric Overview

Running a parametric analysis enables you to simulate several design variations using a single model. You define a series of variable values within a range, or a variable sweep definition, and Maxwell generates a solution for each design variation. You can then compare the results to determine how each design variation affects the performance of the design.

You can vary design parameters that are assigned a quantity, such as geometry dimensions, material properties, and boundary and excitation properties (See the help topic for the specific parameter you want to vary). The number of variations that can be defined in a parametric sweep setup is limited only by your computing resources.

To perform a parametric analysis, you first create a nominal design. A nominal design is created like any other design, except that variables are assigned to those aspects of the model you want to change. You can create a parametric setup before defining variables but all variables must be defined before you start the parametric analysis. Although you are not required to solve the nominal design before performing a parametric analysis, doing so helps ensure that the model is set up and operates as intended. Alternatively, you can perform a validation check on the nominal design before performing a parametric analysis.

Parametric analyses are often used as precursors to optimization analyses because they enable you to determine a reasonable range of variable values within which optimal conditions will occur.

Setting Up a Parametric Analysis

A *parametric setup* specifies all of the design variations that Optimetrics drives the software to solve. A parametric setup is made up of one or more variable sweep definitions, which are a set

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

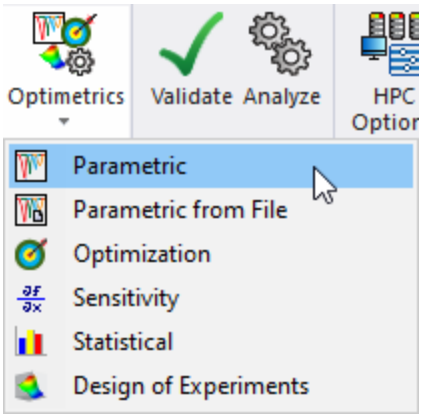
of variable values within a range that you want the software to solve when you run the parametric setup.

You can define more than one parametric setup per design.

> **Note:**
>
> Once you have created a parametric setup, you can copy and paste it, and then make changes to the copy, rather than redoing the whole process for minor changes.

To add a parametric setup to a design:

1. Click **Maxwell** > **Optimetrics Analysis** > **Add Parametric** .
   - Alternatively, right-click **Optimetrics** in the Project Manager and then click **Add** > **Parametric** on the shortcut menu.
   - Select the **Simulation** tab in the ribbon, and select **Parametric** from the drop-down menu under the Optimetrics icon:



   The **Setup Sweep Analysis** dialog box appears.

2. Add a variable sweep definition.

   After you define a parametric sweep, a shortcut menu becomes available when you right-click the setup name in the Project Manager.

> **Note:**
>
> Sweeping or using a complex variable is not allowed in any optimetrics setup, including optimization, statistical, sensitivity, and tuning setups.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: If you are performing an Optimetrics\parametric analysis of a Maxwell 2D transient design that uses expression cache, cut down on simulation time by using the HPC settings **Optimetrics Variations** and **Single level only**. For more information on setting up a Maxwell 2D parametric transient design that uses expression cache, see Defining Expressions for Transient Solutions.

Adding a Variable Sweep Definition

A parametric setup is made up of one or more *variable sweep definitions*. A variable sweep definition is a set of variable values within a range that Optimetrics drives to solve when the parametric setup is analyzed. You can add one or more sweep definitions to a parametric setup.

Note:

Sweeping a complex variable is not allowed in any optimetrics setup, including optimization, statistical, sensitivity, and tuning setups.

1. Click **Maxwell** > **Optimetrics Analysis** > **Add Parametric** .
   - Alternatively, right-click **Optimetrics** in the project tree, and then click **Add** > **Parametric** on the shortcut menu.

   The **Setup Sweep Analysis** dialog box appears.
2. Under the **Sweep Definitions** tab, click **Add**.

   The **Add/Edit Sweep** dialog box appears.

   All the independent variables associated with the design are listed in the **Variable** drop-down menu of the **Add/Edit Sweep** dialog.
3. Click the variable for which you are defining the sweep definition from the **Variable** drop-down menu.

   If you do not define a sweep definition for a variable in the list, the variable's current value in the nominal design is used in the parametric analysis.
4. Specify the variable values to be included in the sweep.
5. Click **Add** and then click **OK**.

   You return to the **Setup Sweep Analysis** dialog box. The variable sweep is listed in the top half of the window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. View the design variations that are to be solved in table format under the **Table** tab. Viewing the sweep definition in table format enables you to visualize the design variations that are to be solved and manually adjust sweep points if necessary.
7. Click **OK**.

Specifying Variable Values for a Sweep Definition

To specify the variable values to include in a sweep definition:

1. Select one of the following in the **Add/Edit Sweep** dialog box:

Single value	Specify a single value for the sweep definition.
Linear step	Specify a linear range of values with a constant step size.
Linear count	Specify a linear range of values and the number, or count of points within this range.
Decade count	Specify a logarithmic (base 10) series of values, and the number of values to calculate in each decade.
Octave count	Specify a logarithmic (base 2) series of values, and the number of values to calculate in each octave.
Exponential count	Specify an exponential (base e) series of values, and the number of values to calculate.

2. If you selected **Single value**, type the value of the sweep definition in the **Value** box.

   If you selected another sweep type, do the following:

   a. Type the starting value of the variable range in the **Start** text box.
   b. Type the final value of the variable range in the **Stop** text box.

   > **Warning:**
   >
   > Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any optimetric analysis.

3. If you selected **Linear step** as the sweep type, type the step size in the **Step** box.

   The step size is the difference between variable values in the sweep definition. The step size determines the number of design variations between the start and stop value. The model is solved at each step in the specified range, including the start and stop values. The step size can be negative, when the **Stop** value is less than the **Start** value

   If you selected another sweep type, type the number of points, or variable values, in the sweep definition in the **Count** text box. For **Decade count** and **Octave count**, the **Count**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

value specifies the number of points to calculate in every decade or octave. For **Exponential count**, the **Count** value is the total number of points. The total number of points includes the start and stop values.

Synchronizing Variable Sweep Definitions

By default, variable sweep definitions are nested. Alternatively, you can synchronize the variable sweep definitions if they have the same number of sweep points.

For example, if you synchronize a sweep definition that includes values of 1, 2, and 3 inches with a second sweep definition that includes values of 4, 5, and 6 inches, 3 design variations are solved. The first variation is solved at the variable values of 1 and 4; the second variation is solved at the variable values 2 and 5; and the third variation is solved at the final variable values 3 and 6.

To synchronize variable sweep definitions:

1. Under the **Sweep Definitions** tab of the **Setup Sweep Analysis** dialog box, select the rows containing the sweep definitions you want to synchronize.
2. Click **Sync**.

   The synchronized sweeps are given a group number, which is listed in the **Sync #** column.

   Optionally, view the design variations that are to be solved in table format under the **Table** tab.

Modifying a Variable Sweep Definition Manually

You can manually modify the variable values that are solved for a parametric setup by explicitly changing, adding, or deleting existing points in a variable sweep definition under the **Table** tab of the **Setup Sweep Analysis** dialog box.

To manually modify a variable sweep definition:

1. Click the **Table** tab of the **Setup Sweep Analysis** dialog box.

   The design variations that will be solved for the parametric setup are listed in table format.

2. Do one of the following:
   - To modify a variable value, click a value text box in the table and type a new value.
   - To delete a variable value from the sweep definition, click the row you want to delete, and then click **Delete**.
   - To add a new variable value to the sweep definition, click **Add**. Then click in the value text box and type a new value.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Warning:**
>
> Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any optimetric analysis.

Your modifications are tracked and available for viewing at the bottom of the **Setup Sweep Analysis** dialog box under the **Sweep Definitions** tab. The operations you performed are listed with descriptions.

> **Warning:**
>
> If you modify an original sweep definition using the **Add/Edit Sweep** dialog box after you have manually modified its table of design variations, your manual modifications become invalid and are removed. A warning is displayed to inform you that your manual values are about to become invalid, so you can decide whether or not to proceed.

Overriding a Variable's Current Value in a Parametric Setup

If you choose not to sweep a variable, the variable's current value set for the nominal design is used when it solves the parametric setup. To override the current variable value for a parametric setup:

1. In the **Setup Sweep Analysis** dialog box, click the **General** tab.

   Under **Starting Point**, all of the current independent design variable values are listed.

2. Click the **Override** box of the design variable with the value you want to override for the parametric setup.
3. Type a new value in the **Value** box, and then press **Enter**.

   The **Override** option is now selected. This indicates that the value you entered will be used for the parametric setup. For this parametric setup, the new value will override the current value in the nominal design.

   > **Note:**
   >
   > Alternatively, you can select the **Override** option first, and then type a new variable value in the **Value** box.

4. Optionally, click a new unit in the **Units** box.

To revert to the current variable value, clear the **Override** option.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Warning:**
>
> Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any optimetric analysis.

Specifying a Solution Setup for a Parametric Setup

To specify the solution setup that the software analyzes when it solves a parametric setup:

1. In the **Setup Sweep Analysis** dialog box, click the **General** tab.
2. Select the solution setup you want the software to use when it solves the parametric setup.

   The parametric setup is solved using the solution setup you select. If you select more than one, results are generated for all selected solution setups.

Specifying the Solution Quantity to Evaluate for Parametric Analysis

When you add a parametric setup, you can identify one or more solution quantities to be presented in the *Post Analysis Display* dialog box. The solution quantities are specified by mathematical expressions that are composed of basic quantities, such as output variables. When you view the results, Maxwell extracts the solution quantities and lists them in the results table.

1. In the *Setup Sweep Analysis* dialog box, click the **Calculations** tab.

   This displays a table that will show Solutions and associated Calculations. Below the table, are control buttons to **Setup Calculations...** and **Delete**.

2. Click **Setup Calculations**.

   This displays the *Add/Edit Calculation* dialog box. The dialog contains panes to set the **Context**, the **Trace** tab for the **Calculation Expression**, and the **Calculation Range** tab for the **Calculation Range**.

   Follow the procedure to Setup Calculations for Optimetrics.

3. Click **Add Calculation** to add the expression in the *Add/Edit Calculation* dialog box's **Calculation Expression** field to the Calculations tab of the *Setup Sweep Analysis* dialog box.
4. Click **Done** to close the *Add/Edit Calculation* dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting up Calculations for Optimetrics

The **Setup** dialog boxes for each of the Optimetrics types include a **Setup Calculations** button. Clicking this button displays the **Add/Edit Calculation** dialog box, which contains distinct panes and tabs to set the **Context**, the **Calculation Expression**, and the **Calculation Range**.

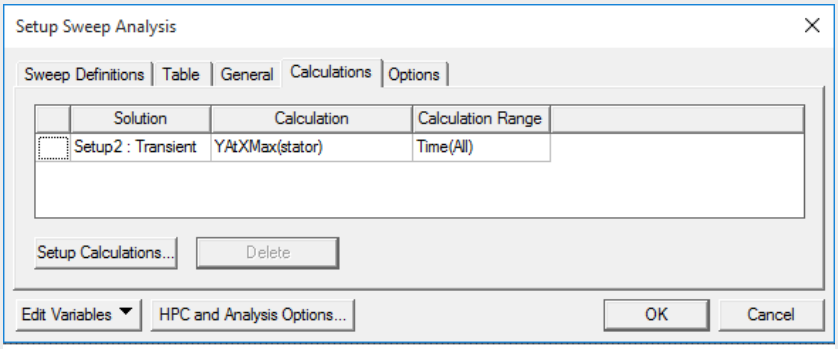
The **Context** pane contains fields for the Report Type to use, the Solution, and depending on the Report Type selection, the Geometry.

The **Trace** tab contains fields for the Calculation expression and, to build the expression, a Category list, a Quantity list with a Text Filter field, and a list of Functions available for the selected Category. The Range function button opens a dialog box in which you can define a range function to apply a function to the expression.

The Category list for the **Trace** tab includes Variables and Output Variables. An Output Variables... button lets you open a dialog box to define and edit the Output Variables.

Note:

When specifying a field calculation for a solution that is defined with varying save field, select "All" time points as the Calculation Range and apply a range function to the Calculation, as shown in the parametric setup example below:



A field calculation with geometry context is not supported for a solution that is defined with varying save field.

To set up an Optimetrics calculation:

1. Click **Setup Calculations** to open the **Add/Edit/Calculation** dialog box.
2. In the **Report Type** text field in the **Context** pane, select from the drop-down menu of available types.

   Selecting Fields as the Report type causes the **Geometry** field to display.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In the **Solution** text box, select from the drop-down menu of available solutions.
4. If the **Geometry** field is available, select from the drop-down menu.
5. In the **Trace** tab, specify the solution Category, a Quantity, and Functions. The resulting expression will be displayed in the *Calculation Expression* field.
    a. Select the **Category** from the list.

       The selection appears in the **Calculation Expression** field, and the Quantity and Function fields list what is available for the corresponding selection.

    b. Select the **Quantity** from the list.

       The selected quantity appears in the **Calculation Expression** field.

       If the **Quantity** list is long, you can filter it for easier selection by typing in the text filter field. Only quantities that contain those alphanumeric characters anywhere in their name will remain visible in the list.

       If you want to create an output variable that represents the solution quantity, do the following:

       i. Click **Output Variables**.

          The **Output Variables** dialog box appears.

       ii. Add the expression you want to evaluate, and then click **Done**.

          The recently created output variable appears in the **Quantity** list.

       iii. Click a new output variable in the **Quantity** drop-down menu.

       > **Note:**
       >
       > The calculation you specify must be able to be evaluated into a single, real number.

       The selected Quantity appears in the **Calculation Expression** field.

    c. Select the **Function** from the list.

       The selected function is applied to the **Quantity** in the **Calculation Expression** field.
6. To apply a **Range function** to the **Calculation Expression**, see Setting a Range function.
7. Click **Add Calculation** to add the expression in the **Add/Edit Calculation** dialog box's **Calculation Expression** field to the Calculations tab of the **Setup Sweep Analysis** dialog box.
8. Click **Done** to close the **Add/Edit Calculation** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specifying a Solution Quantity's Calculation Range

The calculation range of a solution quantity determines the value of intrinsic variables such as frequency (F) at which the solution quantity will be extracted. For a parametric setup, the calculation range must be a single value. For a Driven Modal or Driven Terminal design, if you selected to extract the solution from the last adaptive solution, Optimetrics uses the adaptive frequency defined in the solution setup. If you selected to extract the solution quantity from a frequency sweep solution, Optimetrics by default will use the starting frequency in the sweep.

1. In the **Setup Sweep Analysis** dialog box, click the **Calculations** tab.
2. Click the **Setup Calculations** button.

   The **Add/Edit/Calculation** dialog box appears.

3. Select the **Calculation Range** tab.
4. In the **Variable** list, click an intrinsic variable.

   Depending on the variable and Report Type, either a single value or All appears in the Value field.

5. In the **Value** box, click the ellipsis button to display a dialog box that lets you select particular values, or specify a range of values (for Time sweeps). By default, **Use all values** is selected.
   - For variables other than Time, unchecking **Use all values** enables a list of available values. You can use the buttons to **Clear All** or **Select All** values.
     a. Select the **SweepDefault** or **Edited** radio buttons to specify whether to accept the default or edited sweeps.
     b. To edit the sweeps further, select the ellipsis [**...**] button to display an **Edit Sweep** dialog box. For frequency variables, this lets you specify a single value, linear step, linear count, decade count, octave count, or exponential count. You can **Add** legal values to the list of sweep values, **Update** the list for changes, or **Delete** selected entries.
     c. When finished selecting/editing sweep values, close the dialog box. The values you select are displayed in the Value field.
   - For Time sweeps:
     a. Select the **Specify range** radio button to enable the **Min** and **Max** value fields.
     b. Enter the desired **Min** and **Max** values, and choose the associated units of measure from the drop down lists.
     c. When finished, close the dialog box. The values you selected are displayed in the Value field.
6. Click **Update**, and then click **Edit**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using Distributed Analysis

If you have purchased the appropriate license, the Electronics Desktop supports distributed solve, which involves distributing rows of a parametric table during Optimetrics solve.

If you do a distributed solve, the Electronics Desktop launches solver engines on multiple machines, assuming that you have configured your HPC and Analysis Options correctly. Also see Large Scale DSO for Parametric Analysis.

To run a distributed analysis:

1. Under **Optimetrics** in the project tree, right-click the specific parametric setup.

   A shortcut menu appears.

2. Select **Analyze** from the shortcut menu.

   > **Note:**
   >
   > After you define a parametric sweep, a shortcut menu becomes available when you right-click the setup name.

   While the analysis is running, you can access parent and child progress bars. By default, only the main progress bar is displayed, while the child progress bars (or subtasks) remain hidden. You can toggle between showing and hiding the child progress bars.

   To show the child progress bars:

   - Right-click the progress window, and select **Show Subtask Progress Bars**.

     To hide the child progress bars:

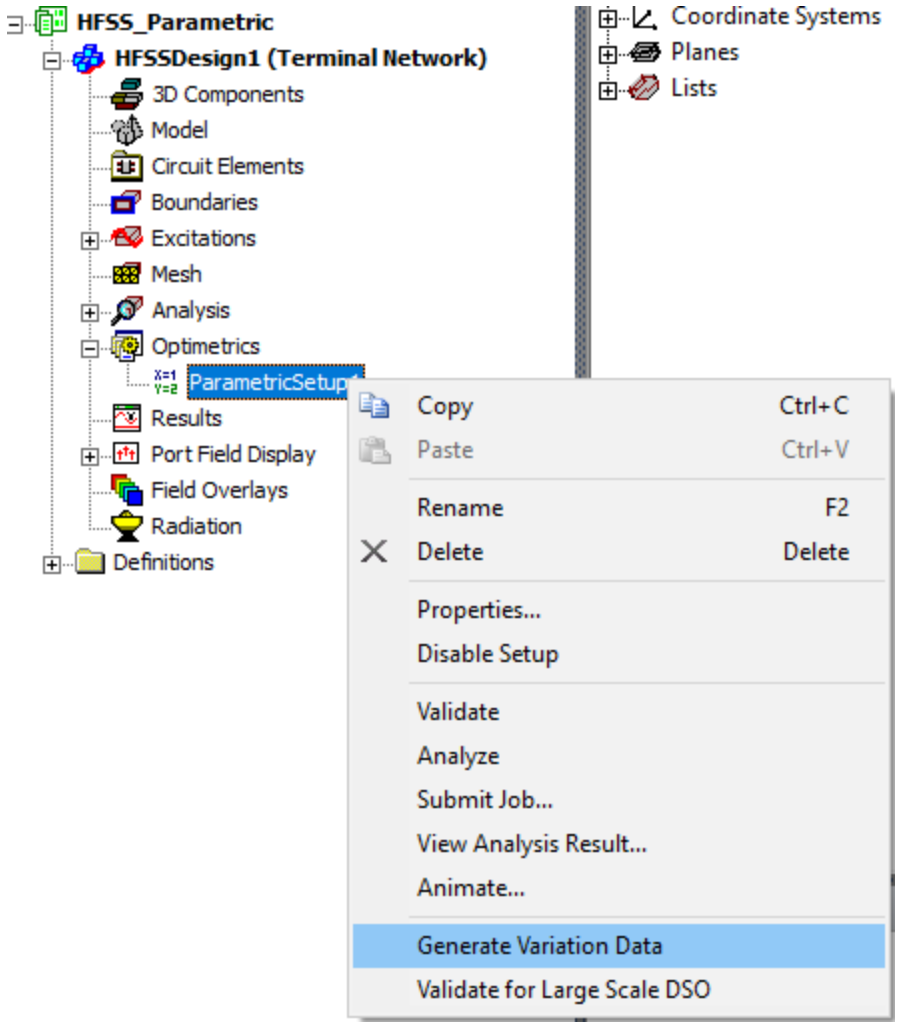
   - Right-click the progress window, and select **Hide Subtask Progress Bars**.

Solving a Parametric Analysis on Another Machine

If you would like to run a parametric analysis on another machine, for example, to run a Discovery-linked model on Linux, you can generate the variation data and then export an archive of the project.

To generate the variation data, right-click on the **ParametricSetup** entry in the Project Manager tree, and select **Generate Variation Data**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



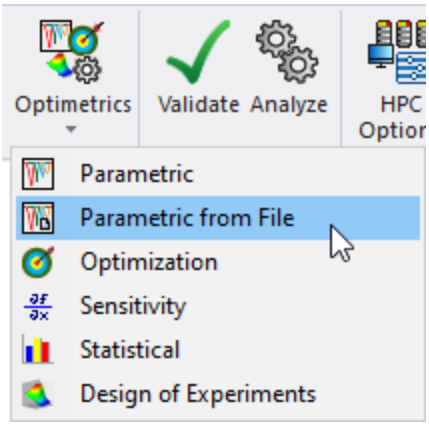
When this option is selected, Maxwell will generate geometry for all variations and cache it in the results folder. Users can archive all the data, and then solve on other machines.

> **Note:** If the parametric analysis uses a Discovery-linked model, the machine that the archive is moved to does not have to have Discovery installed.

Adding a Parametric Sweep from a File

You can specify the parameters for a parametric sweep in a spreadsheet that uses either a .csv (comma delimited) or .txt (tab delimited) format. You can then import the parametric sweep using the **Maxwell** > **Optimetrics Analysis** > **Add Parametric from File** command, or with the **Simulation** tab selected, use the drop-down menu under the Optimetrics icon and select **Parametric from File**:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



These methods open a file browser for a comma delimited file (.csv) or a tab delimited .txt file.

For example, a .txt spreadsheet file could resemble the following:

```
a $b $c[in] d[m] $e $f
0.1 mil 2mm 11 21 0.6in 8
0.2mil 3mm 1.3 2.6mm 3 9cm
...
```

The first row lists the Project and Design Variable names, and when followed by parentheses, the units. When unit not present, SI unit is assumed. The following rows provide the variable values and units, where SI is assumed if not specified. Project or Design variables must be defined before they are accepted from a file. The characters in variable names are not case sensitive. Consecutive separators are treated as one separator.

The header row also takes units in ( ) as well as the conventional [ ].

After you have imported a valid file, the detailed information regarding sweep values can be found at the lower portion of the Parametric **Setup Sweep Analysis** dialog**Sweep Definitions** tab (Operation/Description panel). It is treated as: first row as single point. then each time a new sweep point is added.

Viewing Results for Parametric Solution Quantities

1. In the project tree, right-click the parametric setup for which you want to view the results calculated for the solution quantities, and then click **View Analysis Result** on the shortcut menu.

   The **Post Analysis Display** dialog box appears.

2. Select the parametric setup with the results you want to view from the drop-down menu at the top of the dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. If it is not already selected, select **Table** as the view type.

   The results for the selected solution quantities are listed in table format for each solved design variation. The variation column in the table lists the entries in order. Clicking the Vision header inverts the order. Clicking other headers sorts the entries by value, and clicking again inverts the order.

4. Optionally, select **Show complete output name**.

   The complete name of the solution for which the results are being displayed will be listed in the column headings.

5. Optionally, click a design variation in the table, and then click **Apply** (at the far right side of the dialog box).

   The design displayed in the **Modeler** window is changed to represent the selected design variation.

Plotting Solution Quantity Results vs. a Swept Variable

To plot solution quantity results versus a swept variable's values on a rectangular (x - y) plot:

1. In the project tree, right-click the parametric setup for which you want to view the results, and select **View Analysis Result**.

   The **Post Analysis Display** dialog box appears.

2. If it is not already selected, select **Plot** as the view type.
3. Select the variable with the swept values you want to plot on the x-axis from the **X** drop-down menu.
4. Only one sweep variable at a time can be plotted against solution quantity results. Any other variables that were swept during the parametric analysis remain constant.

   Optionally, to modify the constant values of other swept variables, do the following:

   a. Click **Set Other Sweep Variables Value**.

      The **Setup Plot** dialog box appears. All of the other solved variable values are listed.

   b. Click the row with the variable value you want to use as the constant value in the plot, and then click **OK**.

5. Select the solution quantity results you want to plot on the y-axis from the **Y** drop-down menu.

   The xy plot appears in the view window.

6. Right-click in the plot area to get the shortcut menu where you can set modify the plots display properties, print, copy to the clipboard, or export the data to a file.

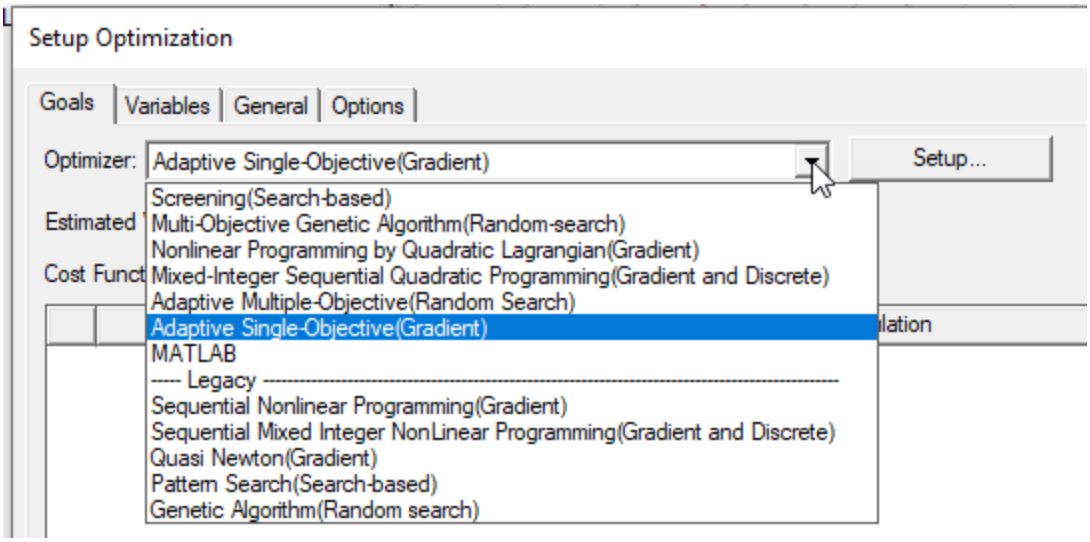
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Optimization Overview

Optimetrics interfaces with Ansys Electromagnetics products to help you optimize a wide variety of design parameters based on variable geometry, materials, excitations, component values, etc. Optimization is the process of locating the minimum of a user-defined cost function. Optimetrics modifies the variable values until the minimum is reached with acceptable accuracy.

Choosing an Optimizer

Conducting an optimization analysis allows you to determine an optimum solution for your problem. In optimization analyses, there are many available optimizers.



Goal Driven Optimizers

These use a Decision Support Process (DSP) based on satisfying criteria as applied to the parameter attributes using a weighted aggregate method. In effect, the DSP can be viewed as a post-processing action on the Pareto fronts as generated from the results of the various optimization methods.

- Screening (Search based) – a non-iterative direct sampling method that uses a quasi-random number generator based on the Hammersley algorithm. You can start with Screening to locate the multiple tentative optima and then refine with NLPQL or MISQP to zoom in on the individual local maximum or minimum value. Usually Screening is used for preliminary design, which can lead you to apply one of the other approaches for more refined optimization results.
- Multi-Objective Genetic Algorithm – an iterative random search algorithm that can optimize problems with continuous input parameters. It is better for calculating the global

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

optima. You can start with MOGA to locate the multiple tentative optima and then refine with NLPQL or MISQP to zoom in on the individual local maximum or minimum value.

- Nonlinear Programming by Quadratic Lagrangian (Gradient) – a gradient-based, single-objective optimizer based on quasi-Newton methods. Ideally suited for local optimization.
- Mixed-Integer Sequential Quadratic Programming (Gradient and Discrete) – a gradient-based, single-objective optimizer that solves mixed-integer non-linear programming problems by a modified sequential quadratic programming (SQP) method. Ideally suited for local optimization.
- Adaptive Multiple Objective – an iterative, multi-objective optimizer that employs a Kriging response surface and MOGA. In this method, the use of a Kriging response surface allows for a more rapid optimization process because all design points are not evaluated except when necessary and part of the population is simulated by evaluations of the Kriging response surface, which is constructed of all design points submitted by Multi-Objective Genetic Algorithm (MOGA).
- Adaptive Single Objective (Gradient) – a gradient-based, single-objective optimizer that employs an OSF (Optimal Space-Filling) DOE, a Kriging response surface, and MISQP.
- MATLAB

All optimizers assume that the nominal problem you are analyzing is close to the optimal solution; therefore, you must specify a domain that contains the region in which you expect to reach the optimum value.

All optimizers allow you to define a maximum limit to the number of iterations to be executed. This prevents you from consuming your remaining computing resources and allows you to analyze the obtained solutions. From this reduced range, you can further narrow the domain of the problem and regenerate the solutions.

All optimizers also allow you to enter a coefficient in the **Add Constraints** window to define the linear relationship between the selected variables and the entered constraint value. For the SNLP and SMINLP optimizers, the relationship can be linear or nonlinear. For the Quasi Newton and Pattern Search optimizers, the relationship must be linear.

Cost functions can be quite nonlinear. As a result, during the function evaluations of the algorithm, the cost function can vary significantly. Also, it is important to understand the relationship between optimization function evaluation and iteration. Every iteration, depending on the number of parameters to be optimized, performs several function evaluations. These function evaluations, depending on how nonlinear the cost function is, could show drastic changes. The presence of drastic changes has no bearing on whether the optimization algorithm converged or not.

In the case of non-gradient search-based optimization algorithms, such as "pattern search," which are entirely based on function evaluations, one could see drastic changes in the function evaluations depending on how nonlinear the cost function is. This could seem misleading as if the algorithm did not converge since in theory one expects the cost function to decrease from

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

one iteration to the next. The optimetrics, however, reports function evaluations and not necessarily the optimizer performance per iteration.

> **Note:**
>
> The MATLAB optimizer displays function evaluation when the **Show all functions evaluation** check box is selected. If the check box is not selected, it displays iteration.

Legacy Optimizers

These include

- Sequential Nonlinear Programming (Gradient)
- Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete)
- Quasi-Newton (Gradient)
- Pattern Search (Search-based)
- Genetic Algorithm (Random Search)

Quasi Newton (Gradient)

If the Sequential Non Linear Programming Optimizer has difficulty, and if the numerical noise is insignificant during the solution process, use the Quasi Newton optimizer to obtain the results. The Quasi Newton optimizer works on the basis of finding a minimum or maximum of a cost function which relates variables in the model or circuit to overall simulation goals. The user marks one or more variables in the project and defines a cost function in the optimization setup. The cost function relates the variable values to field quantities, design parameters like force or torque, power loss, etc. The optimizer can then maximize or minimize the value of the design parameter by varying the problem variables.

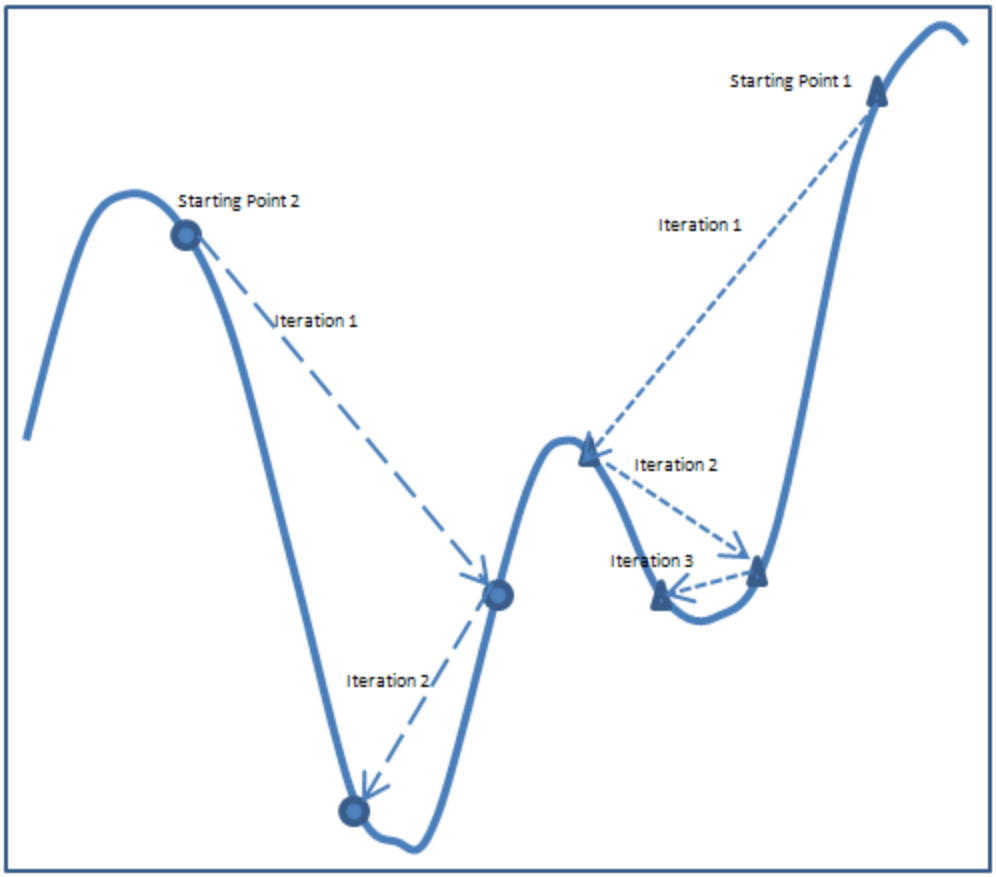
Sir Isaac Newton first showed that the maximum or minimum of any function can be determined by setting the derivative of a function with respect to a variable (x) to zero and solving for the variable. This approach leads to the exact solution for quadratic functions. However, for higher order functions or numerical analysis, an iterative approach is commonly taken. The function is approximated locally by a quadratic and the approximation is solved for the value of x. This value is placed back into the original function and used to calculate a gradient which provides a step direction and size for determining the next best value of x in the iteration process.

In the Quasi-Newton optimization procedure, the gradients and Hessian are calculated numerically. Essentially, the change in x and the change in the gradient are used to estimate the Hessian for the next iteration. The ratio of the change in cost to the change in the values of x provides the gradient, whereas, the ratio of the change in the gradients to the change in the values of x provides the Hessian for the next step and is know as the quasi-Newton condition. In order to perform the Quasi-Newton optimization, at least three solutions are required for each

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

parameter being varied. This can have a significant computational cost depending upon the type of analysis being performed.

There are numerous methods described in the literature for solving for the Hessian and the details of the method used by Optimetrics are beyond the scope of this document. However, as the Quasi-Newton method is, at its heart, a gradient method, it suffers from two fundamental problems common to optimization. The first is the possible presence of local minima. The following figure illustrates the problem of local minima.

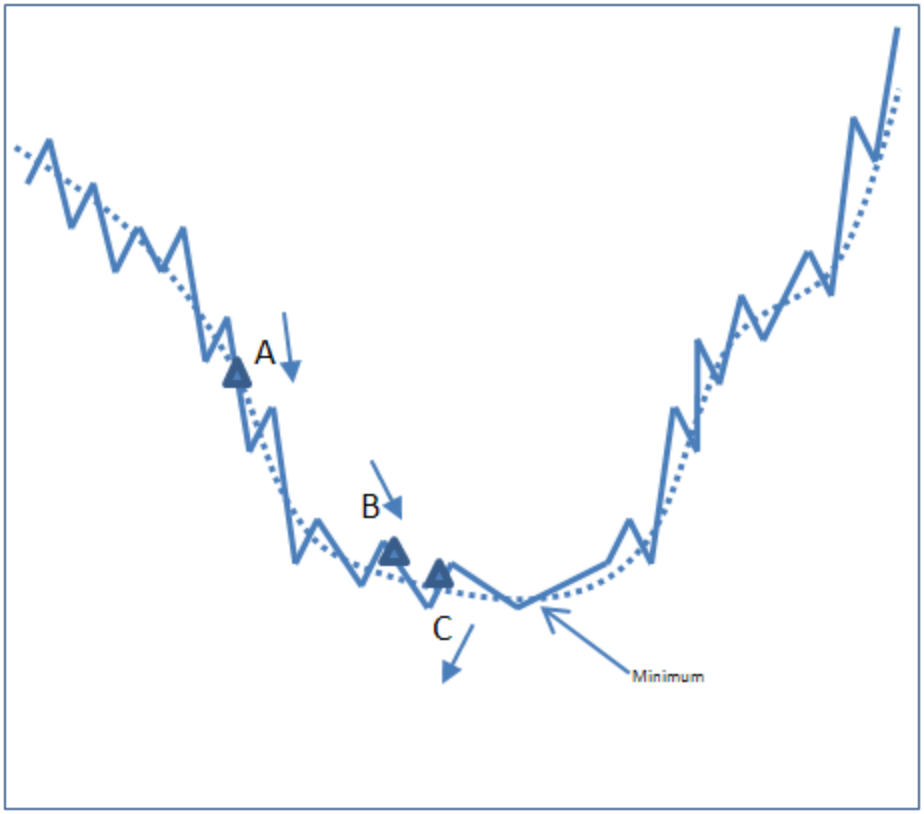


In this scenario, you can see that in order to find the minimum of the function over the domain, a number of factors will determine the overall success including the initial starting point, the initial set of gradients calculated, the allowable step size, etc. Once the optimizer has located a minimum, the Quasi-Newton approach will locate the bottom and will not search further for other possible minima. In the example shown, when the optimizer begins at the point labeled "Starting Point 1" the minima it finds is a local minima and not a good global solution to the problem.

The second basic issue with Quasi-Newton optimization is numerical noise. In gradient optimization, the derivatives are assumed to be smooth, well behaved functions. However, when the gradients are calculated numerically, the calculation involves taking the differences of

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

numbers that get progressively smaller. At some point, the numerical imprecision in the parameter calculations becomes greater than the differences calculated in the gradients and the solution will oscillate and may never reach convergence. To illustrate this, consider the figure shown below.



In this scenario, the optimizer is looking for the point labeled "minimum". Three possible solutions are labeled A, B and C, with each arrow indicating the direction of the derivative of the function at that point. If points A and B represent the last two solution points for the parameter, then it is easy to see that the changes in the magnitude and the consistent direction of the derivatives will serve to push the solution closer to the desired minimum. If, however, points A and C are the last two solution points respectively, the magnitude indicates the proper direction of movement, but the derivatives are opposite, possibly causing the solution to move away from the minimum, back in the direction of point A.

In order to use the Quasi-Newton optimizer effectively, the cost function should be based on parameters that exhibit a smooth characteristic (little numerical noise) and a starting point of the optimization should be chosen somewhat close to the expected minimum based on an understanding of the physical problem being optimized. This becomes increasingly difficult, however, when multiple parameters are being varied or when multiple parameters are to be optimized. In addition, the computational burden of multivariate optimization with Quasi-Newton

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

increases geometrically with the number of variables being optimized. As a result, this method should only be attempted when 1 or 2 variables are being optimized as a time.

For more information regarding Quasi-Newton optimization methods, see the following reference:

Schoenberg, Ronald. *Optimization with the Quasi-Newton Method*. Aptech Systems, Inc. 2001.

Pattern Search (Search-Based)

If the noise is significant in the nominal project, use the Pattern Search optimizer to obtain the results. It performs a grid-based simplex search, which makes use of simplices: triangles in 2D space or tetrahedra in 3D space. A simplex is a Euclidean geometric spatial element having the minimum number of boundary points, such as a line segment in one-dimensional space, a triangle in two-dimensional space, or a tetrahedron in three-dimensional space.

The cost value is calculated at the vertices of the simplex. The optimizer mirrors the simplex across one of its faces based on mathematical guidelines and determines if the new simplex provides better results. If it does not produce a better result, the next face is used for mirroring and the pattern continues. If no improvement occurs, the grid is refined. If improvement occurs, the step is accepted and the new simplex is generated to replace the original one. The figures below illustrate a triangular simplex mirrored several times to demonstrate the pattern search approach in two variables and the simplices superimposed on a 2D cost function to demonstrate the convergence toward a minimum in the cost function.

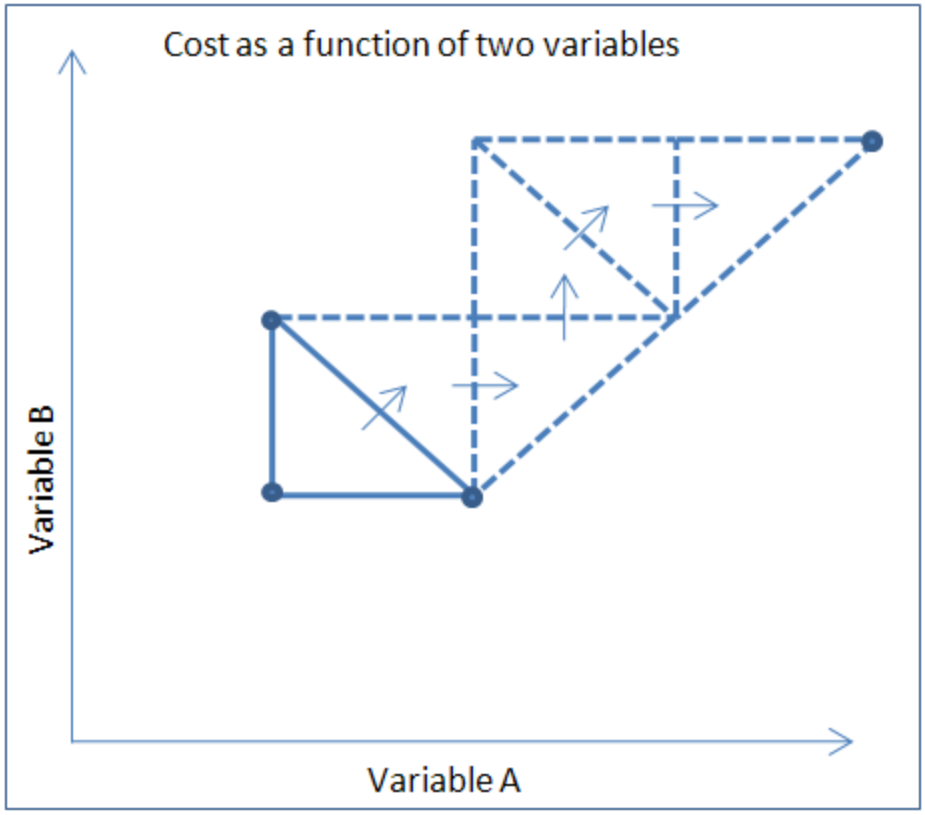
Cost functions can be quite nonlinear. As a result, during the function evaluations of the algorithm, the cost function can vary significantly. Also, it is important to understand the relationship between optimization function evaluation and iteration. Every iteration, depending on the number of parameters to be optimized, performs several function evaluations. These function evaluations, depending on how nonlinear the cost function is, could show drastic changes. The presence of drastic changes has no bearing on whether the optimization algorithm converged or not.

In the case of non-gradient search-based optimization algorithms, such as "pattern search," which are entirely based on function evaluations, one could see drastic changes in the function evaluations depending on how nonlinear the cost function is. This could seem misleading as if the algorithm did not converge since in theory one expects the cost function to decrease from one iteration to the next. The optimetrics, however, reports function evaluations and not necessarily the optimizer performance per iteration.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

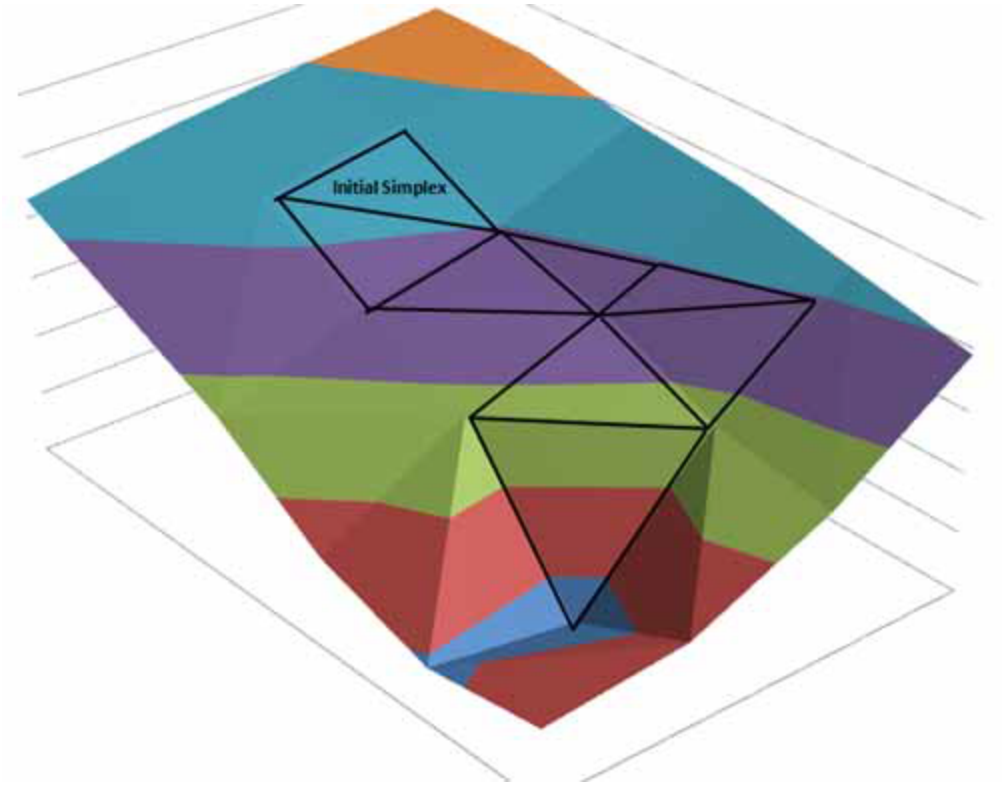
Note:

The MATLAB optimizer displays function evaluation when the **Show all functions evaluation** check box is selected. If the check box is not selected, it displays iteration.



The Pattern Search algorithms are extensible to three variable optimization by using tetrahedral simplices, however, they are not easily represented in graphical form. Generally, Pattern Search algorithms are not used when more than three variables are used in the optimization.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When there is no improvement in the cost function regardless of the direction the simplex is mirrored, then the simplex is subdivided into smaller simplices and the process restarted.

Pattern Search algorithms have several advantages over Quasi-Newton algorithms. First, they are less sensitive to noise because the cost function is evaluated at all node points on the simplex and the numerical noise averages out over the simplex. The second advantage is that the number of initial solutions is generally smaller. However, since the pattern search does not use gradient information to locate the minimum the process converges more slowly toward the true minimum, taking more steps to successively divide the simplices as the minimum is approached.

Sequential Nonlinear Programming (Gradient)

The main advantage of SNLP (Gradient) over Quasi Newton (Gradient) is that it handles the optimization problem in more depth. This optimizer assumes that the optimization variables span a continuous space. As a result, there is no Minimum Step Size specified in this optimizer and the variables may take any value within the allowable constraints and within the numerical precision limits of the simulator. Like Quasi Newton, the SNLP optimizer assumes that the noise is not significant. It does reduce the effect of the noise, but the noise filtering is not strong.

The SNLP optimizer approximates the FEA characterization with Response Surfaces (RS). With the FEA-approximation and with light evaluation of the cost function, SNLP has a good

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

approximation of the cost function in terms of the optimization variables. This approximation allows the SNLP optimizer to estimate the location of improving points. The overall cost approximations are more accurate. This allows the SNLP optimizer a faster practical convergence speed than that of quasi Newton.

The SNLP Optimizer creates the response surface using a polynomial approximation from the FEA simulation results available from past solutions. The response surface is most accurate in the local vicinity. The response surface is used in the optimization loop to determine the gradients and calculate the next step direction and distance. The response surface acts as a surrogate for the FEA simulation, reducing the number of FEA simulations required and greatly speeding the problem. Convergence improves as more FEA solutions are created and the response surface approximation improves.

The SNLP method is similar to the Sequential Quadratic Programming (SQP) method in two ways: Both are sequential, updating the optimizer state to the current optimal values and iterating. Sequential optimization can be thought of as walking a path, step by step, toward an optimal goal. SNLP and SQP optimizers are also similar in that both use local and inexpensive surrogates. However, in the SNLP case, the surrogate can be of a higher order and is more generally constrained. The goal is to achieve a surrogate model that is accurate enough on a wider scale, so that the search procedures are well lead by the surrogate, even for relatively large steps. All functions calculated by the supporting finite element product (for example, Maxwell 3D or HFSS) is assumed to be expensive, while the rest of the cost calculation (for example, an extra user-defined expression) — which is implemented in Optimetrics — is assumed to be inexpensive. For this reason, it makes sense to remove inexpensive evaluations from the finite element problem and, instead, implement them in Optimetrics. This optimizer holds several advantages over the Quasi Newton and Pattern Search optimizers.

Most importantly, due to the separation of expensive and inexpensive evaluations in the cost calculation, the SNLP optimizer is more tightly integrated with the supporting FEA tools. This tight integration provides more insight into the optimization problem, resulting in a significantly faster optimization process. A second advantage is that the SNLP optimizer does not require cost-derivatives to be approximated, protecting against uncertainties (noise) in cost evaluations. In addition to derivative-free state of the RS-based SNLP, the RS technique also proves to have noise suppression properties.

Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete)

The Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete) optimizer is equivalent to the SNLP (Gradient) optimizer with only one difference. Many problems require variables take only discrete values. One example might be to optimize on the number of turns in a coil. To be able to optimize on number of turns or quarter turns, the optimizer must handle discrete optimization variables. The SMINLP optimizer can mix continuous variables among the integers, or can have only integers, and works if all variables are continuous. The setup

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

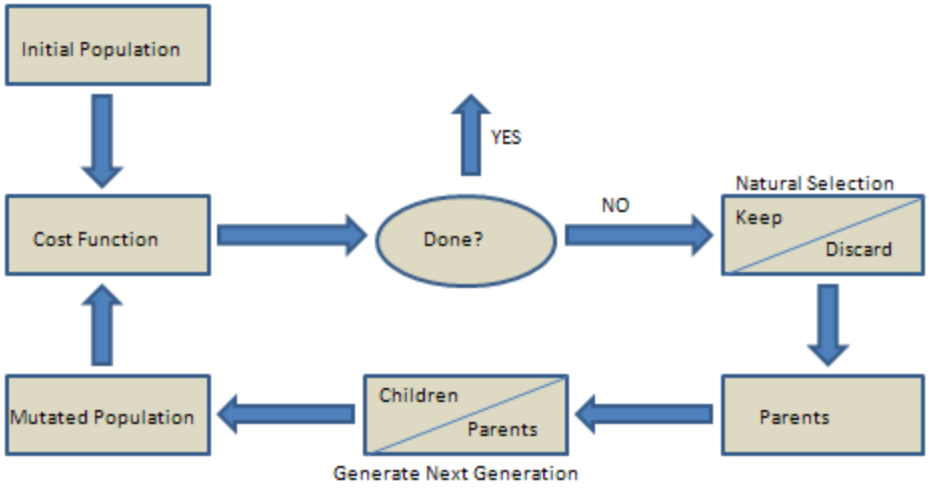
resembles the setup for SNLP, except that you must flag the integer variables.supporting integer variables. You can set up internal variables based on the integer optimization variable.

For example, consider N to be an integer optimization variable. By definition it can only assume integer values. You can establish another variable, which further depends on this one: K = 2.345 * N, or K = sin(30 * N). This way K has a discrete value, but is not necessarily integer. Or, one can use N directly as a design parameter.

Genetic Algorithm (Random Search)

Genetic Algorithm (Random Search) optimizers are part of a class of optimization techniques called stochastic optimizers. They do not use the information from the experiment or the cost function to determine where to further explore the design space. Instead, they use a type of random selection and apply it in a structured manner. The random selection of evaluations to proceed to the next generation has the advantage of allowing the optimizer to jump out of a local minima at the expense of many random solutions which do not provide improvement toward the optimization goal. As a result, the GA optimizer will run many more iterations and may be prohibitively slow.

The Genetic Algorithm search is an iterative process that goes through a number of generations (see picture below). In each generation some new individuals (Children / Number of Individuals) are created and the grown population participates in a selection (natural-selection) process that in turn reduces the size of the population to a desired level (Next Generation / Number of Individuals).



When a smaller set of individuals must be created from a bigger set, the GA selects individuals from the original set. During this process, better fit (in relation to the cost function) individuals are preferred. In the elitist selection, simply the best so many individuals are selected, but if you turn on the roulette selection, then the selection process gets relaxed. An iterative process starts

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

selecting the individuals and fills up the resulting set, but instead of selecting the best so many, we use a roulette wheel that has for each selection-candidate divisions made proportional to the fitness level (relative to the cost function) of the candidate. This means that the fitter the individual is, the larger the probability of his survival will be.

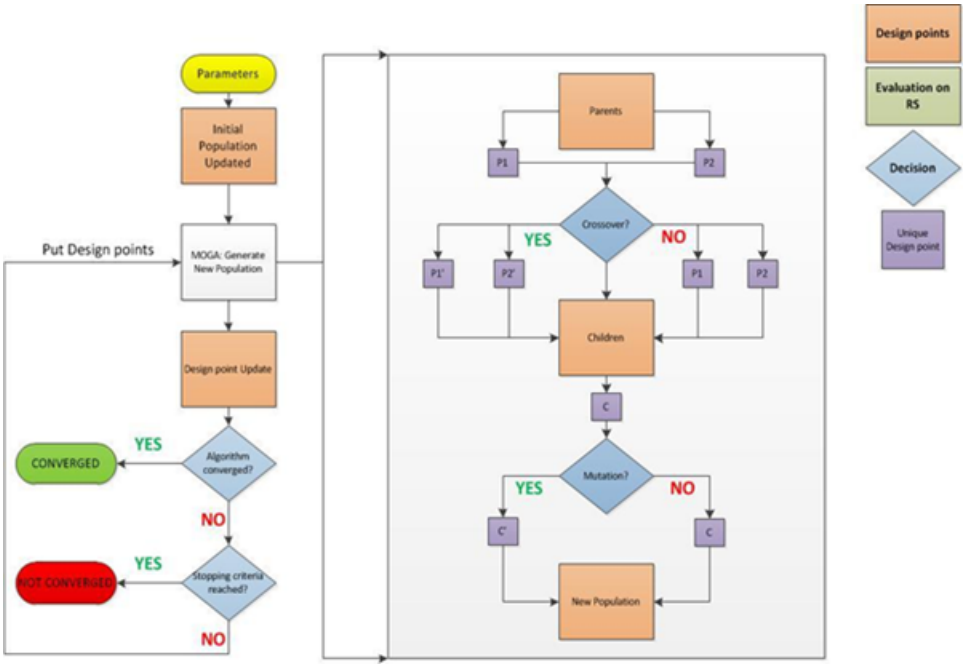
Multi-Objective Genetic Algorithm (MOGA)

The Multi-Objective Genetic Algorithm (MOGA) used in GDO is a hybrid variant of the popular NSGA-II (Non-dominated Sorted Genetic Algorithm-II) based on controlled elitism concepts. It supports all types of input parameters. The Pareto ranking scheme is done by a fast, non-dominated sorting method that is an order of magnitude faster than traditional Pareto ranking methods. The constraint handling uses the same non-dominance principle as the objectives. Therefore, penalty functions and Lagrange multipliers are not needed. This also ensures that the feasible solutions are always ranked higher than the infeasible solutions.

The first Pareto front solutions are archived in a separate sample set internally and are distinct from the evolving sample set. This ensures minimal disruption of Pareto front patterns already available from earlier iterations. You can control the selection pressure (and, consequently, the elitism of the process) to avoid premature convergence by altering the Maximum Allowable Pareto Percentage property. For more information about this and other MOGA properties, see Setup Multi-Objective Genetic Algorithm.

MOGA Workflow

The MOGA workflow follows:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MOGA Steps

1. **First Population of MOGA**

   The initial population is used to run MOGA.

2. **MOGA Generates a New Population**

   MOGA is run and generates a new population via cross-over and mutation. After the first iteration, each population is run when it reaches the number of samples defined by the Number of Samples Per Iteration property. For details, see **MOGA Steps to Generate New Population** below.

3. **Design Point Update**

   The design points in the new population are updated.

4. **Convergence Validation**

   The optimization is validated for convergence.

   - **Yes: Optimization Converged**

   MOGA converges when the Maximum Allowable Pareto Percentage or the Convergence Stability Percentage has been reached.

   - **No: Optimization Not Converged**

   If the optimization is not converged, the process continues to the next step.

5. **Stopping Criteria Validation**

   If the optimization has not converged, it is validated for fulfillment of stopping criteria.

   - **Yes: Stopping Criteria Met**

   When the Maximum Number of Iterations criterion is met, the process is stopped without having reached convergence.

   - **No: Stopping Criteria Not Met**

   If the stopping criteria have not been met, MOGA is run again to generate a new population (return to Step 2).

6. **Conclusion**

Steps 2 through 5 are repeated in sequence until the optimization has converged or the stopping criteria have been met. When either of these things occurs, the optimization concludes.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MOGA Steps to Generate a New Population

The process MOGA uses to generate a new population has two main steps: **Cross-over** and **Mutation**.

1. **Cross-over**

   Cross-over combines (mates) two chromosomes (parents) to produce a new chromosome (offspring). The idea behind cross-over is that the new chromosome can be better than both of the parents if it takes the best characteristics from each of the parents. Cross-over occurs during evolution according to a user-definable cross-over probability.

   - **Cross-over for Continuous Parameters**

     A cross-over operator that linearly combines two parent chromosome vectors to produce two new offspring according to the following equations:

     *Offspring1 = **a***Parent1 + (1-**a**)*Parent2*

     *Offspring2 = (1 - **a**)*Parent1 + **a***Parent2*

     Consider the following two parents (each consisting of four floating genes), which have been selected for cross-over:

     Parent1: (0.3) (1.4) (0.2) (7.4)

     Parent2: (0.5) (4.5) (0.1) (5.6)

     If a = 0.7, the following two offspring would be produced:

     Offspring1: (0.36) (2.33) (0.17) (6.86)

     Offspring2: (0.402) (2.981) (0.149) (6.842)

   - **Cross-over for Discrete Parameters and Continuous Parameters with Manufacturable Values**

     Each discrete parameter or continuous parameter with manufacturable values is represented by a binary chain corresponding to the number of levels. For example, a parameter with two values (levels) is encoded to one bit, a parameter with seven values is encoded to three bits, and an -bits chain represents a parameter with values.

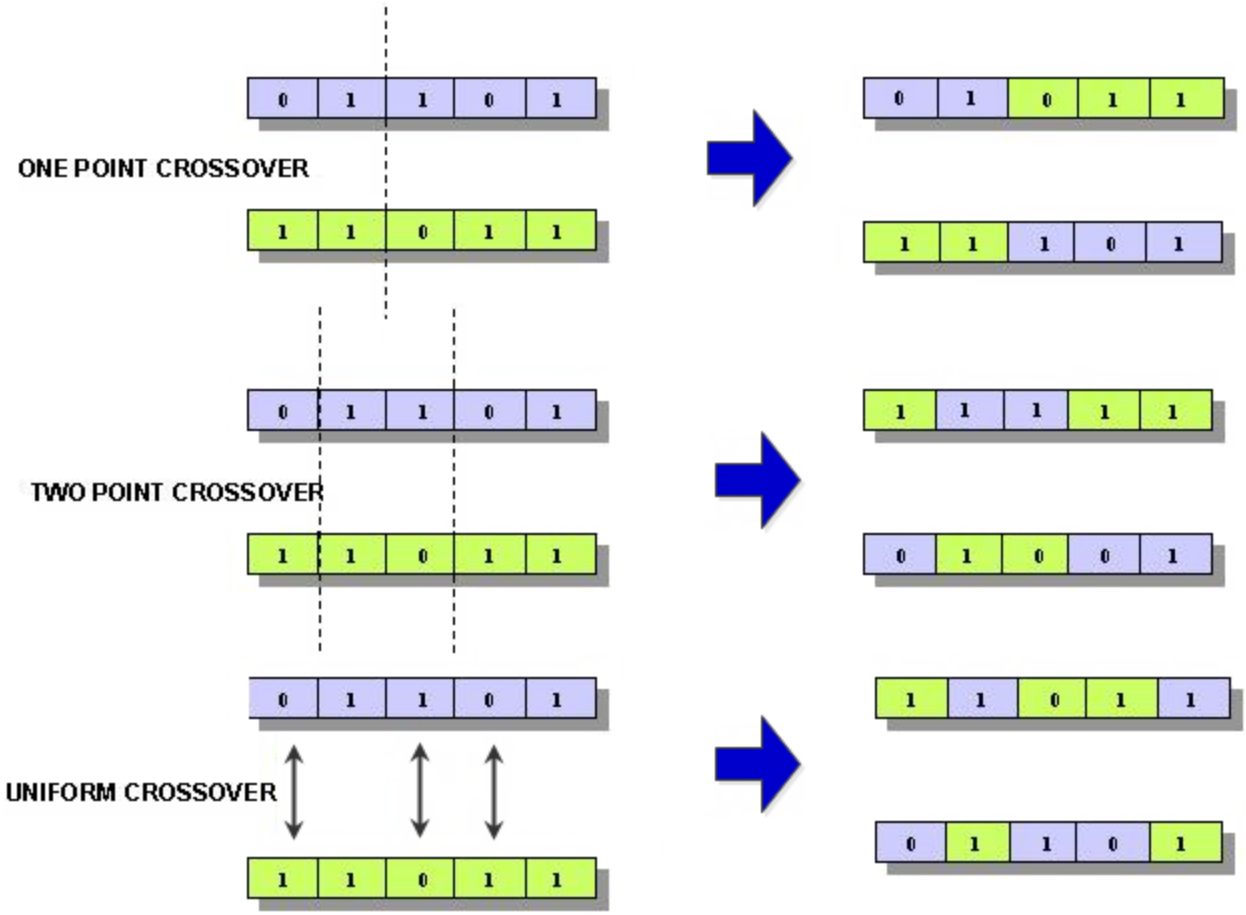
     The concatenation of these chains forms the chromosome, which crosses over with another chromosome.

     Three different kinds of cross-over are available:

     - **One-Point**: A one-point cross-over operator that randomly selects a cross-over point within a chromosome then interchanges the two parent chromosomes at this point to produce two new offspring.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Two-Point**: A two-point cross-over operator randomly selects two cross-over points within a chromosome then interchanges the two parent chromosomes between these points to produce two new offspring.
- **Uniform**: A uniform cross-over operator decides (with some probability, which is known as the "mixing ratio") which parent contributes each of the gene values in the offspring chromosomes. This allows the parent chromosomes to be mixed at the gene level rather than the segment level (as with one and two-point cross-over). For some problems, this additional flexibility outweighs the disadvantage of destroying building blocks.



2. **Mutation**

   Mutation alters one or more gene values in a chromosome from its initial state. This can result in entirely new gene values being added to the gene pool. With these new gene values, the genetic algorithm might be able to arrive at a better solution than was previously possible. Mutation is an important part of the genetic search, as it helps to prevent the population from stagnating at any local optima. Mutation occurs during evolution according to a user-defined mutation probability.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Mutation for Continuous Parameters**

  For continuous parameters, a polynomial mutation operator is applied to implement mutation:

  $$C = P + (UpperBound - LowerBound)\delta$$

  where C is the child, P is the parent, and δ is a small variation calculated from a polynomial distribution.

- **Mutation for Discrete Parameters and Continuous Parameters with Manufacturable Values**

  For discrete parameters or continuous parameters with manufacturable values, a mutation operator inverts the value of the chosen gene (0 goes to 1 and 1 goes to 0) with a probability of 0.5. This mutation operator can only be used for binary genes. The concatenation of these chains forms the chromosome, which crosses over with another chromosome.

Cost Function

Convergence Criteria in MOGA-Based Multi-Objective Optimization

Convergence criteria are the conditions that indicate when the optimization has converged. In the Multi-Objective Genetic Algorithm (MOGA)-based multi-objective optimization methods, the following convergence criteria are available:

Maximum Allowable Pareto Percentage

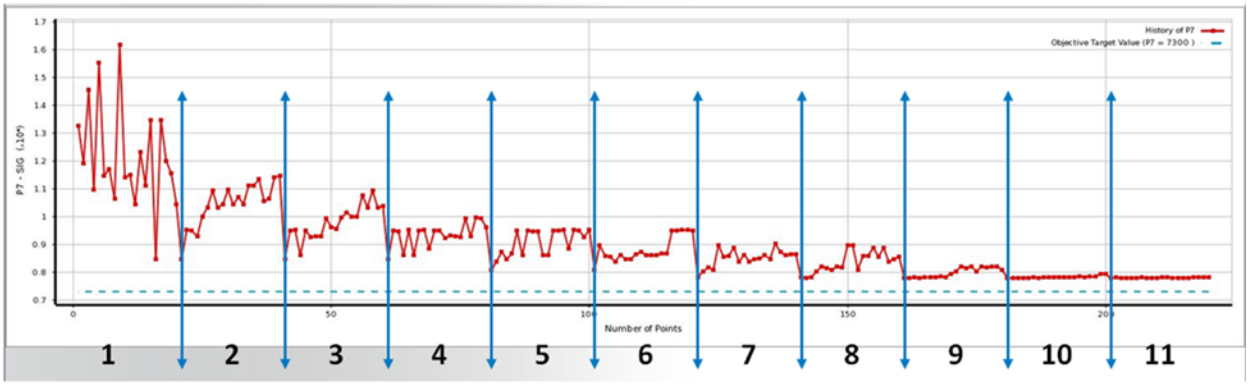
The **Maximum Allowable Pareto Percentage** criterion looks for a percentage that represents a specified ratio of Pareto points per Number of Samples Per Iteration. When this percentage is reached, the optimization is converged.

Convergence Stability Percentage

The **Convergence Stability Percentage** criterion looks for population stability, based on mean and standard deviation of the output parameters. When a population is stable with regards to the previous one, the optimization is converged. The criterion functions in the following sequence:

- **Population 1**: When the optimization is run, the first population is not taken into account. Because this population was not generated by the MOGA algorithm, it is not used as a range reference for the output range (for scaling values).
- **Population 2**: The second population is used to set the range reference. The minimum, maximum, range, mean, and standard deviation are calculated for this population.
- **Populations 3 – 11**: Starting from the third population, the minimum and maximum output values are used in the next steps to scale the values (on a scale of 0 to 100). The mean variations and standard deviation variations are checked. If both of these are smaller than the value for the Convergence Stability Percentage property, the algorithm is converged.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



At each iteration and for each active output, convergence occurs if

$$\frac{|Mean_i - Mean_{i-1}|}{Max - Min} < \frac{S}{100}$$

and

$$\frac{|StdDev_i - StdDev_{i-1}|}{Max - Min} < \frac{S}{100}$$

where

S = Stability Percentage

$Mean_i$ = Mean of the *i*-th Population

$StdDev_i$ = Standard Deviation of the *i*-th Population

Max = Maximum Output Value calculated on the first generated population of MOGA

Min = Minimum Output Value calculated on the first generated population of MOGA

MATLAB Optimizer

The MATLAB optimizer option lets you pass a script to MATLAB to perform the optimization. When the optimization is analyzed, MATLAB is launched and a script is passed in to MATLAB to perform the optimization. During the optimization, MATLAB will call back into our application to perform the solve and compute the cost. The cost will be reported back to MATLAB, and MATLAB's optimization will determine the next step in the optimization.

The optimization script is specified as part of the optimization setup. By modifying the optimization script, users can change the optimization parameters and optimization method as well as use the full power of MATLAB in their optimization.

Running the Optimization

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The MATLAB optimization is launched just like any other optimization. The Message Window will display status messages when MATLAB is being launched, and status messages will be generated for each solve that is being performed.

In most cases, MATLAB will terminate when the optimization has been completed. Some reasons why MATLAB would not terminate are:

- The user has modified the MATLAB script to not terminate MATLAB after the optimization.
- A syntax error or some other has occurred.
- The user has added some other code which runs after the optimization has completed.

System Requirements

In order to use MATLAB to perform optimizations from your application:

- A version of MATLAB must be installed on your system.
- The computing platform (e.g., 32/64 bit or Linux) of MATLAB MUST match the platform of the Ansys application you are using it with.
- You must have the MATLAB Optimization Toolkit installed.
- The MATLAB installation must include the MATLAB Optimization Toolbox. In addition, the MATLAB license must support using the Optimization Toolbox.

To see if the optimization toolbox is installed, users can type the "ver" command at the command prompt of a running MATLAB instance. For example:

```
>> ver
-----------------------------------------------------------------------
---------------------------------
MATLAB Version: 8.1.0.604 (R2023a)
MATLAB License Number: 162684
Operating System: Microsoft Windows 10
Java Version: Java 1.6.0_17-b04 with Sun Microsystems Inc. Java
HotSpot(TM) 64-Bit Server VM mixed mode
-----------------------------------------------------------------------
---------------------------------
MATLAB Version 8.1 (R2023a)
Simulink Version 8.1 (R2023a)
Optimization Toolbox Version 6.3 (R2023a)
```

To see if the optimization toolbox is licensed, you can use the "license('test','optimization_ toolbox')" command at the MATLAB command prompt:

```
>> license('test','optimization_toolbox')
ans =
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
1
```

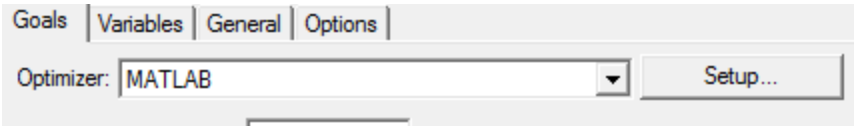
The answer will be 1 if the MATLAB Optimization Toolbox is licensed or 0 otherwise.

Specifying the MATLAB Location

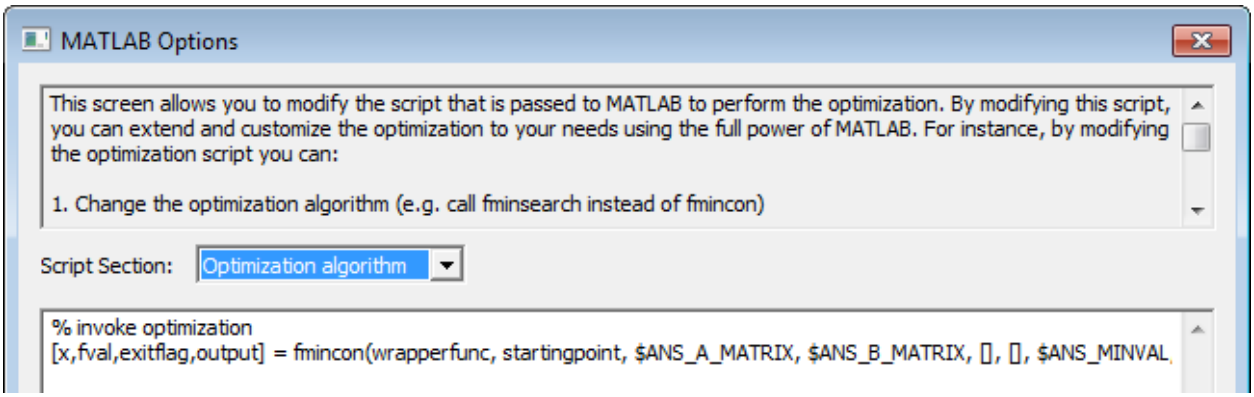
The Tools> General Options: Miscellaneous group contains a setting for the MATLAB location. This setting must to point to the version of MATLAB to be used for performing the optimization. The platform (32/64 bit or Linux) of the specified version of MATLAB must match the platform of this application.

MATLAB Optimization Setup

MATLAB optimization starts by creating an optimization and selecting MATLAB from the optimizer drop-down menu. If you select MATLAB as the optimizer, the *Setup Optimization* dialog box displays a **Setup...** button.



Select **Setup...** to open the *MATLAB Options* dialog box.



The upper text panel is informative. The Script section drop-down menu lets you select a lower panel display for Optimization algorithm, Options, or the Full script template.

This screen allows you to modify the script that is passed to MATLAB to perform the optimization. The complete script contains all the instructions necessary for MATLAB to connect to our application and perform the optimization, and a lot of that code is unimportant to users. We have addressed this issue by displaying a dropdown to let you view only the portion of code they are interested in without having to view the full script. The choices are:

- **Optimization algorithm**: displays only the line of code invoking the actual optimization function. By changing this line, users can use a different MATLAB function for

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

optimization. By default we use fmincon() which is a derivative based constrained optimization. By modifying this line, users could replace the fmincon() call with fminsearch () to use an unconstrained pattern searching optimizer or another optimization function. See the MATLAB documentation for details about available optimization functions.

- **Options**: Each optimization function contains a multitude of options and parameters which are set in the MATLAB script prior to actually calling the optimization function. By modifying these options, the optimization can be customized as desired. For instance, options can be set for fmincon() to specify the algorithm that it uses internally. See the MATLAB documentation for details about options available for each optimization function.
- **Full script template**: This choice displays the full optimization script that is passed to MATLAB.

The initial Script Section display for the Optimization algorithm shows the following:

```
% invoke optimization
[x,fval,exitflag,output] = fmincon(wrapperfunc, startingpoint, [],
[], [], [], $ANS_MINVAL, $ANS_MAXVAL, nlcon, options)
```

The initial Script Section Options display shows the following:

```
% customers can add their own options below
options = optimset(options, 'display', 'iter')
options = optimset(options, 'Algorithm', 'interior-point')
% options = optimset(options, 'PlotFcns', @optimplotfval)
```

You can modify the script to extend and customize the optimization to your needs. You must ensure that the script follows MATLAB syntax. For instance, by modifying the optimization script you can:

- Change the optimization algorithm (e.g., call fminsearch instead of fmincon).
- Change the parameters/options of the optimization algorithm (see the MATLAB documentation for details).
- Specify a plot function to provide graphical output during optimization.
- Specify a user defined output function to be called at completion or per iteration.

Symbols

When modifying the MATLAB code, users can use symbols to represent values from the optimization setup. The symbols and their definitions are listed below.

$ANS_VARIABLE_LIST:	list of variables we are optimizing
$ANS_STARTING_POINT:	vector of starting values of variables used in the optimization
$ANS_	maximum number of iterations specified in optimization setup

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MAXITERATIONS:	
$ANS_MINVAL:	vector of minimum values from optimization setup
$ANS_MAXVAL:	vector of maximum values from optimization setup
$ANS_MINSTEP:	vector of minimum step sizes from optimization setup
$ANS_MAXSTEP:	vector of maximum step sizes from optimization setup
$ANS_A_MATRIX	matrix of linear constraint coefficients (left-hand side) generated from optimization setup
$ANS_B_MATRIX	matrix of linear constraint bounds (right-hand side) generated from optimization setup

Note:

The linear constraints as generated for MATLAB have the form [A][x] <= [B], where [A] is the coefficient matrix, [x] is the variable list matrix (column vector), and [B] is the bounds matrix (column vector).

Note:

While modifying the script, please ensure that the script follows MATLAB syntax.

MATLAB Optimization Script Template

The script template shown in the Script Section is as follows:

% make sure platform matches

```
if strcmp(computer, '$ANS_EXPECTED_PLATFORM') ~= 1
   h = msgbox('32/64 platform does not match calling application,
  exiting')
   uiwait(h)
   exit
end
```

% add installation dir to search path so .mex file can be found

```
originalpath = addpath('$ANS_EXEDIR')
```

% connect back to opticomengine

```
callbackinterface = optimex('connect', '$ANS_CONNECTIONSTRING')
```

% set up optimization

% variables are: $ANS_VARIABLELIST

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
startingpoint = $ANS_STARTINGPOINT
options = optimset('MaxIter', $ANS_MAXITERATIONS)
iterationCallbackWrapper = @(x, optimValues, state) optimex
('notifyiterationcomplete', callbackinterface, x, optimValues.fval,
state)
options = optimset(options, 'OutputFcn', iterationCallbackWrapper)
```

% halt execution so debugger can be attached

% h = msgbox('attach debugger if desired')

% uiwait(h)

% attributes that user can pass to optimization algorithm

% variables are: $ANS_VARIABLELIST

% this is the objective function which returns cost

```
wrapperfunc = @(x)optimex('eval', callbackinterface, x)
```

% this is our non linear constraint function, returns no constraints

returnempty = @(x)[];

nlcon = @(x) deal(returnempty(x), returnempty(x));

% DO NOT EDIT THIS LINE - START OPTIONS SECTION

% customers can add their own options below

```
options = optimset(options, 'display', 'iter')
options = optimset(options, 'Algorithm', 'interior-point')
```

% options = optimset(options, 'PlotFcns', @optimplotfval)

% DO NOT EDIT THIS LINE - END OPTIONS SECTION

% DO NOT EDIT THIS LINE - START OPTIMIZATION ALGO SECTION

% invoke optimization

```
[x,fval,exitflag,output] = fmincon(wrapperfunc, startingpoint, $ANS_
A_MATRIX, $ANS_B_MATRIX, [], [], $ANS_MINVAL, $ANS_MAXVAL, nlcon,
options)
```

% DO NOT EDIT THIS LINE - END OPTIMIZATION ALGO SECTION

% write exit message to Ansoft message window (warning=0,error=1,info=2)

```
optimex('postansoftmessage', callbackinterface, 2, output.message)
```

% notify opticomengine that optimization is finished

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
optimex('optimizationfinished', callbackinterface, exitflag)
```

% restore original path

```
path = originalpath
```

% note: comment below line if you want MATLAB to remain

% running after optimization

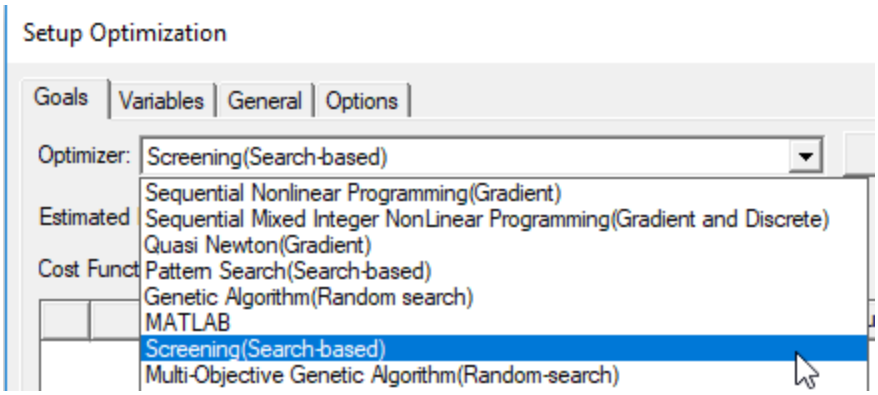
```
exit
```

Screening (Search Based) Optimizer Setup

Following is the procedure for setting up an optimization analysis using the Screening Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

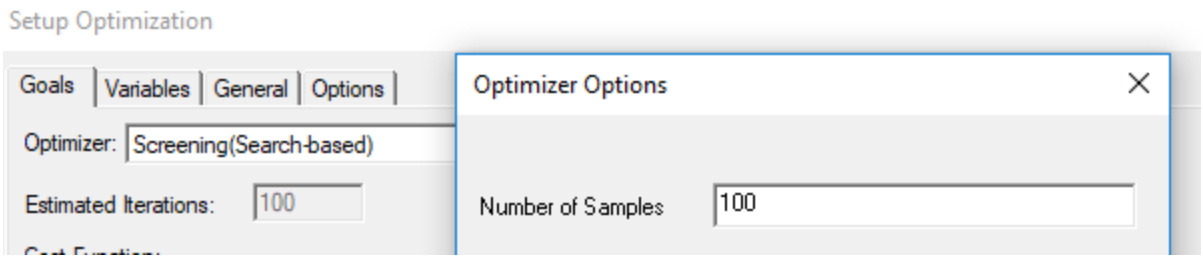
This is a non-iterative direct sampling method that uses a quasi-random number generator based on the Hammersley algorithm. You can start with Screening to locate the multiple tentative optima and then refine with NLPQL or MISQP to zoom in on the individual local maximum or minimum value. Usually Screening is used for preliminary design. Then, if you want to refine, these candidate points are used as starting points for gradient methods.

1. Set up the variables you want to optimize in the *Design Properties* dialog box. The variables must be swept in a Parametric setup.
2. Click **Maxwell** > **Optimetrics Analysis** > **Add Screening & Optimization**. The *Setup Optimization* window appears.
3. Under the **Goals** tab, select the optimizer by selecting **Screening (Search Based)** from the **Optimizer** drop-down menu.



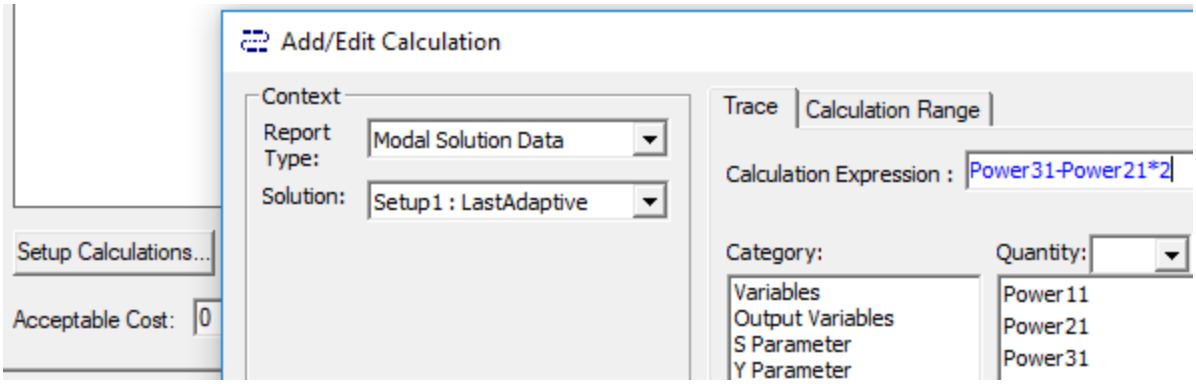
4. Optionally, press the **Setup** button to open the **Optimizer Options window** to change the default number of samples from 100.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



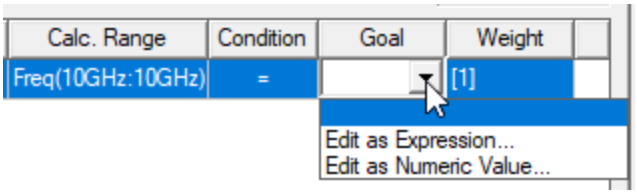
The number of samples must be greater than the number of enabled input parameters. The number of enabled input parameters is also the minimum number of samples required to generate the Sensitivities chart. You can enter a minimum of 2 and a maximum of 10,000. The default is 100 for a Direct Optimization system.

5. Add a cost function by selecting the **Setup Calculations** button to open the **Add/Edit Calculation** window.



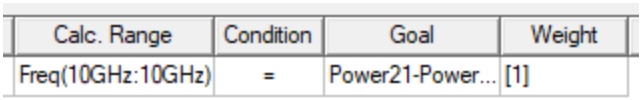
When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the **Add/EditCalculation** window.

6. In the Optimization setup, in the drop-down menu for the Goal column, select either Edit as Expression or Edit as Numeric Value... This reopens the **Add/Edit Calculation** window.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. If you are satisfied with the expression or value displayed, click **Done** to close the window. This enters the expression/value to the **Goal** column.



Calc. Range	Condition	Goal	Weight
Freq(10GHz:10GHz)	=	Power21-Power...	[1]

8. In the **Optimization** setup, if you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** drop-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

     A norm is a function that assigns a positive value to the cost function.

     For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors, which means that it is always less than zero. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

     The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.
9. Optionally, set the Acceptable Cost and Cost Function Noise.
10. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
11. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.
    - Optionally, set Linear constraints.
    - Select the **View all columns** check box to see all columns, including hidden columns.
12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.
13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Mixed-Integer Sequential Quadratic Programming (Gradient and Discrete) Optimizer Setup

Mixed-Integer Sequential Quadratic Programming (MISQP) is a mathematical optimization algorithm as developed by Oliver Exler, Thomas Lehmann and Klaus Schittkowski (NLPQL). This method solves Mixed-Integer Nonlinear Programming (MINLP) of the form:

Minimize:

$$f(x,y)$$

Subject to

$$g_j(x,y)=0,\ j=1,\ldots,m_e,$$
$$g_j(x,y)\geq 0,\ j=m_e+1,\ldots,m$$

where

$$x\in\mathbb{R}^{n_c}, y\in\mathbb{N}^{n_i}$$
$$x_l\leq x\leq x_u$$
$$y_l\leq y\leq y_u$$

The symbols and denote the vectors of the continuous and integer variables, respectively. It is assumed that problem functions and are continuously differentiable subject to all . It is not

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

assumed that integer variables can be relaxed. In other words, problem functions are evaluated only at integer points and never at any fractional values in between.

MISQP solves MINLP by a modified sequential quadratic programming (SQP) method. After linearizing constraints and constructing a quadratic approximation of the Lagrangian function, mixed-integer quadratic programs are successively generated and solved by an efficient branch-and-cut method. The algorithm is stabilized by a trust region method as originally proposed by Yuan for continuous programs. Second-order corrections are retained. The Hessian of the Lagrangian function is approximated by BFGS updates subject to the continuous and integer variables. MISQP is able to solve also non-convex nonlinear mixed-integer programs.

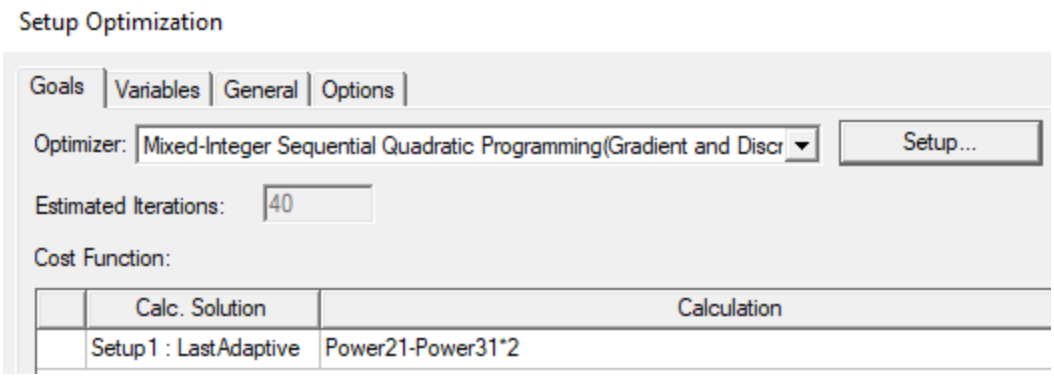
Following is the procedure for setting up an optimization analysis using the Mixed-Integer Sequential Quadratic Programming optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

The Mixed-Integer Sequential Quadratic Programming (Nonlinear Programming by Quadratic Lagrangian) method (MISQP)can be used for Direct Optimization systems. It allows you to generate a new sample set to provide a more refined approach than the Screening method. MISQP is available for both continuous and discrete input parameters, which is why mixed is in its name. MISQP can handle only one output parameter goal.. Other output parameters can be defined as constraints. For more information, see Convergence Rate % and Initial Finite Difference Delta % in NLPQL and MISQP and Nonlinear Programming by Quadratic Lagrangian (NLPQL).

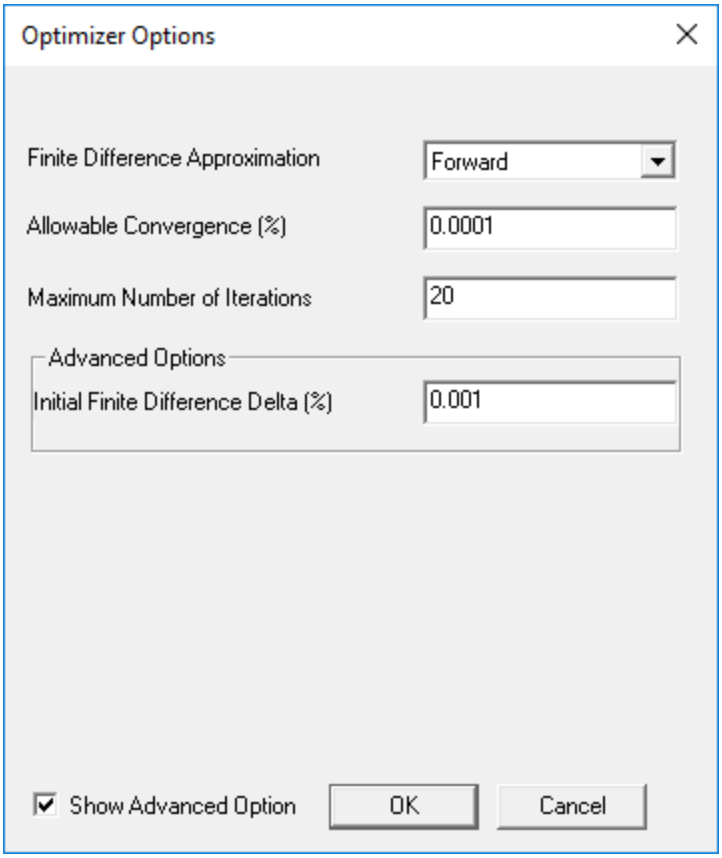
To generate samples and perform an NLPQL optimization:

1. Set up the variables you want to optimize in the *Design Properties* dialog box. The variables must be swept in a Parametric setup.
2. Click **Maxwell** > **Optimetrics Analysis** > **Add Screening & Optimization**. The *Setup Optimization* dialog box appears.
3. Under the **Goals** tab, select the optimizer by selecting **Mixed-Integer Sequential Quadratic Programming (Gradient and Discrete)** from the **Optimizer** drop-down menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Optionally, press the **Setup** button to open the **Optimizer Options window**.



- **Finite Difference Approximation**: When analytical gradients are not available, MISQP approximates them numerically. This property allows you to specify the method of approximating the gradient of the objective function. Choices are:
    - **Central**: Increases the accuracy of the gradient calculations by sampling from both sides of the sample point but increases the number of design point

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

evaluations by 50%. This method makes use of the initial point, as well as the forward point and rear point.

- **Forward**: Uses fewer design point evaluations but decreases the accuracy of the gradient calculations. This method makes use of only two design points, the initial point and forward point, to calculate the slope forward. This is the default method for new Direct Optimization systems.

- **Maximum Number of Iterations**: Stop criterion. Maximum number of iterations that the algorithm is to execute. If convergence happens before this number is reached, the iterations stop. This also provides an idea of the maximum possible number of function evaluations that are needed for the full cycle. For MISQP, the number of evaluations can be approximated according to the Finite Difference Approximation gradient calculation method, as follows:

  For **Central**: number of iterations * (2*number of inputs +1)

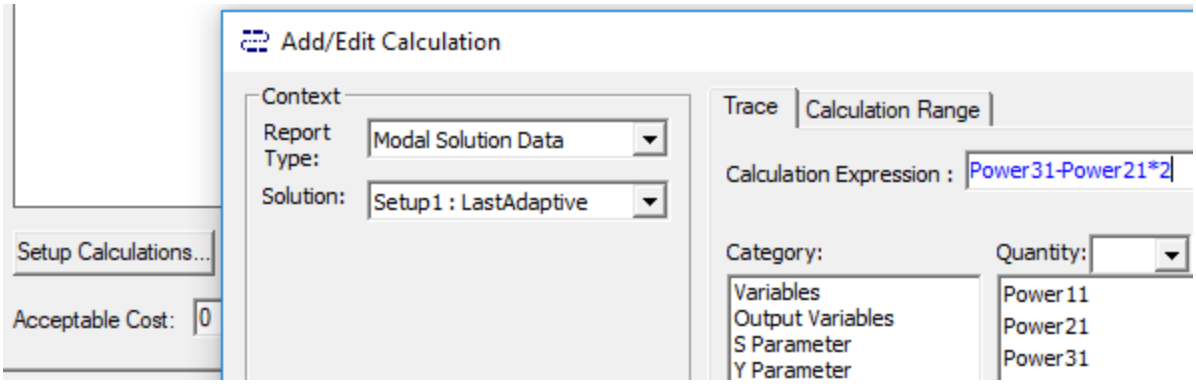
  For **Forward**: number of iterations * (number of inputs+1)

- **Allowable Convergence (%)**: Stop criterion. Tolerance to which the Karush-Kuhn-Tucker (KKT) optimality criterion is generated during the MISQP process. A smaller value indicates more convergence iterations and a more accurate (but slower) solution. A larger value indicates fewer convergence iterations and a less accurate (but faster) solution. For a Direct Optimization system, the default percentage value is 0.0001. The maximum percentage value is 100. These values are consistent across all problem types because the inputs, outputs, and gradients are scaled during the MISQP solution.
- **Initial Finite Difference Delta (%)**: Advanced option for specifying the relative variation used to perturb the current point to compute gradients. Used in conjunction with Allowable Convergence (%) to ensure that the delta in MISQP's calculation of finite differences is large enough to be seen above the noise in the simulation problem. This wider sampling produces results that are more clearly differentiated so that the difference is less affected by solution noise and the gradient direction is clearer. The value should be larger than both the value for Initial Finite Difference Delta (%) and the noise magnitude of the model. However, smaller values produce more accurate results, so set Initial Finite Difference Delta (%) only as high as necessary to be seen above simulation noise.

The default percentage value is 0.001. The minimum is 0.0001, and the maximum is 1. For parameters with Allowed Values set to **Manufacturable Values** or Snap to Grid, the value for Initial Finite Difference Delta (%) is ignored. In such cases, the closest allowed value is used to determine the finite difference delta.

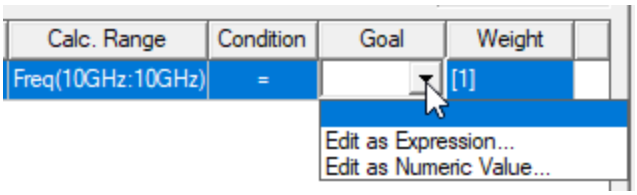
5. Add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



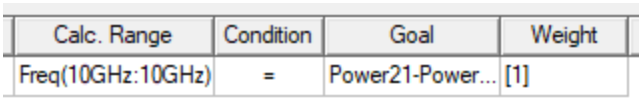
When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the *Add/EditCalculation* dialog box.

6. In the Optimization setup, in the drop-down menu, for the Goal column, select either Edit as Expression or Edit as Numeric Value...



This reopens the *Add/Edit Calculation* dialog box.

7. If you are satisfied with the expression or value displayed, click **Done** to close the dialog box. This enters the expression/value to the **Goal** column.



8. In the **Optimization** setup, if you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** drop-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors, which means that it is always less than zero. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.

9. Optionally, set the Acceptable Cost and Cost Function Noise.
10. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
11. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.
    - Optionally, set Linear constraints.
    - Select the **View all columns** check box to see all columns, including hidden columns.
12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

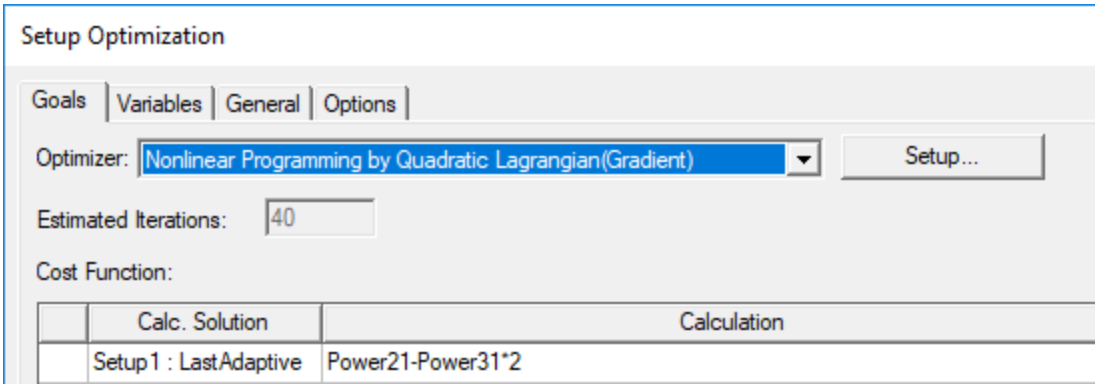
Nonlinear Programming by Quadratic Lagrangian (Gradient) Optimizer Setup

Following is the procedure for setting up an optimization analysis using the Nonlinear Programming by Quadratic Lagrangian (Gradient) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

The NLPQL (Nonlinear Programming by Quadratic Lagrangian) method can be used for Direct Optimization systems. It allows you to generate a new sample set to provide a more refined approach than the Screening method. Available for continuous input parameters only, NLPQL can handle only one output parameter goal. Other output parameters can be defined as constraints. For more information, see Convergence Rate % and Initial Finite Difference Delta % in NLPQL and MISQP and Nonlinear Programming by Quadratic Lagrangian (NLPQL).

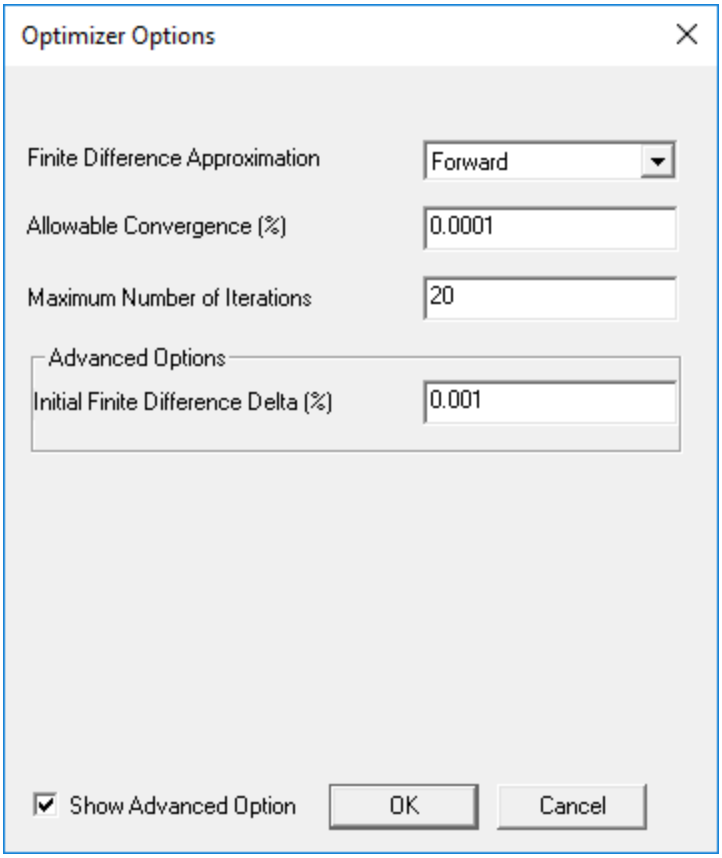
To generate samples and perform an NLPQL optimization:

1. Set up the variables you want to optimize in the *Design Properties* dialog box. The variables must be swept in a Parametric setup.
2. Click **Maxwell** > **Optimetrics Analysis** > **Add Screening & Optimization**. The *Setup Optimization* dialog box appears.
3. Under the **Goals** tab, select the optimizer by selecting **Nonlinear Programming by Quadratic Lagrangian (Gradient)** from the **Optimizer** drop-down menu.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Optionally, press the **Setup** button to open the *Optimizer Options window*.



- **Finite Difference Approximation**: When analytical gradients are not available, NLPQL approximates them numerically. This property allows you to specify the method of approximating the gradient of the objective function. Choices are:
  - **Central**: Increases the accuracy of the gradient calculations by sampling from both sides of the sample point but increases the number of design point evaluations by 50%. This method makes use of the initial point, as well as the forward point and rear point.
  - **Forward**: Uses fewer design point evaluations but decreases the accuracy of the gradient calculations. This method makes use of only two design points, the initial point and forward point, to calculate the slope forward. This is the default method for new Direct Optimization systems.
- **Maximum Number of Iterations**: Stop criterion. Maximum number of iterations that the algorithm is to execute. If convergence happens before this number is reached, the iterations stop. This also provides an idea of the maximum possible number of function evaluations that are needed for the full cycle. For NLPQL, the number of evaluations can be approximated according to the Finite Difference

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Approximation gradient calculation method, as follows:

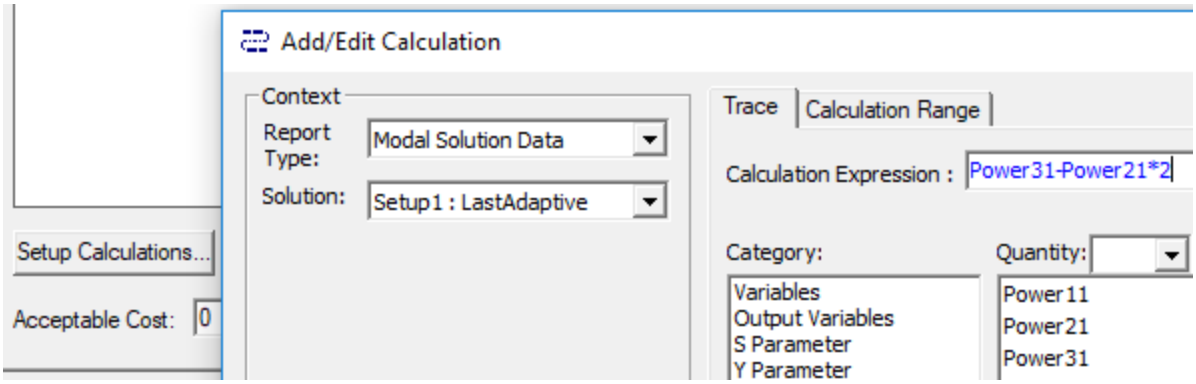
For **Central**: number of iterations * (2*number of inputs +1)

For **Forward**: number of iterations * (number of inputs+1)

- **Allowable Convergence (%)**: Stop criterion. Tolerance to which the Karush-Kuhn-Tucker (KKT) optimality criterion is generated during the NLPQL process. A smaller value indicates more convergence iterations and a more accurate (but slower) solution. A larger value indicates fewer convergence iterations and a less accurate (but faster) solution. For a Direct Optimization system, the default percentage value is 0.1. The maximum percentage value is 100. These values are consistent across all problem types because the inputs, outputs, and gradients are scaled during the NLPQL solution.
- **Initial Finite Difference Delta (%)**: Advanced option for specifying the relative variation used to perturb the current point to compute gradients. Used in conjunction with Allowable Convergence (%) to ensure that the delta in NLPQL's calculation of finite differences is large enough to be seen above the noise in the simulation problem. This wider sampling produces results that are more clearly differentiated so that the difference is less affected by solution noise and the gradient direction is clearer. The value should be larger than both the value for Initial Finite Difference Delta (%) and the noise magnitude of the model. However, smaller values produce more accurate results, so set Initial Finite Difference Delta (%) only as high as necessary to be seen above simulation noise.

The default percentage value is 0.001. The minimum is 0.0001, and the maximum is 1. For parameters with Allowed Values set to **Manufacturable Values** or Snap to Grid, the value for Initial Finite Difference Delta (%) is ignored. In such cases, the closest allowed value is used to determine the finite difference delta.

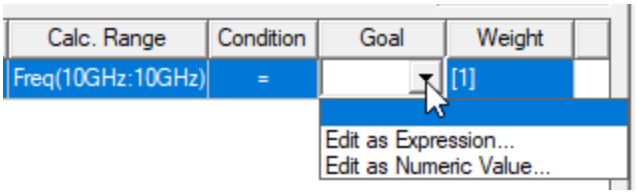
5. Add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

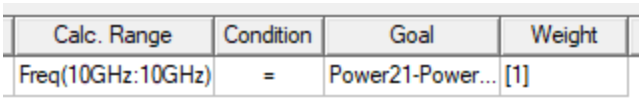
When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the *Add/EditCalculation* dialog box.

6. In the Optimization setup, in the drop-down menu for the Goal column, select either Edit as Expression or Edit as Numeric Value...



This reopens the *Add/Edit Calculation* dialog box.

7. If you are satisfied with the expression or value displayed, click **Done** to close the dialog box. This enters the expression/value to the **Goal** column.



8. In the **Optimization** setup, if you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** drop-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors, which means that it is always less than zero. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.

9. Optionally, set the Acceptable Cost and Cost Function Noise.
10. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
11. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
- Optionally, modify the values of fixed variables that are not being optimized.
- Optionally, set Linear constraints.
- Select the **View all columns** check box to see all columns, including hidden columns.

12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

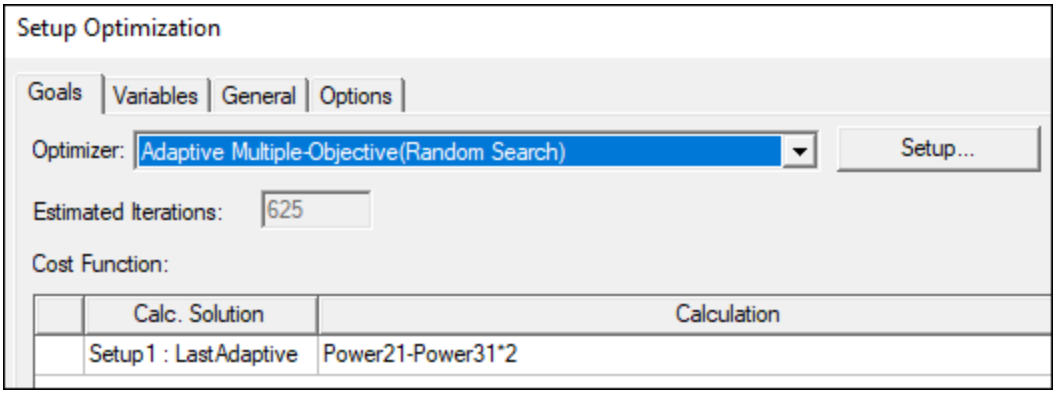
Adaptive Multiple-Objective (Random Search) Optimizer Setup

Following is the procedure for setting up an optimization analysis using the Adaptive Multiple Objective (Random Search) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

The Adaptive Multiple Objective (Kriging + MOGA) is an iterative algorithm that allows you to either generate a new sample set or use an existing set, providing a more refined approach than the Screening method. It uses the same general approach as MOGA, but applies the Kriging error predictor to reduce the number of evaluations needed to find the global optimum. The Adaptive Multiple-Objective method is available only for continuous input parameters, including those with manufacturable values. It can handle multiple objectives and multiple constraints. For more information, see Adaptive Multiple-Objective Optimization.

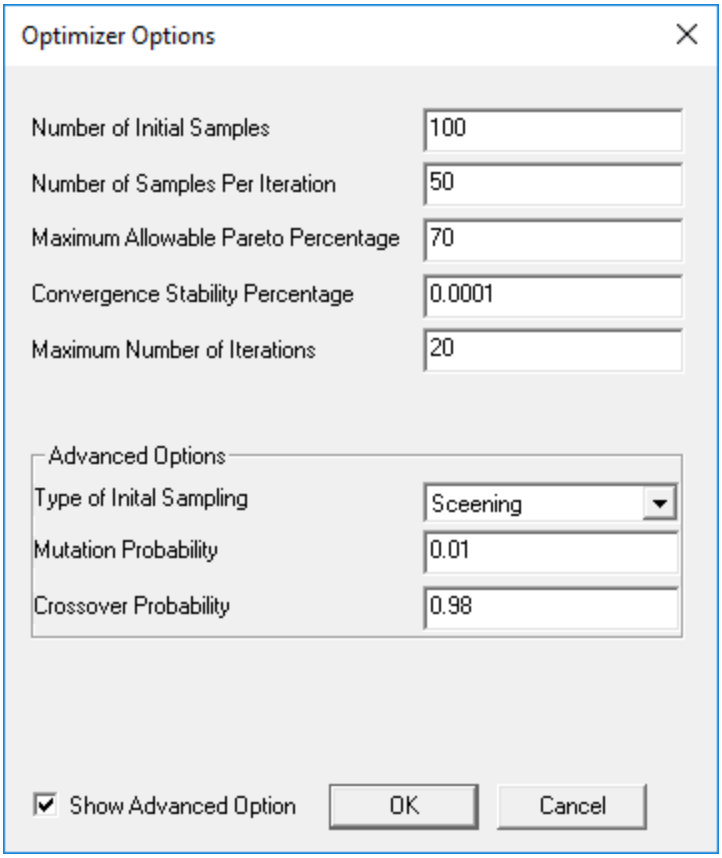
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Set up the variables you want to optimize in the *Design Properties* dialog box. The variables must be swept in a Parametric setup.
2. Click **Maxwell** > **Optimetrics Analysis** > **Add Screening & Optimization**. The *Setup Optimization* dialog box appears.
3. Under the **Goals** tab, select the optimizer by selecting **Adaptive Multiple Objective (Random-search)** from the **Optimizer** drop-down menu.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Optionally, press the **Setup** button to open the *Optimizer Options* window.



- **Number of Initial Samples**: Initial number of samples to use. This number must be greater than the number of enabled input parameters. The minimum recommended number of initial samples is 10 times the number of enabled input parameters. The larger the initial sample set, the better your chances of finding the input parameter space that contains the best solutions.

  The number of enabled input parameters is also the minimum number of samples required to generate the Sensitivities chart. You can enter a minimum of 2 and a maximum of 10000. The default is 100.

  If you switch from the Screening method to the MOGA method, MOGA generates a new sample set. For the sake of consistency, enter the same number of initial samples as you used for the Screening method.

- **Number of Samples Per Iteration**: Number of samples to iterate and update with each iteration. This number must be greater than the number of enabled input parameters but less than or equal to the number of initial samples. The default is for a Direct Optimization system.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can enter a minimum of 2 and a maximum of 10000.

- **Maximum Allowable Pareto Percentage**: Convergence criterion. Percentage value that represents the ratio of the number of desired Pareto points to the number of samples per iteration. When this percentage is reached, the optimization is converged. For example, a value of 70 with Number of Samples Per Iteration set to 200 would mean that the optimization should stop once the resulting front of the MOGA optimization contains at least 140 points. Of course, the optimization stops before that if the maximum number of iterations is reached.
- If the **Maximum Allowable Pareto Percentage** is too low (below 30), the process can converge prematurely. If the value is too high (above 80), the process can converge slowly. The value of this property depends on the number of parameters and the nature of the design space itself. The default is 70. Using a value between 55 and 75 works best for most problems. For more information, see Convergence Criteria in MOGA-Based Multi-Objective Optimization.
- **Convergence Stability Percentage**: Convergence criterion. Percentage value that represents the stability of the population based on its mean and standard deviation. This criterion allows you to minimize the number of iterations performed while still reaching the desired level of stability. When the specified percentage is reached, the optimization is converged. The default percentage is 0.0001. To not take the convergence stability into account, set to 0. For more information, see Convergence Criteria in MOGA-Based Multi-Objective Optimization.
- **Maximum Number of Iterations**: Stop criterion. Maximum number of iterations that the algorithm is to execute. If this number is reached without the optimization having reached convergence, iterations stop. This also provides an idea of the maximum possible number of function evaluations that are needed for the full cycle, as well as the maximum possible time it can take to run the optimization. For example, the absolute maximum number of evaluations is given by:

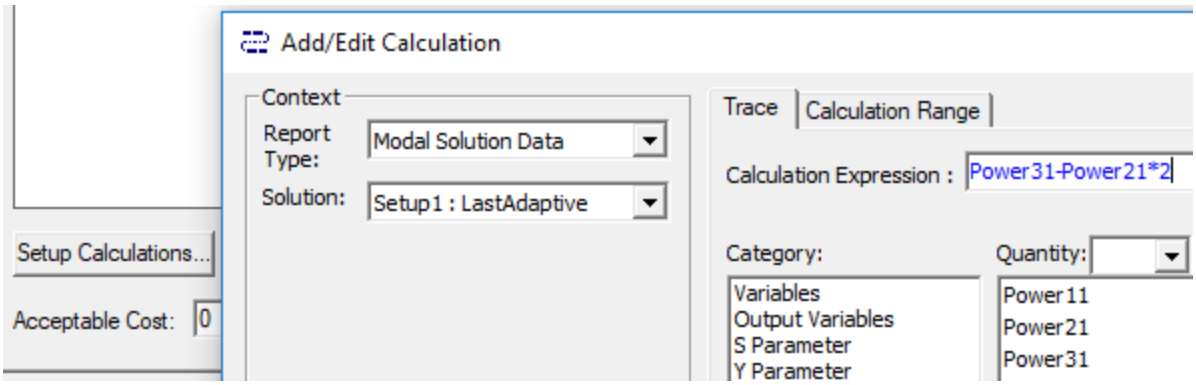
  Number of Initial Samples + Number of Samples Per Iteration * (Maximum Number of Iterations - 1)

- **Type of Initial Sampling**: Advanced option for generating different kinds of sampling. If you do not have any parameter relationships defined, set to Screening (default) or Optimal Space-Filling. If you have parameter relationships defined, the initial sampling must be performed by the constrained sampling algorithms because parameter relationships constrain the sampling. For such cases, this property is automatically set to Constrained Sampling.
- **Mutation Probability**: Advanced option for specifying the probability of applying a mutation on a design configuration. The value must be between 0 and 1. A larger value indicates a more random algorithm. If the value is 1, the algorithm becomes a pure random search. A low probability of mutation (<0.2) is recommended. The default is 0.01. For more information on mutation, see MOGA Steps to Generate a New Population.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

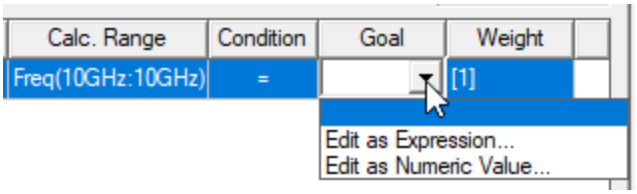
- **Crossover Probability**: Advanced option for specifying the probability with which parent solutions are recombined to generate offspring solutions. The value must be between 0 and 1. A smaller value indicates a more stable population and a faster (but less accurate) solution. If the value is 0, the parents are copied directly to the new population. A high probability of crossover (>0.9) is recommended. The default is 0.98.

5. Add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.

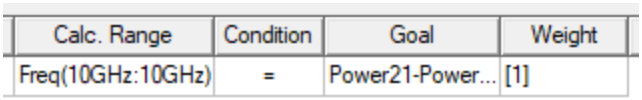


When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the *Add/EditCalculation* dialog box.

6. In the Optimization setup, in the Goal column drop-down menu, select either **Edit as Expression...** or **Edit as Numeric Value...**. This reopens the *Add/Edit Calculation* dialog box.



7. If you are satisfied with the expression or value displayed, click **Done** to close the dialog box. This enters the expression/value into the **Goal** column.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. In the **Optimization** setup, if you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The *Cost Function Norm Type* drop-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors, which means that it is always less than zero. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.
9. Optionally, set the Acceptable Cost and Cost Function Noise.
10. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
11. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.
    - Optionally, set Linear constraints.
    - Select the **View all columns** check box to see all columns, including hidden columns.
12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.
13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Adaptive Single-Objective (Gradient) Optimizer Setup

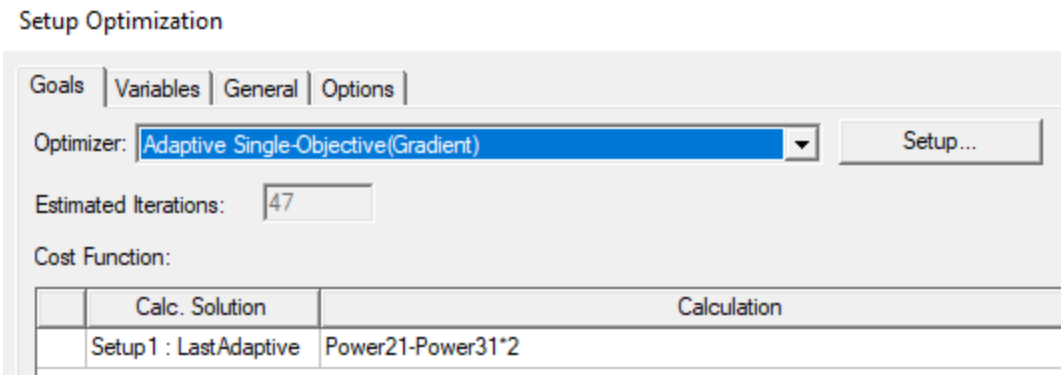
Following is the procedure for setting up an optimization analysis using the Adaptive Single-Objective (OSF + Kriging + MISQP) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

This gradient-based method employs automatic intelligent refinement to provide the global optima. It requires a minimum number of design points to build the Kriging response surface, but, in general, this method reduces the number of design points necessary for the optimization. Failed design points are treated as inequality constraints, making it fault-tolerant.

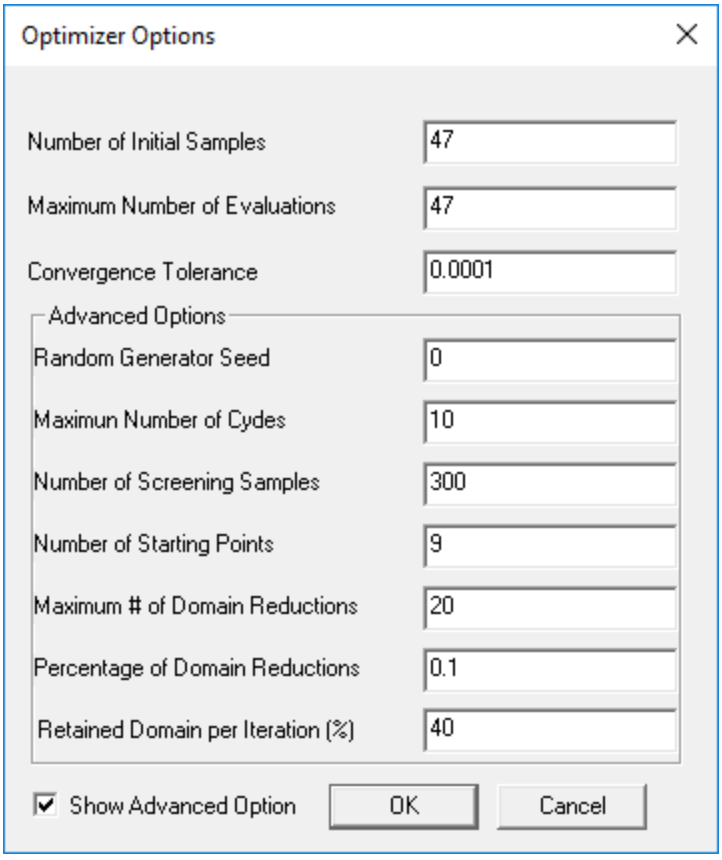
The Adaptive Single-Objective method is available for input parameters that are continuous, including those with manufacturable values. It can handle only one output parameter goal, although other output parameters can be defined as constraints. It does not support the use of parameter relationships in the optimization domain. For more information, see Adaptive Single-Objective Optimization (ASO). It requires advanced options. Ensure that the Show Advanced Options check box is selected.

1. Set up the variables you want to optimize in the *Design Properties* dialog box. The variables must be swept in a Parametric setup.
2. Click **Maxwell** > **Optimetrics Analysis** > **Add Screening & Optimization**. The *Setup Optimization* dialog box appears.
3. Under the **Goals** tab, choose the optimizer by selecting **Adaptive Single Objective (Gradient)** from the **Optimizer** drop-down menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Optionally, press the **Setup** button to open the *Optimizer Options* window and check **Advanced Options**.



- **Number of Initial Samples**: Number of samples generated for the initial Kriging and after all domain reductions for the construction of the next Kriging. You can enter a minimum of (NbInp+1)*(NbInp+2)/2 (also the minimum number of OSF samples required for the Kriging construction) or a maximum of 10,000. The default

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is (NbInp+1)*(NbInp+2)/2.

Because of the Adaptive Single-Objective workflow (in which a new OSF sample set is generated after each domain reduction), increasing the number of OSF samples does not necessarily improve the quality of the results and significantly increases the number of evaluations

- **Maximum Number of Evaluations**: Stop criterion. Maximum number of evaluations (design points) that the algorithm is to calculate. If convergence occurs before this number is reached, evaluations stop. This value also provides an idea of the maximum possible time it takes to run the optimization. The default is 20*(NbInp +1).
- **Convergence Tolerance**: Stop criterion. Minimum allowable gap between the values of two successive candidates. If the difference between two successive candidates is smaller than the value for Convergence Tolerance multiplied by the maximum variation of the parameter, the algorithm is stopped. A smaller value indicates more convergence iterations and a more accurate (but slower) solution. A larger value indicates fewer convergence iterations and a less accurate (but faster) solution. The default is 1E-06.
- **Random Generator Seed**: The value for initializing the random number generator invoked internally by OSF. The value must be a positive integer. This property allows you to generate different samplings by changing the value or to regenerate the same sampling by keeping the same value. The default is 0.
- **Maximum Number of Cycles**: Number of optimization loops that the algorithm needs, which in turns determines the discrepancy of the OSF. The optimization is essentially combinatorial, so a large number of cycles slows down the process. However, this makes the discrepancy of the OSF smaller. The value must be greater than 0. For practical purposes, 10 cycles is usually good for up to 20 variables. The default is 10.
- **Number of Screening Samples**: Number of samples for the screening generation on the current Kriging. This value is used to create the next Kriging (based on error prediction) and verified candidates.

  You can enter a minimum of (NbInp+1)*(NbInp+2)/2 (also the minimum number of OSF samples required for the Kriging construction) or a maximum of 10,000. The default is 100*NbInp for a Direct Optimization system. There is no default for a Response Surface Optimization system.

  The larger the screening sample set, the better the chances of finding good verified points. However, too many points can result in a divergence of the Kriging.

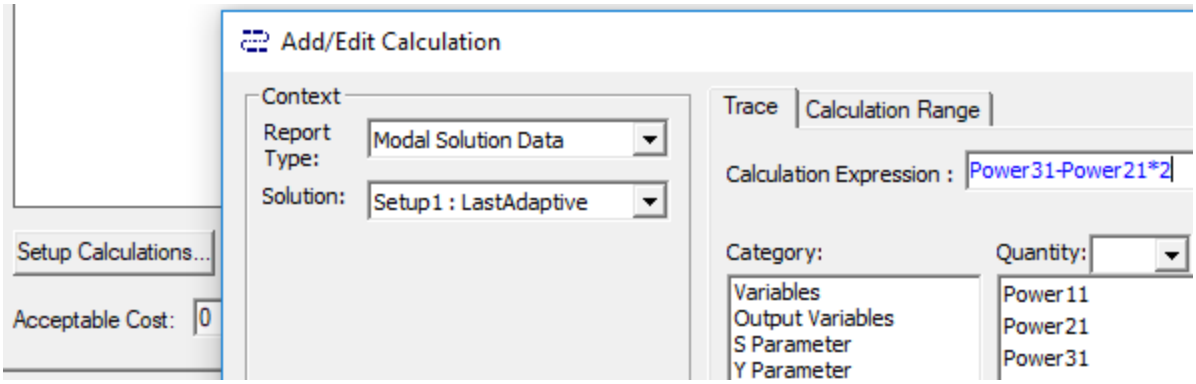
- **Number of Starting Points**: Determines the number of local optima to explore. The larger the number of starting points, the more local optima explored. In the case of a linear surface, for example, it is not necessary to use many points. This value must be less than the value for **Number of Screening Samples** because these

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

samples are selected in this sample. The default is the value for **Number of Initial Samples**.

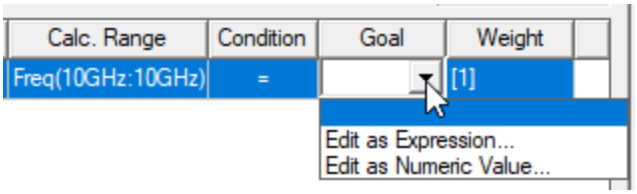
- **Maximum Number of Domain Reductions**: Stop criterion. Maximum number of domain reductions for input variation. (No information is known about the size of the reduction beforehand.) The default is 20.
- **Percentage of Domain Reductions**: Stop criterion. Minimum size of the current domain according to the initial domain. For example, with one input ranging between 0 and 100, the domain size is equal to 100. The percentage of domain reduction is 1%, so the current working domain size cannot be less than 1 (such as an input ranging between 5 and 6). The default is 0.1.
- **Retained Domain per Iteration (%)**: Advanced option that allows you to specify the minimum percentage of the domain you want to keep after a domain reduction. The percentage value must be between 10 and 90. A larger value indicates less domain reduction, which implies better exploration but a slower solution. A smaller value indicates a faster and more accurate solution, with the risk of it being a local one. The default percentage value is 40.

5. Add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.



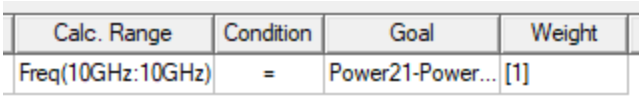
When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the **Add/EditCalculation** dialog box.

6. In the Optimization setup, in the Goal column drop-down menu, select either **Edit as Expression...** or **Edit as Numeric Value...**. This reopens the *Add/Edit Calculation* dialog box.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. If you are satisfied with the expression or value displayed, click **Done** to close the dialog box. This enters the expression/value into the **Goal** column.



Calc. Range	Condition	Goal	Weight
Freq(10GHz:10GHz)	=	Power21-Power...	[1]

8. In the **Optimization** setup, if you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** drop-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors, which means that it is always less than zero. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.
9. Optionally, set the Acceptable Cost and Cost Function Noise.
10. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
11. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.
    - Optionally, set Linear constraints.
    - Select the **View all columns** check box to see all columns, including hidden columns.
12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    Note:

    Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Optimization Variables and the Design Space

Once the optimization variables are specified, the optimizer handles each of them as an *n*-dimensional vector *x*. Any point in the design space corresponds to a particular *x*-vector and to a design instance. Each design instance may be evaluated via FEA and assigned a cost value;

therefore, the cost function is defined over the design space (cost(*x*): $R^n \rightarrow R$, where *n* is the number of optimization variables.

In practice, a solution of the minimization problem is sought only on a bounded subset of the Rn space. This subset is called the feasible domain and is defined via linear constraints.

Setting Up an Optimization Analysis

Optimization allows you to vary predefined variables in the nominal design to search for the solution that best satisfies a set of user defined goals or cost functions. Optimetrics modifies the variable values until the minimum is reached with acceptable accuracy.

Note:

- You can define more than one optimization analysis setup per design.
- You can create an Optimization setup before defining variables but all variables must be defined before you start the Optimization analysis.
- Once you have created an optimization analysis setup, you can copy and paste it, and then make changes to the copy, rather than redoing the whole process for minor changes.

To provide a broad range of capability, Optimetrics incorporates the following types of numerical optimizers:

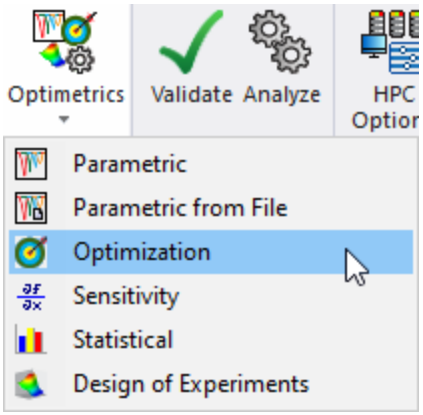
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Screening (Search based)
- Multi-Objective Genetic Algorithm
- Nonlinear Programming by Quadratic Lagrangian (Gradient)
- Mixed-Integer Sequential Quadratic Programming (Gradient and Discrete)
- Adaptive Multiple Objective (Gradient)
- Adaptive Single Objective
- MATLAB

Legacy Optimizers include

- Sequential Nonlinear Programming (Gradient)
- Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete)
- Quasi Newton (Gradient)
- Pattern Search (Search-based)
- Genetic Algorithm (Random search)

Click on the links above to view the setup procedure for each optimizer. Options for the analysis are listed in the table. Besides setting up an Optimization analysis from the Optimization menu, you can also the **Simulation** tab of the ribbon, and select from the menu under the Optimetrics icon:



The following *optional* optimization solution setup options can also be used:

- Modify the starting variable value.
- Edit the Calc. Range text field or use the Edit Calculation Range dialog box.
- Modify the minimum and maximum values of variables that will be optimized.
- Exclude variables from optimization.
- Modify the values of fixed variables that are not being optimized.
- Set the minimum and maximum step size between solved design variations (for the Quasi Newton (Gradient) and Pattern Search (Search based optimizers), **Variables** tab).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Set the minimum and maximum focus size (for the SNLP Gradient and SMINLP Gradient and Discrete optimizers, **Variables** tab).
- Set Linear constraints.
- Request that Optimetrics solve a parametric sweep before an optimization analysis.
- Request that Optimetrics solve a parametric sweep during an optimization analysis.
- Automatically update optimized variables to the optimal values during an optimization or after an optimization analysis is completed.
- Change the norm used for the cost function calculation (Advanced Option)
- Open the HPC and Analysis Options window.

> **Note:**
>
> Sweeping or using a complex variable is not allowed in any optimetrics setup, including optimization, statistical, sensitivity, and tuning setups.

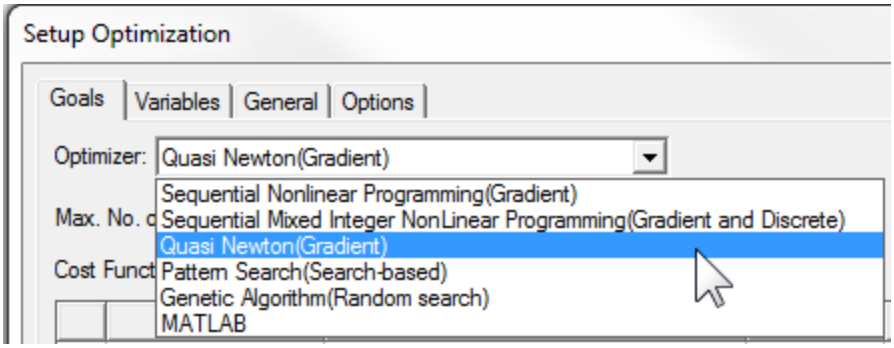
Quasi Newton (Gradient) Optimizer Setup

Following is the procedure for setting up an optimization analysis using the Quasi Newton (Gradient) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

1. Set up the variables you want to optimize in the *Design Properties* dialog box.
2. Click **Maxwell** > **Optimetrics Analysis** > **Add Screening & Optimization**.

   The *Setup Optimization* dialog box appears.

3. Under the **Goals** tab, select the optimizer by selecting **Quasi Newton (Gradient)** from the **Optimizer** drop-down menu. Selecting Quasi Newton (Gradient) enables the **Acceptable Cost** and **Noise** fields.



4. Type the maximum number of iterations you want Optimetrics to perform during the optimization analysis in the **Max. No. of Iterations** text box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Under **Cost Function**, add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.
6. Type the value of the cost function at which the optimization process should stop in the **Acceptable Cost** text box. Note that for Quasi Newton, if you specify 0 as the acceptable cost, the simulation stops after the first analysis.
7. Type the cost function noise in the **Noise** text box.
8. If you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** pull-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.
9. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
10. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization, and the **Min/Max Step Size** for the analysis.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.
    - Optionally, set Linear constraints.
    - Select the **View all columns** check box to see all columns, including hidden columns.
11. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.
12. Under the **Options** tab, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

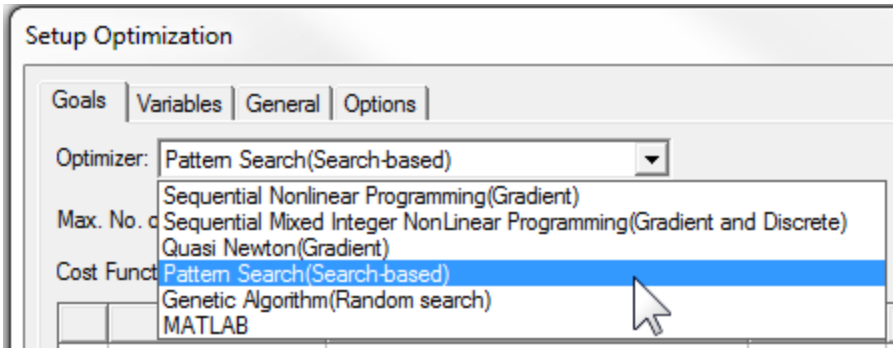
Pattern Search (Search-based) Optimizer Setup

Following is the procedure for setting up an optimization analysis using the Pattern Search (Search-based) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

1. Set up the variables you want to optimize in the *Design Properties* dialog box.
2. Click **Maxwell** > **Optimetrics Analysis** > **Add Screening & Optimization**.

   The *Setup Optimization* dialog box appears.

3. Under the **Goals** tab, select the optimizer by selecting **Pattern Search (Search-based)** from the **Optimizer** drop-down menu.



   Selecting Pattern Search enables the **Acceptable Cost** and **Noise** fields.

4. Type the maximum number of iterations you want Optimetrics to perform during the optimization analysis in the **Max. No. of Iterations** text box.
5. Under **Cost Function**, add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.
6. Type the value of the cost function at which the optimization process should stop in the **Acceptable Cost** text box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Type the cost function noise in the **Noise** text box.
8. If you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** pull-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.
9. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
10. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization, and the **Min/Max Step Size** for the analysis.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.
    - Optionally, set Linear constraints.

    Select the **View all columns** check box to see all columns, including hidden columns.
11. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.
12. Under the **Options** tab, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

> **Note:**
>
> Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Merit-Based Sequential Quadratic Programming(Gradient) Optimizer Setup

Following is the procedure for setting up an optimization analysis using the Merit-based Sequential Quadratic Programming(Gradient) Optimizer or MBSQ Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

1. Set up the variables you want to optimize in the *Design Properties* dialog box.
2. Click **Maxwell** > **Optimetrics Analysis** > **Add Screening & Optimization**.

   The *Setup Optimization* dialog box appears.
3. Under the **Goals** tab, select the optimizer by selecting **Merit-based Sequential Quadratic Programming(Gradient)** from the **Optimizer** drop-down menu.
4. Type the maximum number of iterations you want Optimetrics to perform during the optimization analysis in the **Max. No. of Iterations** text box.
5. Under **Cost Function**, add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.
6. If you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** pull-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.
7. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
8. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization, and the **Min/Max Focus** for the analysis.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
- Optionally, modify the values of fixed variables that are not being optimized.
- Optionally, set Linear constraints.
- Select the **View all columns** check box to see all columns, including hidden columns.

9. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

   Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

10. Use the **Options tab** if you want to enable use of a fast calculation-update algorithm to speed up Optimetrics and report updates during Optimetrics analyses, and to save the solution data for solved design variations in the analysis.

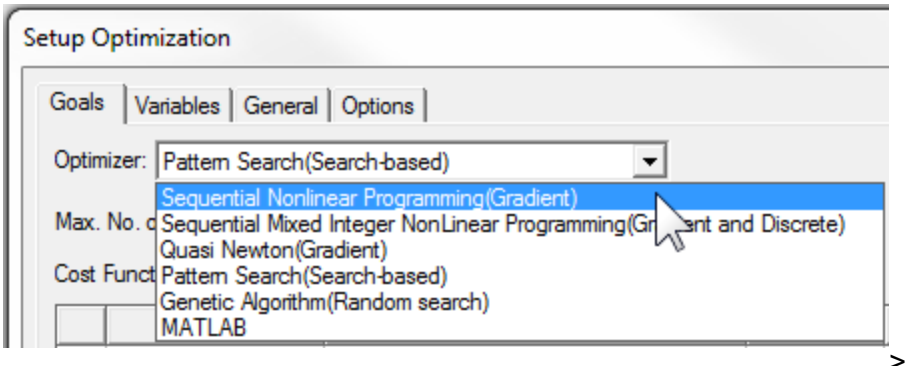
Sequential Nonlinear Programming (Gradient) Optimizer Setup

Following is the procedure for setting up an optimization analysis using the Sequential Nonlinear Programming (Gradient) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

1. Set up the variables you want to optimize in the *Design Properties* dialog box.
2. Click **Maxwell** > **Optimetrics Analysis** > **Add Screening & Optimization**.

   The *Setup Optimization* dialog box appears.

3. Under the **Goals** tab, select the optimizer by selecting **Sequential Nonlinear Programming (Gradient)** from the **Optimizer** drop-down menu.



4. Type the maximum number of iterations you want Optimetrics to perform during the optimization analysis in the **Max. No. of Iterations** text box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Under **Cost Function**, add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.
6. If you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** pull-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors, which means that it is always less than zero. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.
7. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
8. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization, and the **Min/Max Focus** for the analysis.
   - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
   - Optionally, modify the values of fixed variables that are not being optimized.
   - Optionally, set Linear constraints.
   - Select the **View all columns** check box to see all columns, including hidden columns.
9. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

   Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.
10. Under the **Options** tab, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

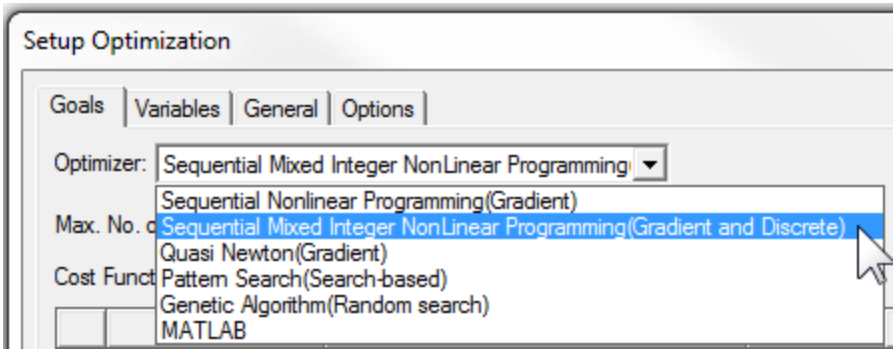
Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete) Optimizer Setup

Following is the procedure for setting up an optimization analysis using the Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

1. Set up the variables you want to optimize in the *Design Properties* dialog box.
2. Click **Maxwell** > **Optimetrics Analysis** > **Add Screening & Optimization**.

   The *Setup Optimization* dialog box appears.

3. Under the **Goals** tab, select the optimizer by selecting **Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete)** from the **Optimizer** drop-down menu.



4. Type the maximum number of iterations you want Optimetrics to perform during the optimization analysis in the **Max. No. of Iterations** text box.
5. Under **Cost Function**, add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. If you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** pull-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.
7. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
8. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization, and the **Min/Max Focus** for the analysis.
   - Check the Integer box for integer variables.
   - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
   - Optionally, modify the values of fixed variables that are not being optimized.
   - Optionally, set Linear constraints.
   - Select the **View all columns** check box to see all columns, including hidden columns.
9. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

   Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.
10. Under the **Options** tab, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

> **Note:**
>
> Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

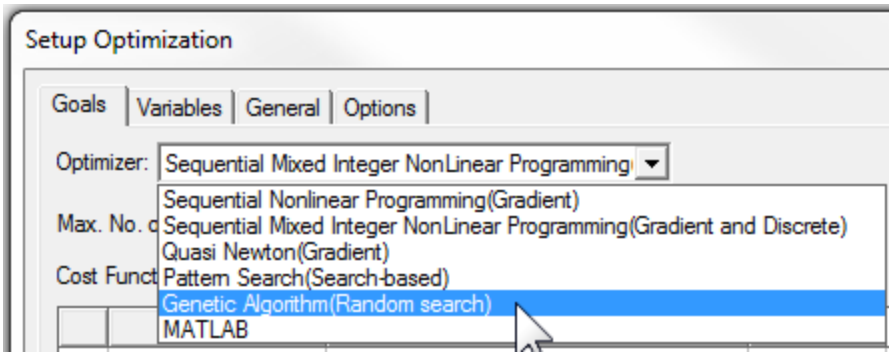
Genetic Algorithm (Random Search) Optimizer Setup

Following is the procedure for setting up an optimization analysis using the Genetic Algorithm (Random search) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

1. Set up the variables you want to optimize in the *Design Properties* dialog box.
2. Click **Maxwell** > **Optimetrics Analysis** > **Add Screening & Optimization**.

   The *Setup Optimization* dialog box appears.

3. Under the **Goals** tab, select the optimizer by selecting **Genetic Algorithm(Random search)** from the **Optimizer** drop-down list.



4. Click **Setup...** to modify the Advanced Genetic Algorithm Optimizer Options.
5. Under **Cost Function**, add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.
6. If you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** drop-down list appears.

   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

function uses the maximum among all the weighted goal errors. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.

7. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
8. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization, and the **Min/Max Focus** for the analysis.
   - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
   - Optionally, modify the values of fixed variables that are not being optimized.
   - Optionally, set Linear constraints.
   - Select the **View all columns** check box to see all columns, including hidden columns.
9. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

   Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.
10. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

MATLAB Optimizer Setup

Following is the procedure for setting up an optimization analysis using the MATLAB Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Set up the variables you want to optimize in the *Design Properties* dialog box.
2. Click **Maxwell** > **Optimetrics Analysis** > **Add Screening & Optimization**.

   The *Setup Optimization* dialog box appears.
3. Under the **Goals** tab, select the optimizer by selecting **MATLAB** from the **Optimizer** drop-down menu. Selecting **MATLAB** enables the **Acceptable Cost** and **Noise** fields.
4. Click **Setup...** to modify the MATLAB Optimizer Options.
5. Type the maximum number of iterations you want Optimetrics to perform during the optimization analysis in the Max. No. of Iterations text box.
6. Under Cost Function, add a cost function by selecting the Setup Calculations button to open the Add/Edit Calculation dialog box.
7. Type the value of the cost function at which the optimization process should stop in the Acceptable Cost text box.
8. Type the cost function noise in the Noise text box.
9. If you want to select a Cost Function Norm Type:
   - Check the Show Advanced Option check box.

     The **Cost Function Norm Type** drop-down list appears.
   - Select L1, L2, or Maximum.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors. (For more detail, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.
10. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
11. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization, and the **Min/Max Focus** for the analysis.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.
    - Optionally, set Linear constraints.
    - Select the **View all columns** check box to see all columns, including hidden columns.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

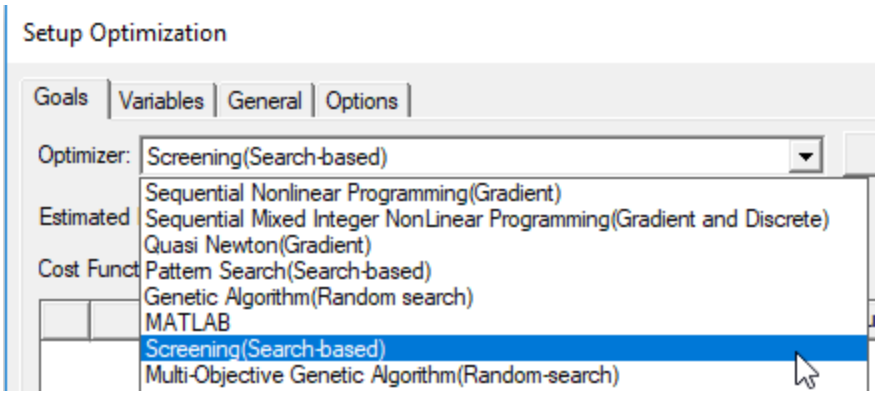
Screening (Search Based) Optimizer Setup

Following is the procedure for setting up an optimization analysis using the Screening Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

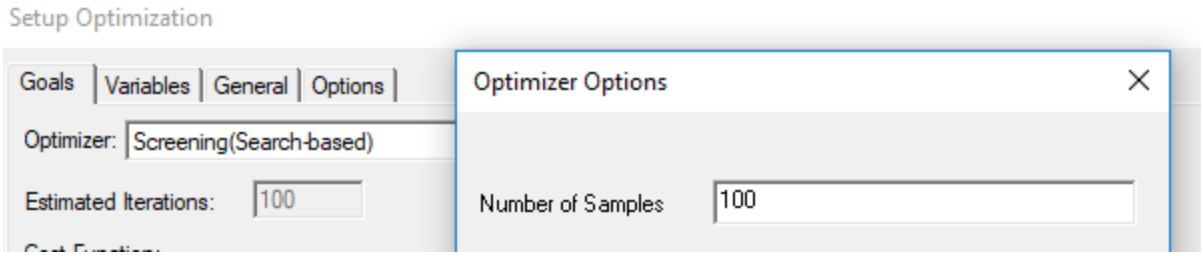
This is a non-iterative direct sampling method that uses a quasi-random number generator based on the Hammersley algorithm. You can start with Screening to locate the multiple tentative optima and then refine with NLPQL or MISQP to zoom in on the individual local maximum or minimum value. Usually Screening is used for preliminary design. Then, if you want to refine, these candidate points are used as starting points for gradient methods.

1. Set up the variables you want to optimize in the *Design Properties* dialog box. The variables must be swept in a Parametric setup.
2. Click **Maxwell** > **Optimetrics Analysis** > **Add Screening & Optimization**. The *Setup Optimization* window appears.
3. Under the **Goals** tab, select the optimizer by selecting **Screening (Search Based)** from the **Optimizer** drop-down menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

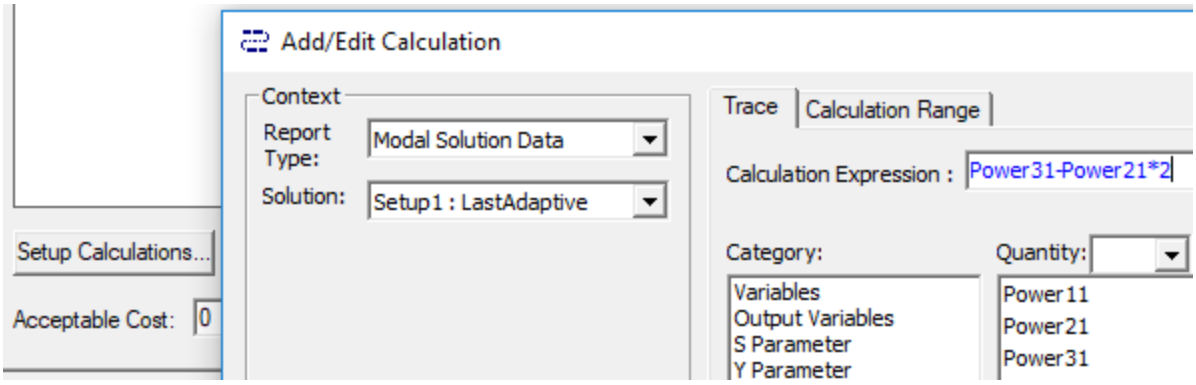


4. Optionally, press the **Setup** button to open the **Optimizer Options window** to change the default number of samples from 100.



The number of samples must be greater than the number of enabled input parameters. The number of enabled input parameters is also the minimum number of samples required to generate the Sensitivities chart. You can enter a minimum of 2 and a maximum of 10,000. The default is 100 for a Direct Optimization system.

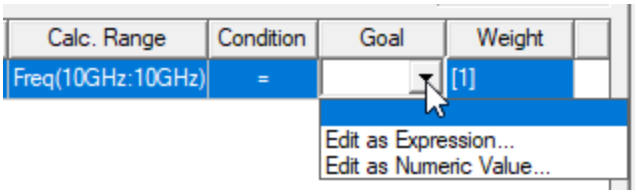
5. Add a cost function by selecting the **Setup Calculations** button to open the **Add/Edit Calculation** window.



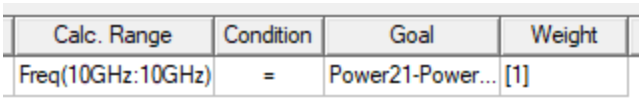
When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the **Add/EditCalculation** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. In the Optimization setup, in the drop-down menu for the Goal column, select either Edit as Expression or Edit as Numeric Value... This reopens the **Add/Edit Calculation** window.



7. If you are satisfied with the expression or value displayed, click **Done** to close the window. This enters the expression/value to the **Goal** column.



8. In the **Optimization** setup, if you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** drop-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

     A norm is a function that assigns a positive value to the cost function.

     For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors, which means that it is always less than zero. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

     The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.
9. Optionally, set the Acceptable Cost and Cost Function Noise.
10. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
11. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.
    - Optionally, set Linear constraints.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select the **View all columns** check box to see all columns, including hidden columns.

12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

> **Note:**
>
> Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Multi-Objective Genetic Algorithm(Random Search) Optimizer Setup

Following is the procedure for setting up an optimization analysis using the Multi-Objective Genetic Algorithm (Random Search) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

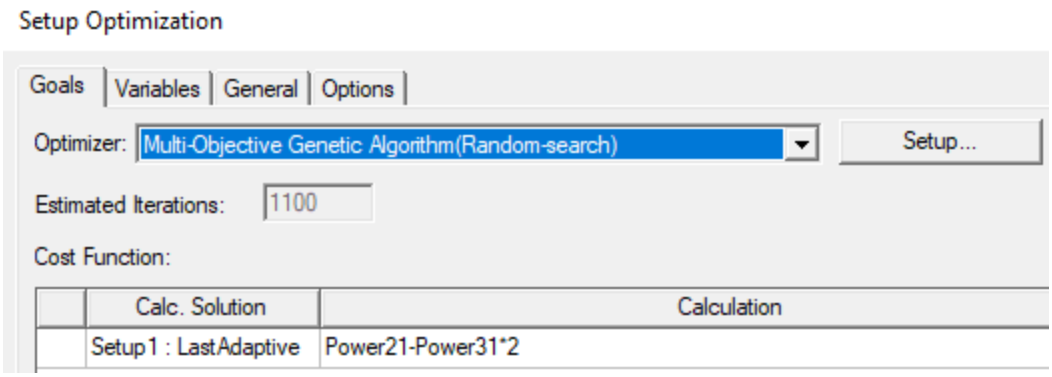
The Multi-Objective Genetic Algorithm (MOGA) is a hybrid variant of the popular NSGA-II (Non-dominated Sorted Genetic Algorithm-II) based on controlled elitism concepts. It supports all types of input parameters. The Pareto ranking scheme is done by a fast, non-dominated sorting method that is an order of magnitude faster than traditional Pareto ranking methods. The constraint handling uses the same non-dominance principle as the objectives. Therefore, penalty functions and Lagrange multipliers are not needed. This also ensures that the feasible solutions are always ranked higher than the infeasible solutions.

The first Pareto front solutions are archived in a separate sample set internally and are distinct from the evolving sample set. This ensures minimal disruption of Pareto front patterns already available from earlier iterations. You can control the selection pressure (and, consequently, the elitism of the process) to avoid premature convergence by altering the Maximum Allowable Pareto Percentage property.

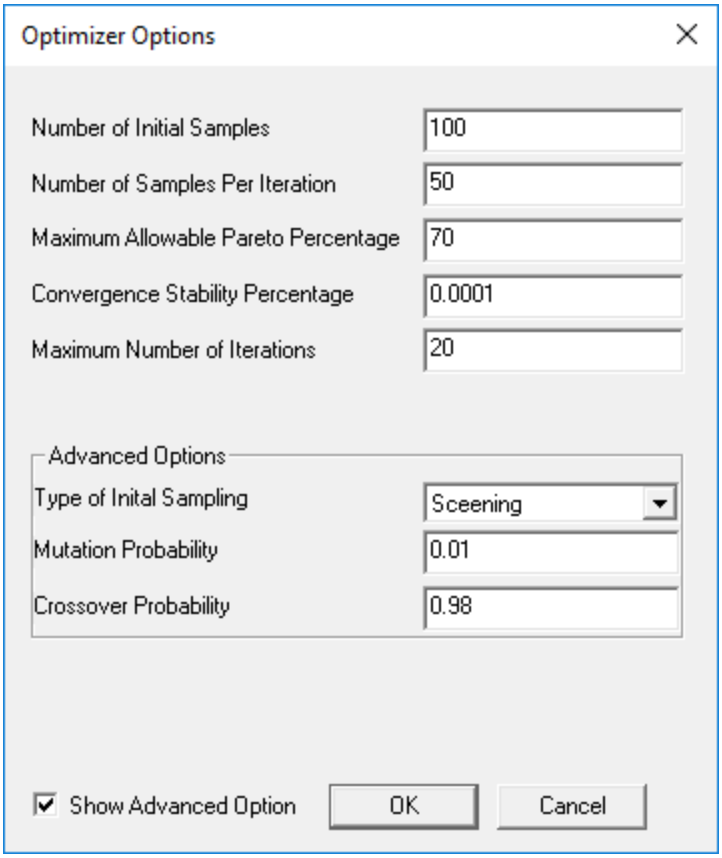
1. Set up the variables you want to optimize in the *Design Properties* dialog box. The variables must be swept in a Parametric setup.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **Maxwell** > **Optimetrics Analysis** > **Add Screening & Optimization**. The *Setup Optimization* dialog box appears.
3. Under the **Goals** tab, select the optimizer by selecting **Multi-Objective Genetic Algorithm (Random Search)** from the **Optimizer** drop-down menu.



4. Optionally, press the **Setup** button to open the **Optimizer Options window**.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Number of Initial Samples**: Initial number of samples to use. This number must be greater than the number of enabled input parameters. The minimum recommended number of initial samples is 10 times the number of enabled input parameters. The larger the initial sample set, the better your chances of finding the input parameter space that contains the best solutions.

  The number of enabled input parameters is also the minimum number of samples required to generate the Sensitivities chart. You can enter a minimum of 2 and a maximum of 10000. The default is 100.

  If you switch from the Screening method to the MOGA method, MOGA generates a new sample set. For the sake of consistency, enter the same number of initial samples as you used for the Screening method.

- **Number of Samples Per Iteration**: Number of samples to iterate and update with each iteration. This number must be greater than the number of enabled input parameters but less than or equal to the number of initial samples. The default is for a Direct Optimization system.

  You can enter a minimum of 2 and a maximum of 10000.

- **Maximum Allowable Pareto Percentage**: Convergence criterion. Percentage value that represents the ratio of the number of desired Pareto points to the number of samples per iteration. When this percentage is reached, the optimization is converged. For example, a value of 70 with Number of Samples Per Iteration set to 200 would mean that the optimization should stop once the resulting front of the MOGA optimization contains at least 140 points. Of course, the optimization stops before that if the maximum number of iterations is reached.
- If the **Maximum Allowable Pareto Percentage** is too low (below 30), the process can converge prematurely. If the value is too high (above 80), the process can converge slowly. The value of this property depends on the number of parameters and the nature of the design space itself. The default is 70. Using a value between 55 and 75 works best for most problems. For more information, see Convergence Criteria in MOGA-Based Multi-Objective Optimization.
- **Convergence Stability Percentage**: Convergence criterion. Percentage value that represents the stability of the population based on its mean and standard deviation. This criterion allows you to minimize the number of iterations performed while still reaching the desired level of stability. When the specified percentage is reached, the optimization is converged. The default percentage is 0.0001. To not take the convergence stability into account, set to 0. For more information, see Convergence Criteria in MOGA-Based Multi-Objective Optimization.
- **Maximum Number of Iterations**: Stop criterion. Maximum number of iterations that the algorithm is to execute. If this number is reached without the optimization having reached convergence, iterations stop. This also provides an idea of the maximum possible number of function evaluations that are needed for the full cycle,

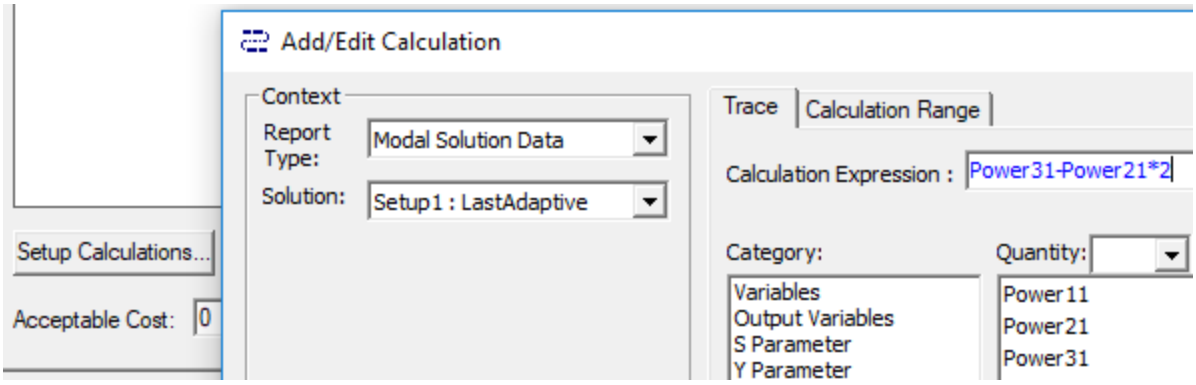
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

as well as the maximum possible time it can take to run the optimization. For example, the absolute maximum number of evaluations is given by

Number of Initial Samples + Number of Samples Per Iteration * (Maximum Number of Iterations - 1)

- **Type of Initial Sampling**: Advanced option for generating different kinds of sampling. If you do not have any parameter relationships defined, set to Screening (default) or Optimal Space-Filling. If you have parameter relationships defined, the initial sampling must be performed by the constrained sampling algorithms because parameter relationships constrain the sampling. For such cases, this property is automatically set to Constrained Sampling.
- **Mutation Probability**: Advanced option for specifying the probability of applying a mutation on a design configuration. The value must be between 0 and 1. A larger value indicates a more random algorithm. If the value is 1, the algorithm becomes a pure random search. A low probability of mutation (<0.2) is recommended. The default is 0.01. For more information on mutation, see MOGA Steps to Generate a New Population.
- **Crossover Probability**: Advanced option for specifying the probability with which parent solutions are recombined to generate offspring solutions. The value must be between 0 and 1. A smaller value indicates a more stable population and a faster (but less accurate) solution. If the value is 0, the parents are copied directly to the new population. A high probability of crossover (>0.9) is recommended. The default is 0.98.

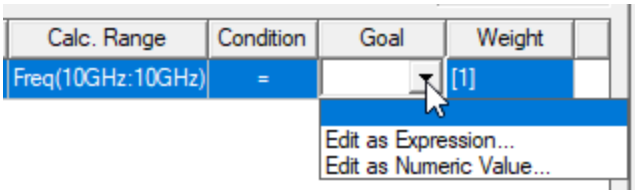
5. Add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.



When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the **Add/EditCalculation** dialog box.

6. In the Optimization setup, in the dropdown for the Goal column, select either **Edit as Expression...** or **Edit as Numeric Value...**.

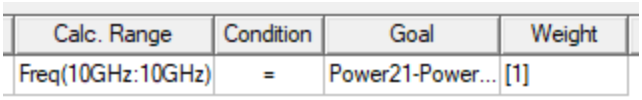
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This reopens the *Add/Edit Calculation* dialog box.

7. If you are satisfied with the expression or value displayed, click **Done** to close the dialog box.

   This enters the expression/value into the **Goal** column.



8. In the **Optimization** setup, if you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** drop-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors, which means that it is always less than zero. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.

9. Optionally, set the Acceptable Cost and Cost Function Noise.
10. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
11. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Optionally, set Linear constraints.
- Select the **View all columns** check box to see all columns, including hidden columns.

12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Nonlinear Programming by Quadratic Lagrangian (Gradient) Optimizer Setup

Following is the procedure for setting up an optimization analysis using the Nonlinear Programming by Quadratic Lagrangian (Gradient) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

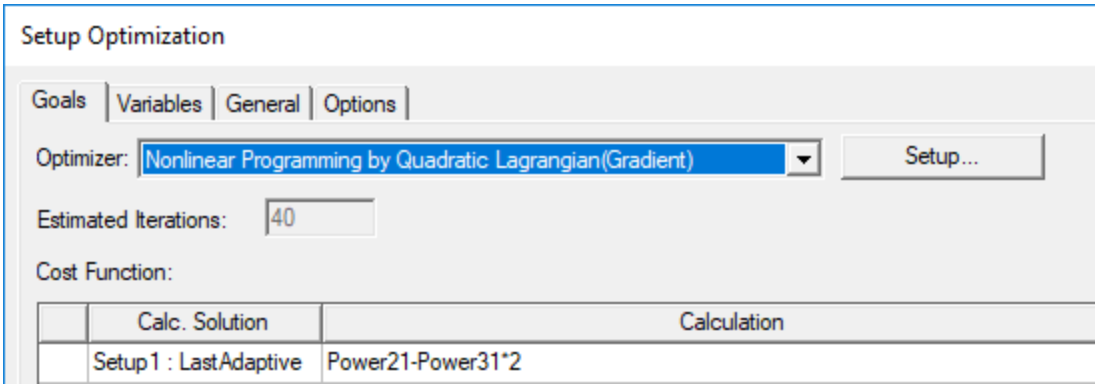
The NLPQL (Nonlinear Programming by Quadratic Lagrangian) method can be used for Direct Optimization systems. It allows you to generate a new sample set to provide a more refined approach than the Screening method. Available for continuous input parameters only, NLPQL can handle only one output parameter goal. Other output parameters can be defined as constraints. For more information, see Convergence Rate % and Initial Finite Difference Delta % in NLPQL and MISQP and Nonlinear Programming by Quadratic Lagrangian (NLPQL).

To generate samples and perform an NLPQL optimization:

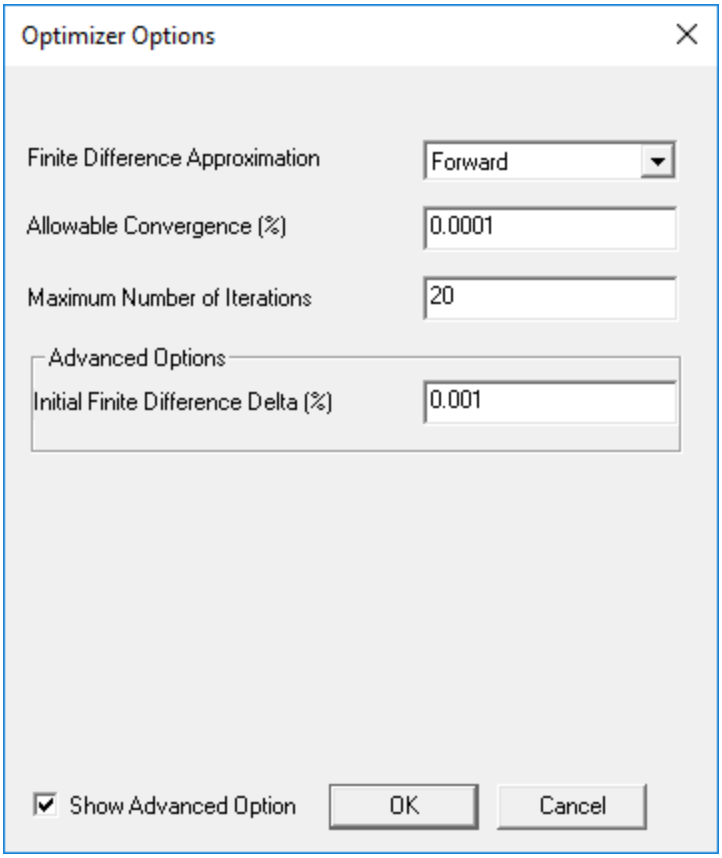
1. Set up the variables you want to optimize in the *Design Properties* dialog box. The variables must be swept in a Parametric setup.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **Maxwell** > **Optimetrics Analysis** > **Add Screening & Optimization**. The *Setup Optimization* dialog box appears.
3. Under the **Goals** tab, select the optimizer by selecting **Nonlinear Programming by Quadratic Lagrangian (Gradient)** from the **Optimizer** drop-down menu.



4. Optionally, press the **Setup** button to open the *Optimizer Options window*.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Finite Difference Approximation**: When analytical gradients are not available, NLPQL approximates them numerically. This property allows you to specify the method of approximating the gradient of the objective function. Choices are:
  - **Central**: Increases the accuracy of the gradient calculations by sampling from both sides of the sample point but increases the number of design point evaluations by 50%. This method makes use of the initial point, as well as the forward point and rear point.
  - **Forward**: Uses fewer design point evaluations but decreases the accuracy of the gradient calculations. This method makes use of only two design points, the initial point and forward point, to calculate the slope forward. This is the default method for new Direct Optimization systems.
- **Maximum Number of Iterations**: Stop criterion. Maximum number of iterations that the algorithm is to execute. If convergence happens before this number is reached, the iterations stop. This also provides an idea of the maximum possible number of function evaluations that are needed for the full cycle. For NLPQL, the number of evaluations can be approximated according to the Finite Difference Approximation gradient calculation method, as follows:

  For **Central**: number of iterations * (2*number of inputs +1)

  For **Forward**: number of iterations * (number of inputs+1)

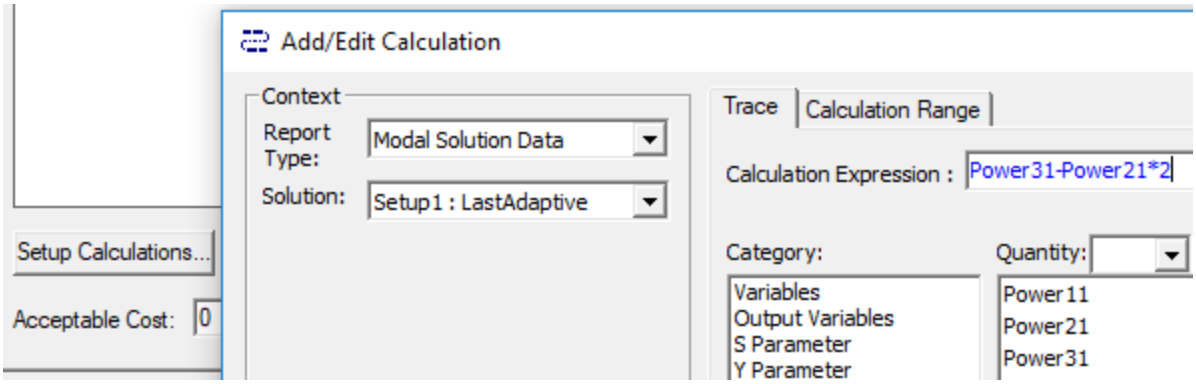
- **Allowable Convergence (%)**: Stop criterion. Tolerance to which the Karush-Kuhn-Tucker (KKT) optimality criterion is generated during the NLPQL process. A smaller value indicates more convergence iterations and a more accurate (but slower) solution. A larger value indicates fewer convergence iterations and a less accurate (but faster) solution. For a Direct Optimization system, the default percentage value is 0.1. The maximum percentage value is 100. These values are consistent across all problem types because the inputs, outputs, and gradients are scaled during the NLPQL solution.
- **Initial Finite Difference Delta (%)**: Advanced option for specifying the relative variation used to perturb the current point to compute gradients. Used in conjunction with Allowable Convergence (%) to ensure that the delta in NLPQL's calculation of finite differences is large enough to be seen above the noise in the simulation problem. This wider sampling produces results that are more clearly differentiated so that the difference is less affected by solution noise and the gradient direction is clearer. The value should be larger than both the value for Initial Finite Difference Delta (%) and the noise magnitude of the model. However, smaller values produce more accurate results, so set Initial Finite Difference Delta (%) only as high as necessary to be seen above simulation noise.

The default percentage value is 0.001. The minimum is 0.0001, and the maximum is 1. For parameters with Allowed Values set to **Manufacturable Values** or Snap to Grid, the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

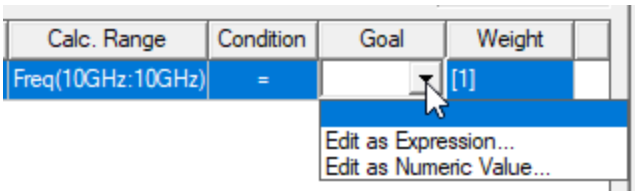
value for Initial Finite Difference Delta (%) is ignored. In such cases, the closest allowed value is used to determine the finite difference delta.

5. Add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.



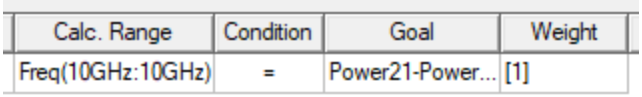
When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the *Add/EditCalculation* dialog box.

6. In the Optimization setup, in the drop-down menu for the Goal column, select either Edit as Expression or Edit as Numeric Value...



This reopens the *Add/Edit Calculation* dialog box.

7. If you are satisfied with the expression or value displayed, click **Done** to close the dialog box. This enters the expression/value to the **Goal** column.



8. In the **Optimization** setup, if you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** drop-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A norm is a function that assigns a positive value to the cost function.

For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors, which means that it is always less than zero. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.

9. Optionally, set the Acceptable Cost and Cost Function Noise.
10. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
11. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.
    - Optionally, set Linear constraints.
    - Select the **View all columns** check box to see all columns, including hidden columns.
12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mixed-Integer Sequential Quadratic Programming (Gradient and Discrete) Optimizer Setup

Mixed-Integer Sequential Quadratic Programming (MISQP) is a mathematical optimization algorithm as developed by Oliver Exler, Thomas Lehmann and Klaus Schittkowski (NLPQL). This method solves Mixed-Integer Nonlinear Programming (MINLP) of the form:

Minimize:

$$f(x,y)$$

Subject to

$$g_j(x,y)=0,\ j=1,\ldots,m_e,$$
$$g_j(x,y)\geq 0,\ j=m_e+1,\ldots,m$$

where

$$x\in\mathbb{R}^{n_c}, y\in\mathbb{N}^{n_i}$$
$$x_l\leq x\leq x_u$$
$$y_l\leq y\leq y_u$$

The symbols and denote the vectors of the continuous and integer variables, respectively. It is assumed that problem functions and are continuously differentiable subject to all . It is not assumed that integer variables can be relaxed. In other words, problem functions are evaluated only at integer points and never at any fractional values in between.

MISQP solves MINLP by a modified sequential quadratic programming (SQP) method. After linearizing constraints and constructing a quadratic approximation of the Lagrangian function, mixed-integer quadratic programs are successively generated and solved by an efficient branch-and-cut method. The algorithm is stabilized by a trust region method as originally proposed by Yuan for continuous programs. Second-order corrections are retained. The Hessian of the Lagrangian function is approximated by BFGS updates subject to the continuous and integer variables. MISQP is able to solve also non-convex nonlinear mixed-integer programs.

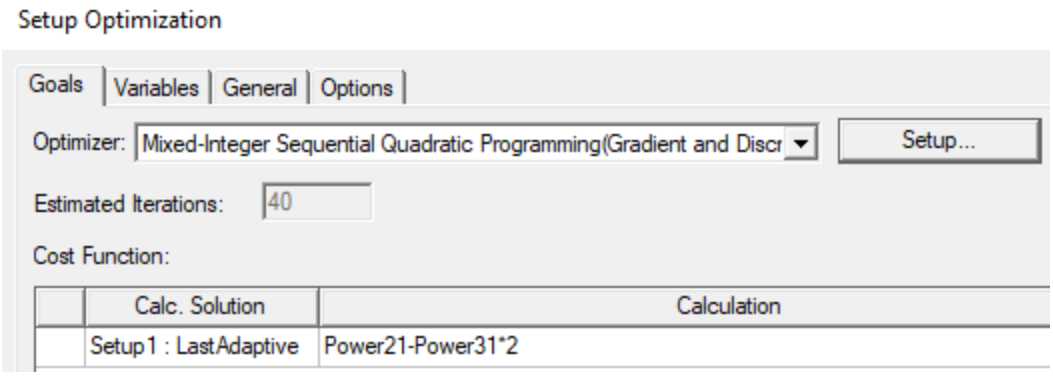
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Following is the procedure for setting up an optimization analysis using the Mixed-Integer Sequential Quadratic Programming optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

The Mixed-Integer Sequential Quadratic Programming (Nonlinear Programming by Quadratic Lagrangian) method (MISQP)can be used for Direct Optimization systems. It allows you to generate a new sample set to provide a more refined approach than the Screening method. MISQP is available for both continuous and discrete input parameters, which is why mixed is in its name. MISQP can handle only one output parameter goal.. Other output parameters can be defined as constraints. For more information, see Convergence Rate % and Initial Finite Difference Delta % in NLPQL and MISQP and Nonlinear Programming by Quadratic Lagrangian (NLPQL).

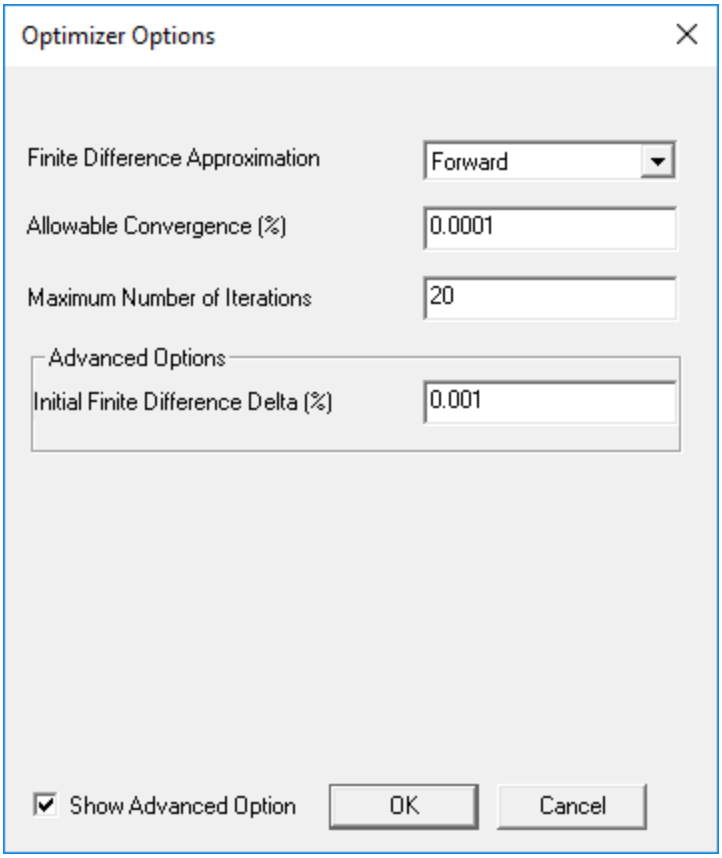
To generate samples and perform an NLPQL optimization:

1. Set up the variables you want to optimize in the *Design Properties* dialog box. The variables must be swept in a Parametric setup.
2. Click **Maxwell** > **Optimetrics Analysis** > **Add Screening & Optimization**. The *Setup Optimization* dialog box appears.
3. Under the **Goals** tab, select the optimizer by selecting **Mixed-Integer Sequential Quadratic Programming (Gradient and Discrete)** from the **Optimizer** drop-down menu.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Optionally, press the **Setup** button to open the **Optimizer Options window**.



- **Finite Difference Approximation**: When analytical gradients are not available, MISQP approximates them numerically. This property allows you to specify the method of approximating the gradient of the objective function. Choices are:
  - **Central**: Increases the accuracy of the gradient calculations by sampling from both sides of the sample point but increases the number of design point evaluations by 50%. This method makes use of the initial point, as well as the forward point and rear point.
  - **Forward**: Uses fewer design point evaluations but decreases the accuracy of the gradient calculations. This method makes use of only two design points, the initial point and forward point, to calculate the slope forward. This is the default method for new Direct Optimization systems.
- **Maximum Number of Iterations**: Stop criterion. Maximum number of iterations that the algorithm is to execute. If convergence happens before this number is reached, the iterations stop. This also provides an idea of the maximum possible number of function evaluations that are needed for the full cycle. For MISQP, the number of evaluations can be approximated according to the Finite Difference Approximation gradient calculation method, as follows:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

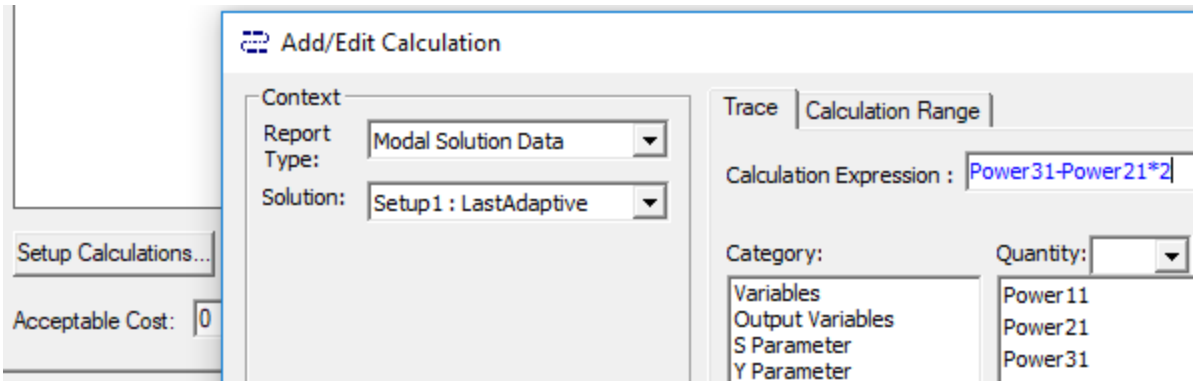
For **Central**: number of iterations * (2*number of inputs +1)

For **Forward**: number of iterations * (number of inputs+1)

- **Allowable Convergence (%)**: Stop criterion. Tolerance to which the Karush-Kuhn-Tucker (KKT) optimality criterion is generated during the MISQP process. A smaller value indicates more convergence iterations and a more accurate (but slower) solution. A larger value indicates fewer convergence iterations and a less accurate (but faster) solution. For a Direct Optimization system, the default percentage value is 0.0001. The maximum percentage value is 100. These values are consistent across all problem types because the inputs, outputs, and gradients are scaled during the MISQP solution.
- **Initial Finite Difference Delta (%)**: Advanced option for specifying the relative variation used to perturb the current point to compute gradients. Used in conjunction with Allowable Convergence (%) to ensure that the delta in MISQP's calculation of finite differences is large enough to be seen above the noise in the simulation problem. This wider sampling produces results that are more clearly differentiated so that the difference is less affected by solution noise and the gradient direction is clearer. The value should be larger than both the value for Initial Finite Difference Delta (%) and the noise magnitude of the model. However, smaller values produce more accurate results, so set Initial Finite Difference Delta (%) only as high as necessary to be seen above simulation noise.

The default percentage value is 0.001. The minimum is 0.0001, and the maximum is 1. For parameters with Allowed Values set to **Manufacturable Values** or Snap to Grid, the value for Initial Finite Difference Delta (%) is ignored. In such cases, the closest allowed value is used to determine the finite difference delta.

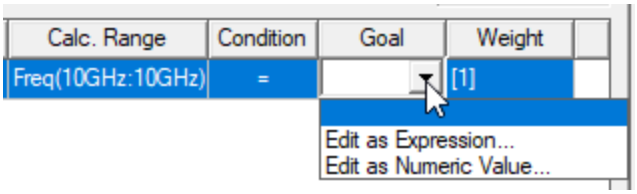
5. Add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.



When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the *Add/EditCalculation* dialog box.

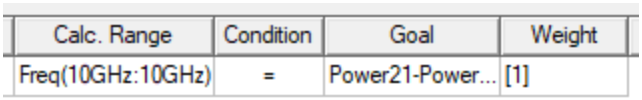
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. In the Optimization setup, in the drop-down menu, for the Goal column, select either Edit as Expression or Edit as Numeric Value...



   This reopens the *Add/Edit Calculation* dialog box.

7. If you are satisfied with the expression or value displayed, click **Done** to close the dialog box. This enters the expression/value to the **Goal** column.



8. In the **Optimization** setup, if you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** drop-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors, which means that it is always less than zero. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.

9. Optionally, set the Acceptable Cost and Cost Function Noise.
10. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
11. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Optionally, set Linear constraints.
- Select the **View all columns** check box to see all columns, including hidden columns.

12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Adaptive Multiple-Objective (Random Search) Optimizer Setup

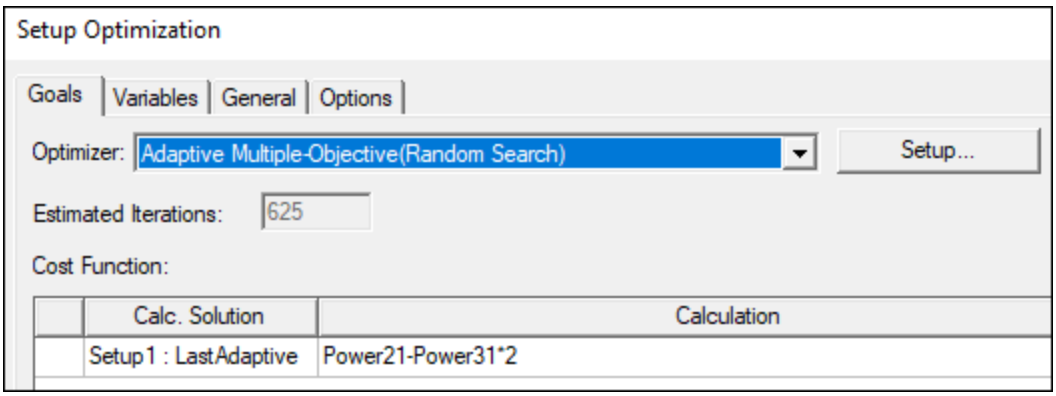
Following is the procedure for setting up an optimization analysis using the Adaptive Multiple Objective (Random Search) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

The Adaptive Multiple Objective (Kriging + MOGA) is an iterative algorithm that allows you to either generate a new sample set or use an existing set, providing a more refined approach than the Screening method. It uses the same general approach as MOGA, but applies the Kriging error predictor to reduce the number of evaluations needed to find the global optimum. The Adaptive Multiple-Objective method is available only for continuous input parameters, including those with manufacturable values. It can handle multiple objectives and multiple constraints. For more information, see Adaptive Multiple-Objective Optimization.

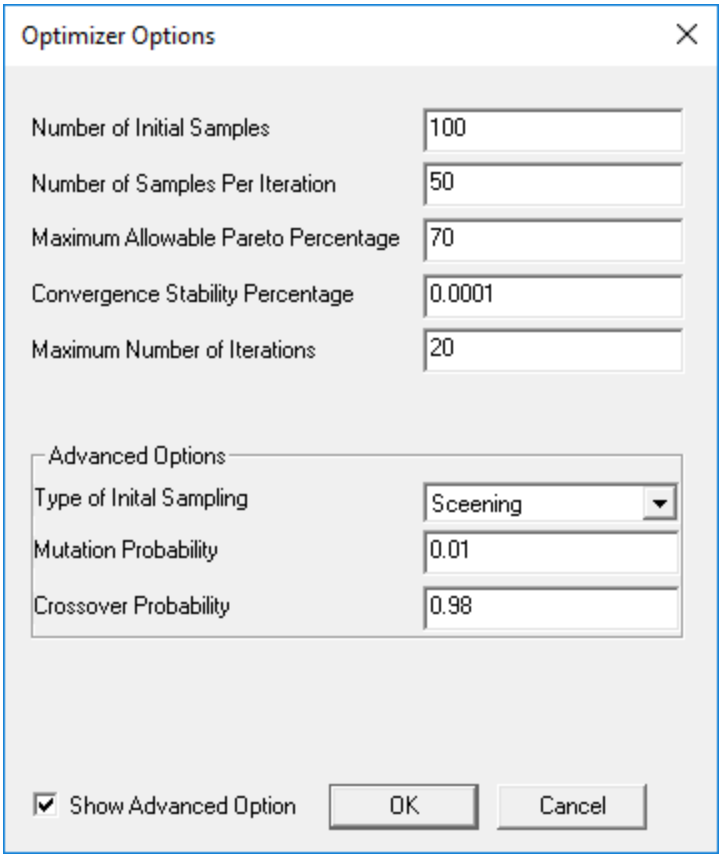
1. Set up the variables you want to optimize in the *Design Properties* dialog box. The variables must be swept in a Parametric setup.
2. Click **Maxwell** > **Optimetrics Analysis** > **Add Screening & Optimization**. The *Setup Optimization* dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Under the **Goals** tab, select the optimizer by selecting **Adaptive Multiple Objective (Random-search)** from the **Optimizer** drop-down menu.



4. Optionally, press the **Setup** button to open the *Optimizer Options* window.



- **Number of Initial Samples**: Initial number of samples to use. This number must be greater than the number of enabled input parameters. The minimum recommended number of initial samples is 10 times the number of enabled input parameters. The

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

larger the initial sample set, the better your chances of finding the input parameter space that contains the best solutions.

The number of enabled input parameters is also the minimum number of samples required to generate the Sensitivities chart. You can enter a minimum of 2 and a maximum of 10000. The default is 100.

If you switch from the Screening method to the MOGA method, MOGA generates a new sample set. For the sake of consistency, enter the same number of initial samples as you used for the Screening method.

- **Number of Samples Per Iteration**: Number of samples to iterate and update with each iteration. This number must be greater than the number of enabled input parameters but less than or equal to the number of initial samples. The default is for a Direct Optimization system.

  You can enter a minimum of 2 and a maximum of 10000.

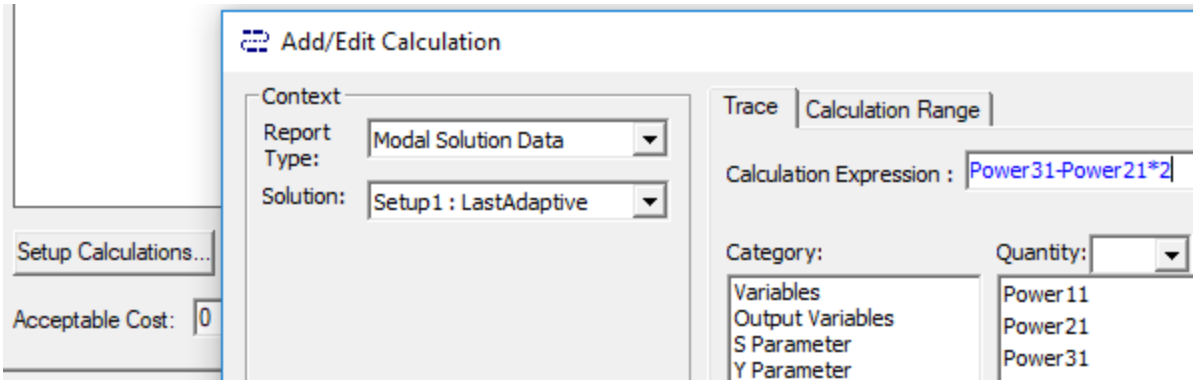
- **Maximum Allowable Pareto Percentage**: Convergence criterion. Percentage value that represents the ratio of the number of desired Pareto points to the number of samples per iteration. When this percentage is reached, the optimization is converged. For example, a value of 70 with Number of Samples Per Iteration set to 200 would mean that the optimization should stop once the resulting front of the MOGA optimization contains at least 140 points. Of course, the optimization stops before that if the maximum number of iterations is reached.
- If the **Maximum Allowable Pareto Percentage** is too low (below 30), the process can converge prematurely. If the value is too high (above 80), the process can converge slowly. The value of this property depends on the number of parameters and the nature of the design space itself. The default is 70. Using a value between 55 and 75 works best for most problems. For more information, see Convergence Criteria in MOGA-Based Multi-Objective Optimization.
- **Convergence Stability Percentage**: Convergence criterion. Percentage value that represents the stability of the population based on its mean and standard deviation. This criterion allows you to minimize the number of iterations performed while still reaching the desired level of stability. When the specified percentage is reached, the optimization is converged. The default percentage is 0.0001. To not take the convergence stability into account, set to 0. For more information, see Convergence Criteria in MOGA-Based Multi-Objective Optimization.
- **Maximum Number of Iterations**: Stop criterion. Maximum number of iterations that the algorithm is to execute. If this number is reached without the optimization having reached convergence, iterations stop. This also provides an idea of the maximum possible number of function evaluations that are needed for the full cycle, as well as the maximum possible time it can take to run the optimization. For example, the absolute maximum number of evaluations is given by:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of Initial Samples + Number of Samples Per Iteration * (Maximum Number of Iterations - 1)

- **Type of Initial Sampling**: Advanced option for generating different kinds of sampling. If you do not have any parameter relationships defined, set to Screening (default) or Optimal Space-Filling. If you have parameter relationships defined, the initial sampling must be performed by the constrained sampling algorithms because parameter relationships constrain the sampling. For such cases, this property is automatically set to Constrained Sampling.
- **Mutation Probability**: Advanced option for specifying the probability of applying a mutation on a design configuration. The value must be between 0 and 1. A larger value indicates a more random algorithm. If the value is 1, the algorithm becomes a pure random search. A low probability of mutation (<0.2) is recommended. The default is 0.01. For more information on mutation, see MOGA Steps to Generate a New Population.
- **Crossover Probability**: Advanced option for specifying the probability with which parent solutions are recombined to generate offspring solutions. The value must be between 0 and 1. A smaller value indicates a more stable population and a faster (but less accurate) solution. If the value is 0, the parents are copied directly to the new population. A high probability of crossover (>0.9) is recommended. The default is 0.98.

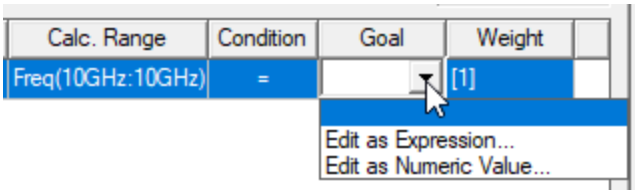
5. Add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.



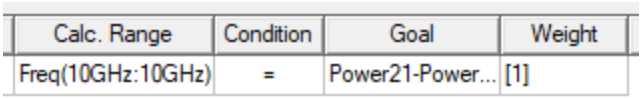
When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the *Add/EditCalculation* dialog box.

6. In the Optimization setup, in the Goal column drop-down menu, select either **Edit as Expression...** or **Edit as Numeric Value...**. This reopens the *Add/Edit Calculation* dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



7. If you are satisfied with the expression or value displayed, click **Done** to close the dialog box. This enters the expression/value into the **Goal** column.



8. In the **Optimization** setup, if you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The *Cost Function Norm Type* drop-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors, which means that it is always less than zero. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.
9. Optionally, set the Acceptable Cost and Cost Function Noise.
10. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
11. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.
    - Optionally, set Linear constraints.
    - Select the **View all columns** check box to see all columns, including hidden columns.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Adaptive Single-Objective (Gradient) Optimizer Setup

Following is the procedure for setting up an optimization analysis using the Adaptive Single-Objective (OSF + Kriging + MISQP) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

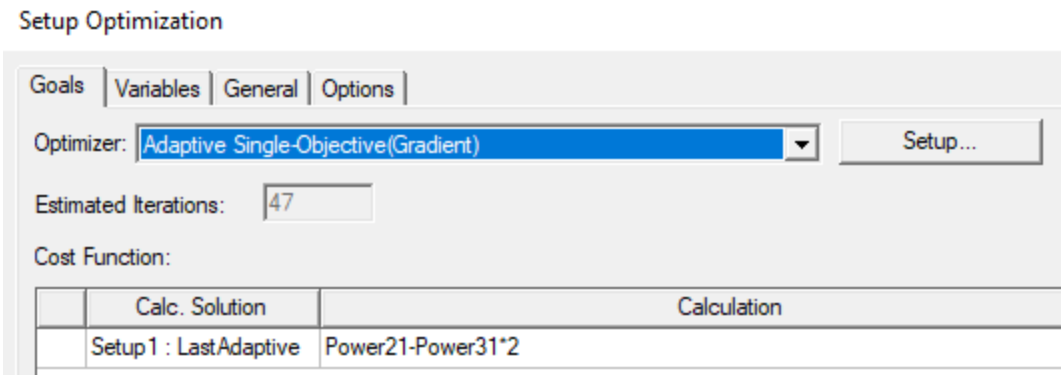
This gradient-based method employs automatic intelligent refinement to provide the global optima. It requires a minimum number of design points to build the Kriging response surface, but, in general, this method reduces the number of design points necessary for the optimization. Failed design points are treated as inequality constraints, making it fault-tolerant.

The Adaptive Single-Objective method is available for input parameters that are continuous, including those with manufacturable values. It can handle only one output parameter goal, although other output parameters can be defined as constraints. It does not support the use of parameter relationships in the optimization domain. For more information, see Adaptive Single-Objective Optimization (ASO). It requires advanced options. Ensure that the Show Advanced Options check box is selected.

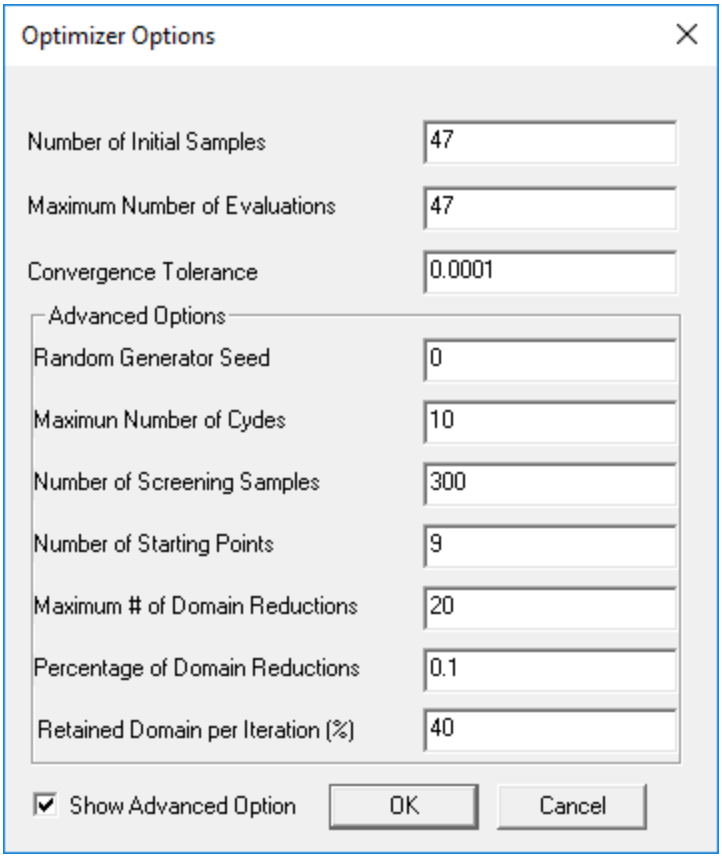
1. Set up the variables you want to optimize in the *Design Properties* dialog box. The variables must be swept in a Parametric setup.
2. Click **Maxwell** > **Optimetrics Analysis** > **Add Screening & Optimization**. The *Setup Optimization* dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Under the **Goals** tab, choose the optimizer by selecting **Adaptive Single Objective (Gradient)** from the **Optimizer** drop-down menu.



4. Optionally, press the **Setup** button to open the *Optimizer Options* window and check **Advanced Options**.



- **Number of Initial Samples**: Number of samples generated for the initial Kriging and after all domain reductions for the construction of the next Kriging. You can

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

enter a minimum of (NbInp+1)*(NbInp+2)/2 (also the minimum number of OSF samples required for the Kriging construction) or a maximum of 10,000. The default is (NbInp+1)*(NbInp+2)/2.

Because of the Adaptive Single-Objective workflow (in which a new OSF sample set is generated after each domain reduction), increasing the number of OSF samples does not necessarily improve the quality of the results and significantly increases the number of evaluations

- **Maximum Number of Evaluations**: Stop criterion. Maximum number of evaluations (design points) that the algorithm is to calculate. If convergence occurs before this number is reached, evaluations stop. This value also provides an idea of the maximum possible time it takes to run the optimization. The default is 20*(NbInp +1).
- **Convergence Tolerance**: Stop criterion. Minimum allowable gap between the values of two successive candidates. If the difference between two successive candidates is smaller than the value for Convergence Tolerance multiplied by the maximum variation of the parameter, the algorithm is stopped. A smaller value indicates more convergence iterations and a more accurate (but slower) solution. A larger value indicates fewer convergence iterations and a less accurate (but faster) solution. The default is 1E-06.
- **Random Generator Seed**: The value for initializing the random number generator invoked internally by OSF. The value must be a positive integer. This property allows you to generate different samplings by changing the value or to regenerate the same sampling by keeping the same value. The default is 0.
- **Maximum Number of Cycles**: Number of optimization loops that the algorithm needs, which in turns determines the discrepancy of the OSF. The optimization is essentially combinatorial, so a large number of cycles slows down the process. However, this makes the discrepancy of the OSF smaller. The value must be greater than 0. For practical purposes, 10 cycles is usually good for up to 20 variables. The default is 10.
- **Number of Screening Samples**: Number of samples for the screening generation on the current Kriging. This value is used to create the next Kriging (based on error prediction) and verified candidates.

  You can enter a minimum of (NbInp+1)*(NbInp+2)/2 (also the minimum number of OSF samples required for the Kriging construction) or a maximum of 10,000. The default is 100*NbInp for a Direct Optimization system. There is no default for a Response Surface Optimization system.

  The larger the screening sample set, the better the chances of finding good verified points. However, too many points can result in a divergence of the Kriging.

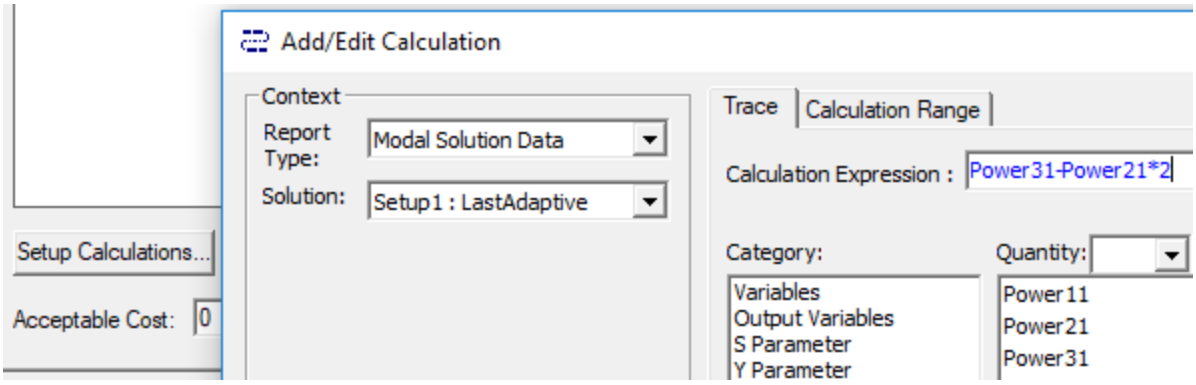
- **Number of Starting Points**: Determines the number of local optima to explore. The larger the number of starting points, the more local optima explored. In the case

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of a linear surface, for example, it is not necessary to use many points. This value must be less than the value for **Number of Screening Samples** because these samples are selected in this sample. The default is the value for **Number of Initial Samples**.

- **Maximum Number of Domain Reductions**: Stop criterion. Maximum number of domain reductions for input variation. (No information is known about the size of the reduction beforehand.) The default is 20.
- **Percentage of Domain Reductions**: Stop criterion. Minimum size of the current domain according to the initial domain. For example, with one input ranging between 0 and 100, the domain size is equal to 100. The percentage of domain reduction is 1%, so the current working domain size cannot be less than 1 (such as an input ranging between 5 and 6). The default is 0.1.
- **Retained Domain per Iteration (%)**: Advanced option that allows you to specify the minimum percentage of the domain you want to keep after a domain reduction. The percentage value must be between 10 and 90. A larger value indicates less domain reduction, which implies better exploration but a slower solution. A smaller value indicates a faster and more accurate solution, with the risk of it being a local one. The default percentage value is 40.

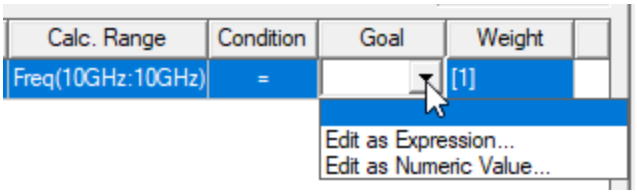
5. Add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.



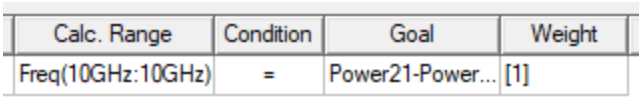
When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the **Add/EditCalculation** dialog box.

6. In the Optimization setup, in the Goal column drop-down menu, select either **Edit as Expression...** or **Edit as Numeric Value...**. This reopens the *Add/Edit Calculation* dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



7. If you are satisfied with the expression or value displayed, click **Done** to close the dialog box. This enters the expression/value into the **Goal** column.



8. In the **Optimization** setup, if you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** drop-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors, which means that it is always less than zero. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.
9. Optionally, set the Acceptable Cost and Cost Function Noise.
10. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
11. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.
    - Optionally, set Linear constraints.
    - Select the **View all columns** check box to see all columns, including hidden columns.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Setting the Maximum Iterations for an Optimization Analysis

The **Max. No. of Iterations** value is the maximum number of design variations that you want Optimetrics to solve during an optimization when using the **Sequential Nonlinear Programming (Gradient)**, **Sequential Mixed Integer NonLinear Programming**, **Quasi Newton (Gradient)**, or **Pattern Search (Search-based) Optimizer**. This value is a stopping criterion; if the maximum number of iterations has been completed, the optimization analysis stops. If the maximum number of iterations has not been completed, the optimization continues by performing another iteration, that is, by solving another design variation.

If the maximum number of iterations has not been reached, the optimizer performs iterations until the acceptable cost function is reached or until the optimizer cannot proceed as a result of other optimization setup constraints, such as when it searches for a variable value with a step size smaller than the minimum step size.

> **Note:**
>
> The **Genetic Algorithm** optimizer does not use the **Max. No. of Iterations** criteria.

To set the maximum number of iterations for an optimization analysis:

- Under the **Goals** tab of the *Setup Optimization* dialog box, type a value in the **Max. No. of Iterations** text box.

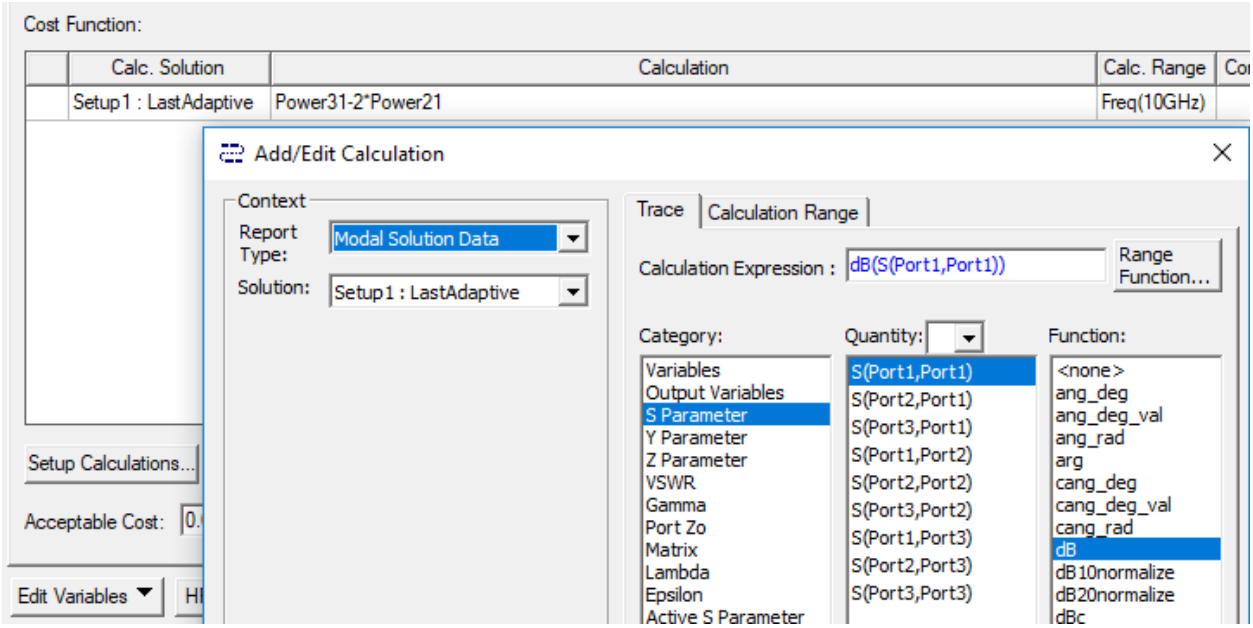
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cost Function

Optimetrics manipulates the model's design variable values to find the minimum location of the cost function; therefore, you should define the cost function so that a minimum location is also the optimum location. For example, if you vary a design to find the maximum transmission from *Wave Port 1* to *Wave Port 2* (S_{21}=>1), define the cost function to be -mag(S(WavePort2,WavePort1)).

When using the Quasi Newton optimizer, which is appropriate for designs that are not sensitive to noise, the best cost function is a smooth, second-order function that can be approximated well by quadratics in the vicinity of the minimum; the slope of the cost function should decrease as Optimetrics approaches the optimum value. The preferred cost function takes values between 0 and 1. In practice, most functions that are smooth around the minimum are acceptable as cost functions. Most importantly, the cost function should not have a sharp dip or pole at the minimum. A well designed cost function can significantly reduce the optimization process time.

The cost function is defined in the **Setup Optimization** dialog box or the **Design of Experiments** setup when you set up an optimization analysis. If you know the exact syntax of the solution quantity on which you want to base the cost function, you can type it directly in the **Calculation** text box. You can also use **Setup Calculations** to add a solution quantity via the *Add/Edit Calculation* dialog box, or to create an output variable that represents the solution quantity in the Output Variables dialog box.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Acceptable Cost

The acceptable cost is the value of the cost function at which the optimization process should stop; otherwise known as the *stopping criterion*.The cost function value must be equal to or below the acceptable cost value for the optimization analysis to stop. The acceptable cost may be a negative value.

Cost Function Noise

The numerical calculation of the electromagnetic field introduces various sources of noise to the cost function, particularly because of changes in the finite element mesh. You must provide the optimizer with an estimate of the noise. The noise indicates whether a change during the solution process is significant enough to support achievement of the cost function.

For example, if the cost function, c, is

$c = 10000 \cdot |L_{11}|^2$

where $|L_{11}|$ is the magnitude of the inductance, at the minimum, $|L_{11}|$ is expected to be very small, $|L_{11}| \approx 0$.

From the solution setup, the error in $|L_{11}|$ is expected to be $E_{L11} \approx 0.01$. The perturbed cost function is therefore

$c_{perturbed} = 10000 \cdot (|L_{11}|_{min} + E_{L11})^2$

Near the minimum, the error in the cost function E_c is given by

$E_c = c_{perturbed} - c_{min} = 10000 \cdot (0.0 + 0.01)^2 - (10000 \cdot 0.0) = 1.0$

Therefore, the cost function noise would be 1.0.

Adding a Cost Function

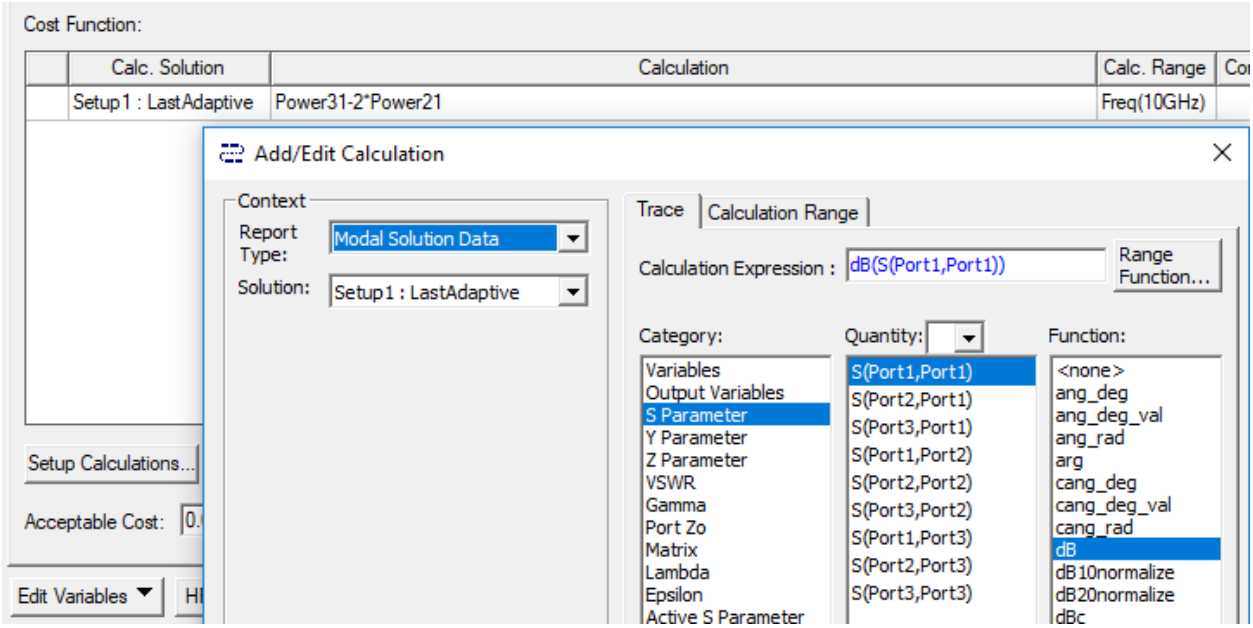
A cost function can include one or more goals for an optimization analysis. Optimetrics manipulates the model's design variable values to fulfill the cost function. The optimization will stop when the solution quantity meets the acceptable cost criterion.

Following is the general procedure for adding a cost function with a single goal:

1. Under the **Goals** tab of the *Setup Optimization* or *Design of Experiments* dialog box, click **Setup Calculations...**

   The Add/Edit Calculation dialog box is displayed.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. In the *Add/Edit Calculation* dialog box, follow these general steps to set up a cost function.
   a. Set the **Context** for the calculation.
   b. Choose the **Category** of available data type depending upon the Solution type of the design being optimized.
   c. Select the **Quantity** to add to the **Calculated Expression** field. Available quantities depend upon the **Category** selection.
   d. You may optionally make a selection from the function list to apply to the calculated expression.

      Note:

      Because Optimetrics works in SI values, you should use ang_deg_val or cang_deg_val functions which return unitless numbers in degrees rather than ang_deg or cang_deg, which return angular values in degree units but evaluate in radians in Optimetrics expressions. For example, you could Optimize successfully by changing the expression from abs(**cang_deg**(S(Port2,Port1))) to abs(**cang_deg_val**(S(Port2,Port1))). See the table of available functions and definitions.

   e. When the **Calculation Expression** has the desired equation, click **Add Calculation** to add the expression to the cost function table.
   f. Repeat to add additional calculations to the cost function or click **Done** to exit the **Add/Edit Calculation** dialog box and return to **Setup Optimization**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. To modify the **Solution** on which the calculation is based, click in the **Solution** column and select the solution from which the cost function is to be extracted from the drop-down menu.
4. To edit the calculation on which to base the cost function goal, select **Edit** from the drop-down menu.
5. In the **Condition** text box, click one of the following conditions from the drop-down menu:

<=	Less than or equal to
=	Equal to
>=	Greater than or equal to
Minimize	Reduce the cost function to a minimum value
Maximize	Identify a maximized condition

6. In the **Goal** text box, type the value of the solution quantity that you want to be achieved during the optimization analysis. If the solution quantity is a complex calculation, the goal value must be complex; two goal values must be specified. The **Minimize** and **Maximize** options do not require you to specify a **Goal** value.

   When Minimize is used as an optimization condition, the value of calculation is used as the cost (there is no target value to compare to). For maximize, the negative of calculation value is used as cost.

7. Optionally, if you have multiple goals and want to assign higher or lower priority to a goal, type a different value for the goal's weight in the **Weight** text box. The goal with the greater weight is given more importance. If the goal is a complex value, the weight value must be complex; two weight values must be specified. The weight value cannot be variable dependent.

   > **Note:**
   >
   > Click the **Edit Goal/Weight** button to open the *Edit Goal Value/Weight* dialog box, where you can modify weights for all goals simultaneously and set the **Goal Values** to expressions.

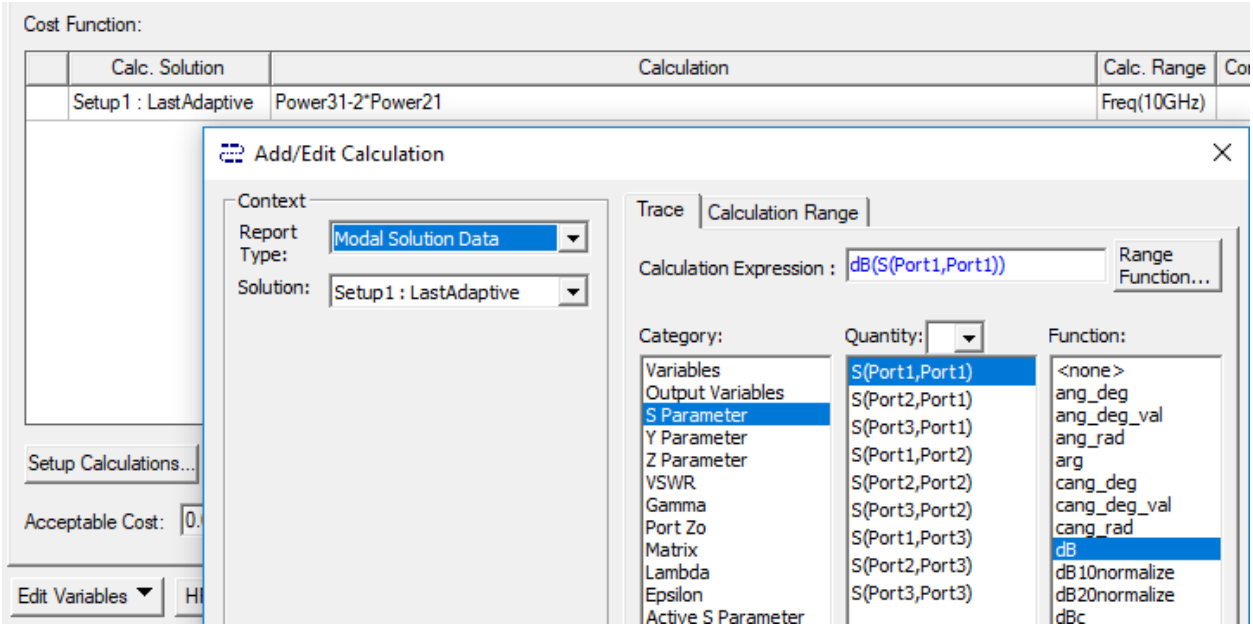
8. Specify other options (such as acceptable cost, noise, and number of passes), and then click **OK**.

The optimization stops when the solution quantity meets the acceptable cost criterion.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adding or Editing a Cost Function Calculation

The **Add/Edit Calculation** dialog box allows you to define the mathematical equation for one or multiple cost functions. It represents the calculation to be performed on the optimization variables to compare to the goal values.



To set up a calculation for a cost function:

1. In the **Context** section of the dialog box:
    - Select the **Report Type** with a drop-down selection list containing the available types for this design.
    - Select the **Solution** from the drop-down selection list. This lists the available setups and sweeps. As a minimum, the **LastAdaptive** solution is available.
    - Select the Geometry from the drop-down selection list or select none (the default). This modifies the list of quantities available to the ones that apply to the specific geometry.
2. The **Output Variables** button opens the Output Variables dialog box allowing you to create special output variables to be used in the cost function.
3. The **Calculated Expression** field in the **Trace** tab is used to enter the equation to be used for the cost function. To enter an expression, you may type it directly into the field or use the **Category**, **Quantity**, and **Function** lists as follows:
    - Select the **Category**, these depend on the Solution type and the design. This lets you specify the category of information to be used in the cost function.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select a **Quantity** from the list. Available quantities depend upon the Solution type, as well as the Geometry and Category selection. Selecting a Quantity automatically enters it into the Calculated Expression field.
- Select a **Function** to apply to the value in the calculated expression.

  > **Note:**
  >
  > Because Optimetrics works in SI values, you should use ang_deg_val or cang_deg_val functions which return unitless numbers in degrees rather than ang_deg or cang_deg, which return angular values in degree units but evaluate in radians in Optimetrics expressions. For example, you could Optimize successfully by changing the expression from abs(**cang_deg**(S(Port2,Port1))) to abs(**cang_deg_val**(S(Port2,Port1))). See the table of available functions and definitions.

- For swept variables, the Range Function button opens the **Set Range Function** dialog to apply functions to the expression that apply over the sweep range.

4. The **Calculation Range** tab applies to swept variables and allows you to specify the range of the sweep over which to apply the calculation.
5. When the desired **Calculated Expression** has been obtained, click **Add Calculation** to add the entry to the cost function table. You may add multiple entries to the table simply by changing the **Calculated Expression** and using the **Add Calculation** button.
6. To update or edit a selected cost function, enter the desired Calculated Expression and click the **Update Calculation** button.
7. Click **Done** to return to the **Setup Calculations** dialog box.

Specifying a Solution Quantity for a Cost Function Goal

When setting up a cost function, you must identify the solution quantity on which to base each goal. Solution quantities are specified by mathematical expressions that are composed of basic quantities, such as matrix parameters, and output variables.

1. Add a row (a goal) to the cost function table:
   a. Under the **Goals** tab of the *Setup Optimization* dialog box, click **Add**.

      A new row is added to the *Cost Function* table.

   b. In the **Solution** column, click the solution from which the cost function is to be extracted.
2. In the **Solution** text box, click the solution from which the solution quantity is to be extracted.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In the **Calculation** text box, specify the solution quantity in one of the following ways:
   - If you know the syntax of the mathematical expression or the output variable's name, type it in the **Calculation** text box.
   - If you want to create an output variable that represents the solution quantity, do the following:
     a. Click **Edit Calculation**.

        The *Output Variables* dialog box appears.

     b. Add the expression you want to evaluate, click **Done**.
     c. Click **Done** to close the *Output Variables* dialog box.

        In the *Setup Optimization* dialog box, the most recently created output variable appears in the *Calculation* text box.

     d. To specify a different defined output variable, click the **Calculation** text box. It becomes a drop-down menu that displays all of the defined output variables. Click an output variable from the drop-down menu.

Setting the Calculation Range of a Cost Function Goal

The calculation range is the range within which you want a cost function goal to be calculated. It can be a single value or a range of values, depending on the solution or solution quantity selected for the goal.

1. Under the **Goals** tab in the *Setup Optimization* dialog box, click **Edit Cal. Range**.
2. In the **Variable** drop-down menu, click a variable.

   If you chose to solve a parametric setup during the optimization analysis, the variables swept in that parametric setup are available in the **Variable** drop-down menu. If you sweep a variable in the parametric setup that is also being optimized, that variable is excluded from the optimization.

   Other examples of available variables include frequency, if the solution quantity is an S-parameter quantity, and phi or theta, if the solution quantity is a radiated field quantity.

3. After you select a variable from the **Variable** drop-down menu, you can select a range of values for the calculation range as follows:
   a. Select **Range**.
   b. In the **Start** text box, type the starting value of the range.
   c. In the **Stop** text box, type the final value of the range.
4. To select a single value for the calculation range:
   a. Select **Single Value**.
   b. In the **Value** text box, type the value of the variable at which the cost function goal is to be extracted.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click **Update** and then click **OK**.

Setting a Goal Value

A goal is the value you want a solution quantity to reach during an optimization analysis. It can be a real value or a complex value. If the solution quantity is a complex calculation, the goal value must be complex. You can type the goal value in the **Goal** text box. Alternatively, you can use the *Edit Goal/Value Weight* dialog box to specify the goal value as a single value, a mathematical expression, or a value dependent on a variable such as frequency.

Specifying a Single Goal Value

1. Under the **Goals** tab in the *Setup Optimization* dialog box, click **Edit Goal/Weight**.

   The *Edit Goal/Weight* dialog box appears.

2. Under the **Goal Value** tab, click **Simple Numeric Value** from the **Type** list.
3. If the goal value is complex, click **real/imag** in the drop-down menu to the right if you want to specify the real and imaginary parts of the goal value.

   Alternatively, click **mag/ang** if you want to specify the magnitude and angle of the goal value.

4. Type the goal value in the **Goal Value** table.

   If the goal value is complex, type both parts of the goal value in the text box below the **Goal Value** heading. For example, type **1, 1** to specify the real part of the goal value as 1 and the imaginary part as 1.

   If the goal value is real, type a real goal value in the text box below the **Goal Value** heading.

5. Click **OK**.

   The goal value you specified appears in the **Goal** text box.

Specifying an Expression as a Goal Value

1. Under the **Goals** tab in the *Setup Optimization* dialog box, click **Edit Goal/Weight**.

   The *Edit Goal/Weight* dialog box appears.

2. Under the **Goal Value** tab, click **Expression** from the **Type** list.
3. If you know the syntax of the mathematical expression or the existing output variable's name, type it in the text box below the **Goal Value** heading.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Alternatively, if you want to create an output variable that represents the goal value, do the following:

a. Click **Edit Expression**.

   The *Output Variables* dialog box appears.

b. Add the expression you want to be the goal value, and then click **Done**.

   The most recently created output variable is entered in the text box below the **Goal Value** heading.

4. Click **OK**.

   The goal value you specified appears in the **Goal** text box.

Specifying a Variable-Dependent Goal Value

1. Under the **Goals** tab in the *Setup Optimization* dialog box, click **Edit Goal/Weight**.

   The *Edit Goal/Weight* dialog box appears.

2. Under the **Goal Value** tab, click **Variable Dependent** from the **Type** list.
3. Click a variable from the pull-down list to the left of the table.
4. Type the value of that variable in the first column of the table.

> **Warning:**
>
> Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any optimetric analysis.

5. Type a corresponding goal value for that variable value in the text box below the **Goal Value** heading.
6. Click **Add** to add another row to the reference curve.
7. Repeat steps 4, 5, and 6 until you have specified the reference curve.
8. Click **OK**.

   The goal value is listed as being variable dependent in the **Goal** text box.

Goal Weight

If an optimization setup has a cost function made up of multiple goals, you can assign a different weight to each goal. The goal with the greater weight is given more importance during the cost calculation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The error function value is a weighted sum of the sub-goal errors. Each sub-goal, at each frequency at which it is evaluated, gives rise to a (positive) error value that represents the discrepancy between the simulated response and the goal value limit. If the response satisfies the goal value limit, then the error value is 0. Otherwise, the error value depends on the differences between the simulated response and the respective goal limit. The error function may be defined as follows:

$$\sum_{j}^{G} \frac{W_j}{N_j} \cdot \sum_{i}^{N_j} e_i$$

where

- G is the number of sub-goals.
- W_j is the weight factor associated with the j^{th} sub-goal.
- N_j is the number of frequencies for the j^{th} sub-goal.
- e_i is the error contribution from the j^{th} sub-goal at the i^{th} frequency.

The value of e_i is determined by the band characteristics, target value, and the simulated response value. The choices for band characteristics are <=, =, and >=.

Band Characteristics (Condition)	**e_i evaluation where s_i is the simulated response and g_i is the desired limit.**
<=	$e_i = \begin{cases} 0 & s_i \le g_i \\ s_i - g_i & s_i > g_i \end{cases}$
=	$e_i = \lvert s_i - g_i \rvert$
>=	$e_i = \begin{cases} 0 & s_i \ge g_i \\ g_i - s_i & s_i < g_i \end{cases}$

If the total error value is within the acceptable cost, the optimization stops.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modifying the Starting Variable Value for Optimization

A variable's starting value is the first value to be solved during the optimization analysis. Optimetrics automatically sets the starting value of a variable to be the current value set for the nominal design. You can modify this value for each optimization setup.

> **Note:**
>
> If you choose to solve a parametric setup before an optimization analysis, a variable's starting value is ignored if a more appropriate starting value is calculated for it during the parametric analysis.

1. In the *Setup Optimization* dialog box, click the **Variables** tab.

   All of the variables that were selected for the optimization analysis are listed.

2. Type a new value in the **Starting Value** text box for the value you want to override, and then press **Enter**.

   The **Override** option is now selected. This indicates that the value you entered is used for this optimization analysis, and the current value set for the nominal model is ignored.

   - Alternatively, you can select the **Override** option first, and then type a new variable value in the **Starting Value** text box.

3. Optionally, click a new unit system in one of the **Units** text boxes.

   > **Note:**
   >
   > To revert to the default starting value, clear the **Override** check box.

Setting the Min. and Max. Variable Values for Optimization

For every optimization setup, Optimetrics automatically sets the minimum and maximum values it will consider for a variable being optimized. Optimetrics sets a variable's minimum value equal to approximately 50% of its starting value. (The starting value is the variable's current value set for the nominal design.) Optimetrics sets the variable's maximum value equal to approximately 150% of the starting value. During the optimization analysis, variable values that lie outside of this range are not considered.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Warning:

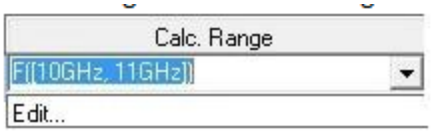
Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any Optimetrics analysis.

Edit Calculation Range

Note:

Functionality featured in the example(s) in this section applies to multiple design types.

In the *Setup Optimization* dialog box, you can enter the Calc. Range Sweep Min/Max by directly editing the Calc. Range field or by accessing an *Edit Calculation Range* dialog box.



The edit field accepts the following forms of text:

- sweep that allows your to select different discrete values:

  discrete values, for example, F(10GHz, 11GHz)

  min/max range, for example, F([10GHz, 11GHz])

- editable sweep, which allows you to customize values (that is a sweep that has an enabled "edited" radio button in sweep selection dialog):

  The min/max is used on top of selected values. For example, if you use the sweep dialog and choose "0 deg, 60 deg, 180 deg, 240 deg", then [60deg, 240deg] will select values "60 deg, 180 deg, 240 deg".

- sweep that uses a full range:

  all values, for example, Time(All)

  min/max range, for example, Time([1ms, 2ms]) HFSS/MAXWELL/TWINBUILDER

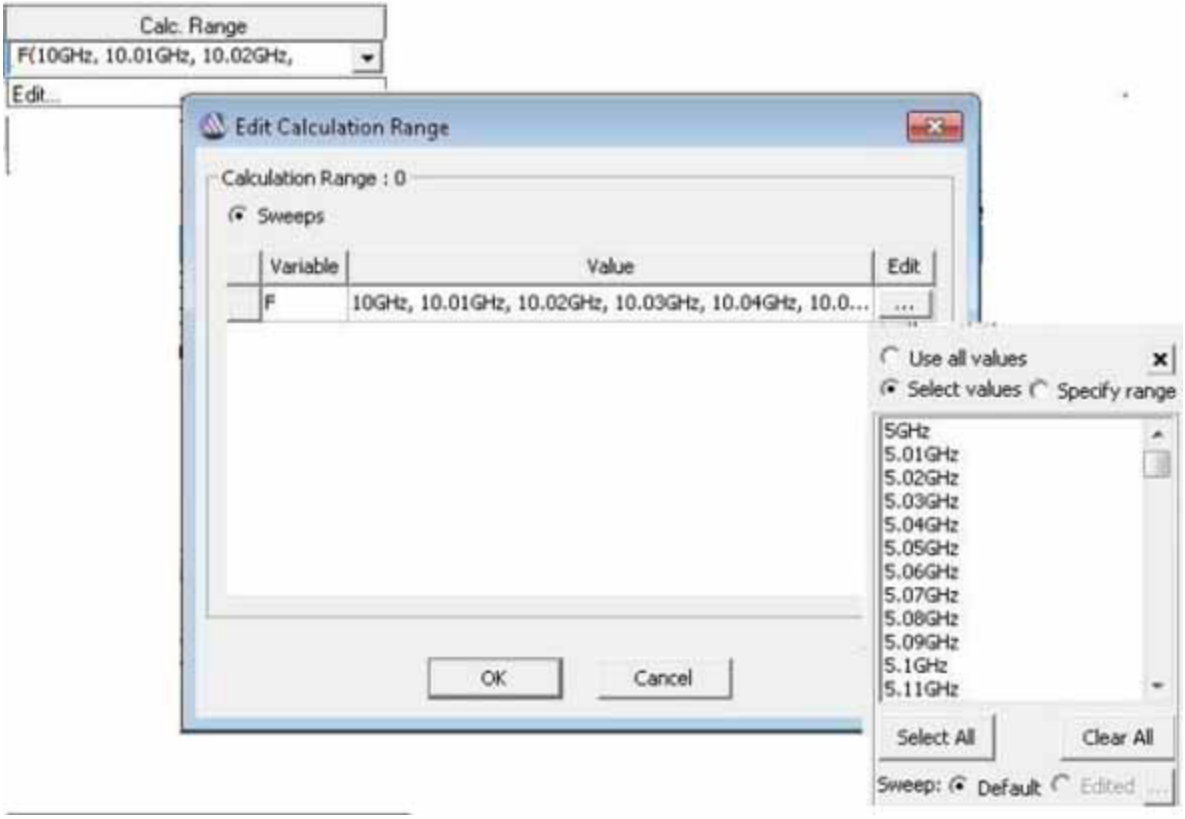
- You solve 1 to 20 GHz step .1 and specify F[10.381GHz, 11.381GHz], it is equivalent to selecting values between 10.4GHz and 11.3GHz.
- You can specify multiple sweep values by separating those with comma ","

  For example, F(1GHz), cap(1pf, 1.2pf)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

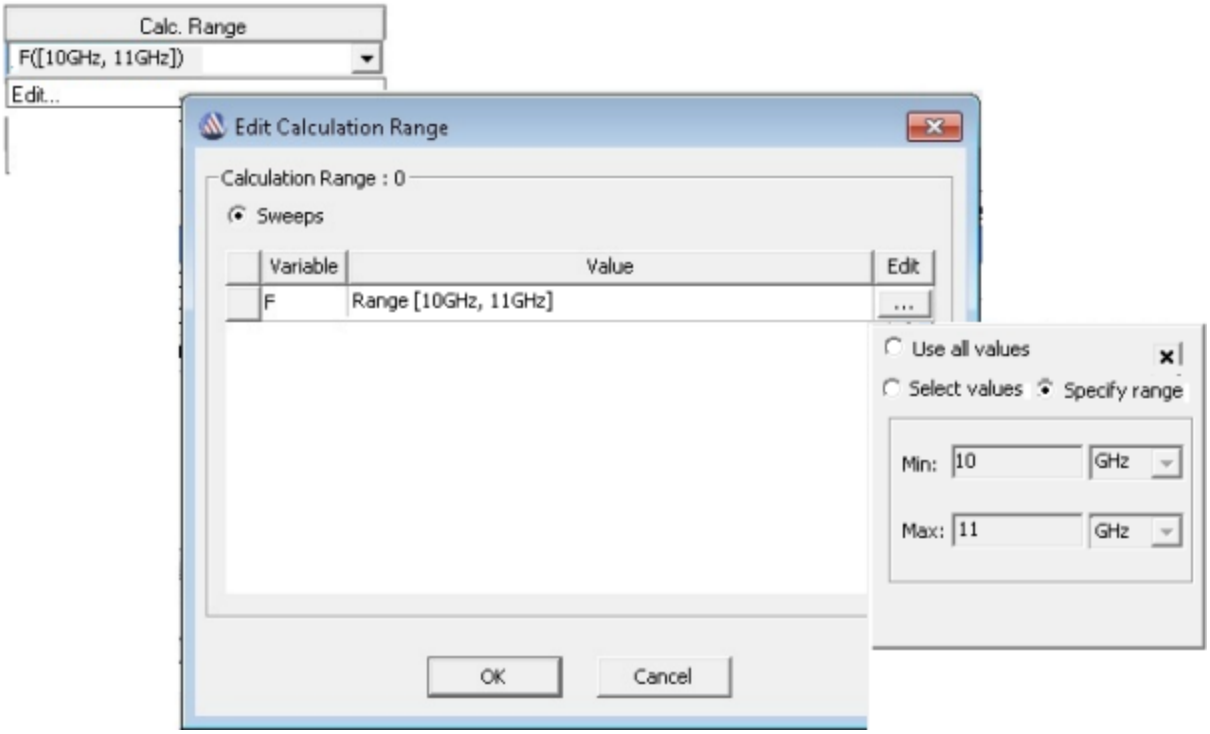
For example, Distance(All), Freq([1ghz,2ghz]), Phase(0 deg)

If you click **Edit..** on the menu, you see the **Edit Calculation Range** dialog box. Click on the ellipsis [...] button to select radio buttons for Use all values, Select Values or Specify range.



This example shows that when you specify a range, how the range appears in the Calc. Range field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You could also enter the range directly in the Cal. Range field.

Overriding the Min. and Max. Variable Values for a Single Optimization Setup

1. In the *Setup Optimization* dialog box, click the **Variables** tab.

   All of the variables that were selected for optimization analysis are listed.

2. Type a new value in the **Min** or **Max** text box for the value you want to override, and then press **Enter**.

   The *Override* option is now selected, indicating that the value you entered is used for this optimization analysis; the variable's current **Min** or **Max** value in the nominal design is ignored.

   - Alternatively, you can select the **Override** option first, and then type a new value in the **Min** or **Max** text box.

3. Optionally, click a new unit system in one of the **Units** text boxes.

To revert to the default minimum and maximum values, clear the **Override** option.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Changing the Min. and Max. Variable Values for Every Optimization Setup

1. Make sure that the variable's minimum and maximum values are not being overridden in any single optimization setup.
2. If the variable is a design variable, do the following: Click **Maxwell**> **Design Properties**.

   If the variable is a project variable, do the following: Click **Project** > **Project Variables**.

   The *Properties* dialog box appears.
3. Select **Optimization**.
4. Type a new value in the **Min** or **Max** text box for the value you want to override, and then press **Enter**.
5. Click **OK**.

   When Optimetrics solves an optimization setup, it does not consider variable values that lie outside of this range.

Step Size

To make the search for the minimum cost value reasonable, the search algorithm is limited in two ways. First, you do not want the optimizer to continue the search if the step size becomes irrelevant or small. This limitation impacts the accuracy of the final optimum. Second, in some cases you do not want the optimizer to take large steps either. In case the cost function is suspected to possess large variations in a relatively small vicinity of the design space, large steps may result in too many trial steps, which do not improve the cost value. In these cases, it is safer to proceed with limited size steps and have more frequent improvements.

For these two limitations, the optimizer uses two independent distance measures. Both are based on user-defined quantities: the minimum and maximum step limits for individual optimization variables. Since the particular step is in a general direction, these measures are combined together in order to derive the limitation for that particular direction.

The step vector between the i^{th} and $(i+1)^{th}$ iterate is as follows:

$$s_i = x_{i+1} - x_i$$

The natural distance measure is

$$\|s_i\| = \sqrt{s_i^T s_i}$$

which is the Euclidean norm.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A more general distance measure incorporates some "stretching" of the design space:

$$\|s_i\|_D = \sqrt{s_i^T D^T D s_i}$$

where the matrix *D* incorporates the linear operation of the stretching of design space. The simplest case is when the *D* matrix is diagonal, meaning that the design space is stretched along the orthogonal direction of the base vectors.

The optimizer stops the search if

$$\|s_i\|_{D_{min}} < 1$$

where D_{min} consists of diagonal elements equal to the inverse of the **Min. Step** value assigned to the corresponding optimization variable. Similarly, the optimizer truncates steps for which

$$\|s_i\|_{D_{max}} > 1$$

where D_{max} has diagonal elements equal to the inverse of **Max. Step** values of the corresponding optimization variables.

Setting the Min. and Max. Step Sizes

For the Quasi Newton and Pattern Search optimizers, the step size is the difference in a variable's value between one solved design variation and the next. The step size is determined when Optimetrics locates the next design variation that should be solved in an effort to meet the cost function.

1. In the *Setup Optimization* dialog box, click the **Variables** tab.
2. Optimetrics displays **Min Step** and **Max Step** columns, with default values for each variable to be optimized.
3. In the **Min Step** text box, type the minimum step size value. Optionally, modify the unit system in the **Units** text box.
4. In the **Max Step** text box, type the maximum step size value. Optionally, modify the unit system in the **Units** text box.

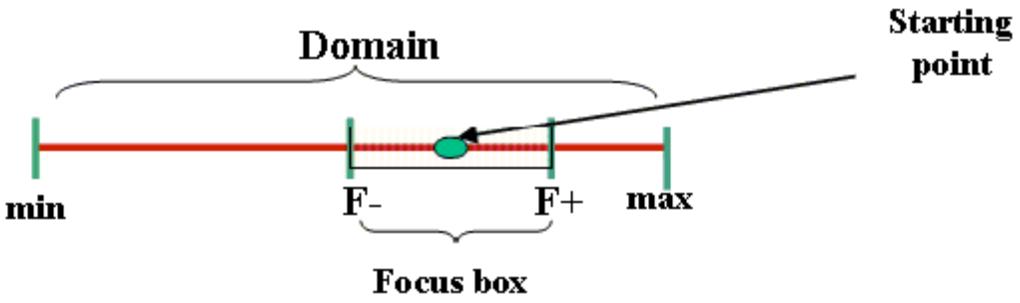
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click **OK**.

> **Tip:**
>
> A value of zero is recommended for the minimum step size.

Setting the Min and Max Focus

For the SNLP, SMINLP and Genetic Algorithm optimizers, the min focus and max focus criteria allow you to specify a sub-range of parameter values where the optimizer should look when performing the optimization. This focus box is where you suspect the optimal solution will be, so it is a hint for the optimizer.



- The domain limits the search. The domain = physical limits.
- The focus box does not limit the search. Rather, the Focus box = an initial guess of optimum search domain. The starting point is the center of the focus box, but the search does extend beyond the box.
- This focus must be inside the domain limits. Consequently, it has to be equal or smaller size. An error message is generated if you specify a focus outside the domain.
- The focus box must be at least one hundredth of the domain size. Otherwise, an error message is sent.

Equalizing the Influence of Different Optimization Variables

The optimizer seeks optimal values for the optimization variables. These variables are usually quantities with specified units. The change in one variable could be measured in [mm] and the change in other variable could be measured in [mA]. Instead of those units, the optimizer uses internal abstract units, so that a change in one variable changes the design behavior about as much as the same change in another variable, where changes are measured in the respective internal abstract units. When you define the focus box, the unit of the abstract internal unit is

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

defined as the difference of the upper and lower focus limits. This way you can use the focus box to equalize the influence of different optimization variables on the design behavior.

Setting the Min and Max Focus Values

1. In the *Setup Optimization* dialog box, click the **Variables** tab.
2. Optimetrics displays **Min. Focus** and **Max. Focus** columns, with default values for each variable to be optimized.

   If you do not have an initial guess based on your knowledge of the problem, make the focus box equal to the domain; that is, the physical limits. This tells SNLP to search the entire decision space.

   - In the **Min. Focus** text box, type the minimum value of the focus range. Optionally, modify the unit system in the **Units** text box.
   - In the **Max. Focus** text box, type the maximum value of the focus range. Optionally, modify the unit system in the **Units** text box.
3. Click **OK**.

Solving a Parametric Setup Before an Optimization

Solving a parametric setup before an optimization setup is useful for guiding Optimetrics during an optimization.

To solve a parametric setup before an optimization setup:

1. In the *Setup Optimization* dialog box, click the **General** tab.
2. In the **Parametric Analysis** drop-down menu, click the parametric setup you want Optimetrics to solve before optimization.

   > **Note:**
   >
   > The parametric setup must include sweep definitions for the variables you are optimizing.

3. Select **Solve the parametric sweep before optimization**.

If the parametric setup has not yet been solved, Optimetrics solves it. Optimetrics uses the cost value evaluated at each parametric design variation to determine the next step in the optimization analysis. This enables you to guide the direction in which the optimizer searches for the optimal design variation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solving a Parametric Setup During an Optimization

Solving a parametric setup during an optimization analysis is useful when you want Optimetrics to solve every design variation specified in the parametric setup at each optimization iteration. A cost function goal could then depend on the value of the variable swept in the parametric setup.

To solve a parametric setup during an optimization analysis:

1. In the *Setup Optimization* dialog box, click the **General** tab.
2. In the **Parametric Analysis** drop-down menu, click the parametric setup you want Optimetrics to solve during an optimization.
3. Select **Solve the parametric sweep during optimization**.
4. Optionally, you can adjust the sweep values to be used during the optimization.
    a. Click on the **Goal** tab, click **Setup Calculations** to specify a calculation.

       The *Add/Edit Calculation* dialog box is displayed.

    b. Click the **Calculation Range** tab.
    c. Click **Edit** for the sweep to be modified.
    d. In the pop-up dialog box, select the sweep values to use.
    e. Close the pup-up dialog box. Click **Done** to close the *Add/Edit Calculation*dialog box.

Automatically Updating a Variable's Value after Optimization

When Optimetrics finds an optimal variable value by solving an optimization setup, it can automatically update that variable's current value set for the nominal model to the optimal value.

1. In the *Setup Optimization* dialog box, click the **General** tab.
2. Select **Update design parameters' values after optimization**.

   When optimization is complete, the current variable value for each optimized variable is changed to the optimal value.

Changing the Cost Function Norm

You can select the norm to be used in the calculation of the cost goal.

1. In the *Setup Optimization* dialog box, click the **Goals** tab.
2. Select **Show Advanced Options**.
3. Select a norm from the drop-down menu in the **Cost Function Norm Type** field. The options are **L1**, **L2**, and **Maximum**. **L2** is the default.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Explanation of L1, L2, and Max Norms in Optimization

When you set multiple goals for an optimization, the question arises as to what is actually going to drive the optimizer which is not a multi-objective one. The cost function will have a lot to do with it. The following discussion explains how the cost function is put together when there are multiple goals.

The general goal setting structure in Optimetrics is a logical sentence with the format:

Calculation$_{(i)}$ *Condition*$_{(i)}$ *Goal*$_{(i)}$ *Weight*$_{(i)}$

The cost function that the optimizer uses is built based on the norm setting as long as there are multiple goals and none of those use the "minimize" or "maximize" conditions. Thus, in this case the error associated with each individual goal (weighted) is combined in a way that is specific for each norm type chosen.

For **L1** norm, the actual cost function uses the sum of absolute weighted values of the individual goal errors:

$$Cost = \sum_{1}^{N} \left| w_i \varepsilon_i \right|$$

For **L2** norm, the cost function computes the weighted sum of the squares of individual goal errors.

$$Cost = \sum_{1}^{N} w_i \varepsilon_i^2$$

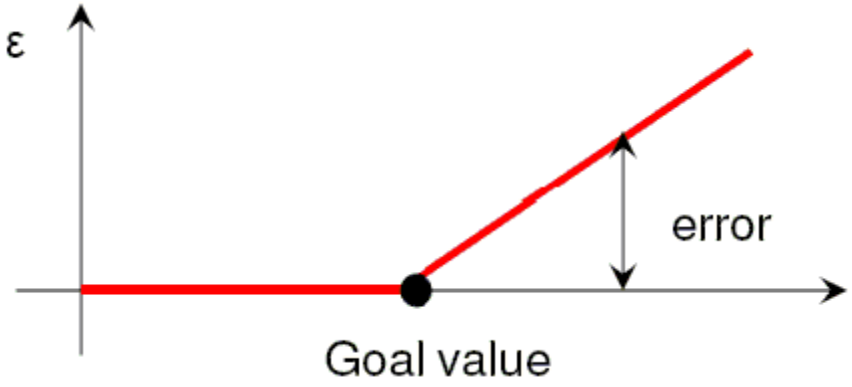
For the **Maximum** norm, the cost function uses the maximum among all the weighted goal errors, which means that cost is always less than zero:

$$Cost = \max_{1}^{N} W_i \cdot \varepsilon_i$$

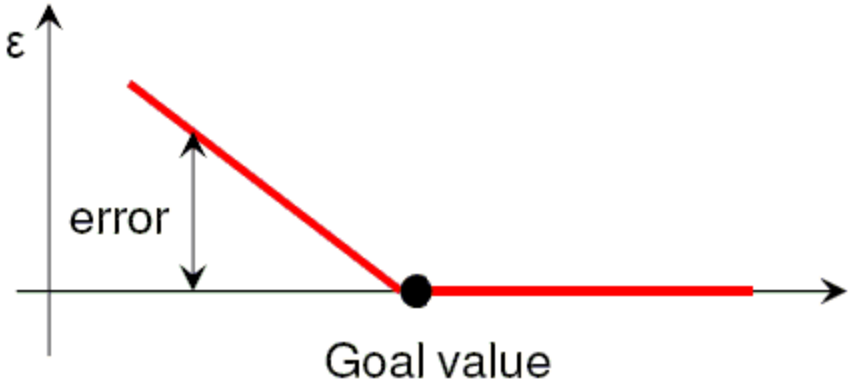
For all the above situations, N is the number of individual goals $w_i \varepsilon_i$ are individual weighting factors and residual error respectively. A minimization of the cost function is performed during optimization since it makes sense to minimize the error in the sense of the chosen norm type.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

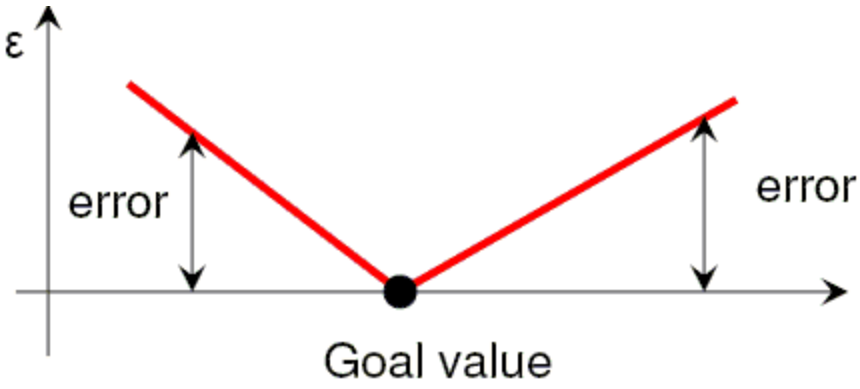
The graphical representation of the error is possible and depends upon the actual condition being used. If a "<" condition is used, the error can be represented as below:



If a ">" condition is used, the error can be represented as below:



If a "=" condition is used, the error is double-sided and can be represented as below:



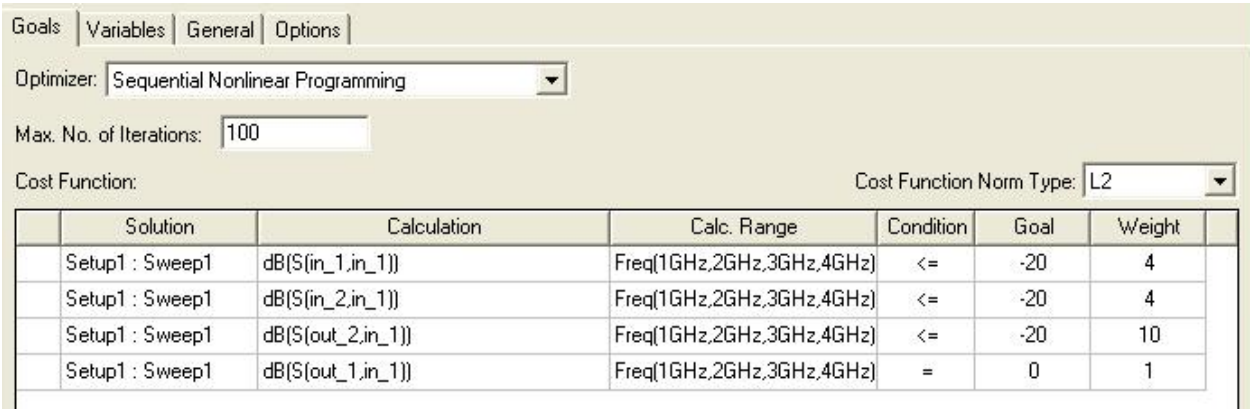
The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios. Note that when using "minimize" or "maximize" settings for the condition there should be a single goal setting which in this case coincides with the cost function.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example of a More Complex Cost Function

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

As an example of a more sophisticated cost function, consider the figure. It belongs to a connector simulated in HFSS with more than four ports.



	Solution	Calculation	Calc. Range	Condition	Goal	Weight
	Setup1 : Sweep1	dB(S(in_1,in_1))	Freq(1GHz,2GHz,3GHz,4GHz)	<=	-20	4
	Setup1 : Sweep1	dB(S(in_2,in_1))	Freq(1GHz,2GHz,3GHz,4GHz)	<=	-20	4
	Setup1 : Sweep1	dB(S(out_2,in_1))	Freq(1GHz,2GHz,3GHz,4GHz)	<=	-20	10
	Setup1 : Sweep1	dB(S(out_1,in_1))	Freq(1GHz,2GHz,3GHz,4GHz)	=	0	1

The cost function given here concentrates only on a signal sent into port in_1. Suppose the specifications to be met are: reflection, backward cross talk and forward cross talk all smaller than or equal to -20 dB, of which the forward cross talk is the most important.

The first three entries in the cost function enforce those specifications, with the weight for the forward cross talk being a larger number than the other weights. The actual values for the weights are somewhat arbitrary and serve as examples only. For this cost function, as long as specifications are not met, the optimizer puts the most effort in getting the forward cross talk close to its specification. Once the three specifications have been satisfied, their contributions to the cost function become zero, and only the fourth entry remains. Remember that the connector has more than four ports, so satisfying the given specs does not guarantee maximum transmission.

The fourth line tries to maximize the transmission by asking for S(out_1, in_1) to be 0 dB. That will never be reached, but its presence forces the optimizer to improve the connector a bit beyond the specifications.

The cost function norm type specifies how the four lines are combined into one cost function with one value. With L1 and L2, all four contribute simultaneously, rather than only the largest of the four at any one time.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Advanced Genetic Algorithm Optimizer Options

The Genetic Algorithm (GA) search for Optimization analysis is an iterative process that goes through a number of generations. In each generation some new individuals (Children / Number of Individuals) are created and the so grown population participates in a selection (natural-selection) process that in turn reduces the size of the population to a desired level (Next Generation / Number of Individuals).

If you select the Genetic Algorithm for an Optimization analysis, a **Setup** button is enabled on the **Setup Optimization** page.

1. Click **Setup** to open the *Advanced Genetic Algorithm Optimizer Options* dialog box.
2. Select the Stopping Criteria. Any of the three following, or any combination of these can be selected.
    - **Maximum number of generations**. If checked, this enables a value field.
    - **Elapsed time**. If checked, this enables a drop-down menu with times ranging from five minutes to two weeks.
    - **Slow convergence**.
3. Specify the Parents.

    The first step toward mating is a selection process that determines the participating individuals. Potential parents are selected from the Current Generation. This is a set of individuals that is always a subset of the current generation.

    - **Number of individuals** value field -- specify the number of parents for the optimizer to use. You can set the Number of Individuals to less than or equal to the size of the "Current Generation". One reason to consider fewer parents than the possible maximum is to steer the GA toward improvement by selecting the better portion of the current generation to be able to mate.
    - **Roulette selection** check box -- if checked, this enables the **Selection pressure** value field. This number defines how many times more probable is the selection of the best individual over the worst individual in an elementary spin of the roulette wheel.
4. Specify the Mating pool.

    The Mating pool is created by selecting randomly from the parents, but with each selection, the parent gets "cloned" so it can be selected again and again.

    - **Number of individuals** field -- specify the number individuals to include in the mating pool.
    - **Reproduction setup**-- this button opens the *Genetic Algorithm Optimizer Reproduction Setup* dialog box.
5. Click **Reproduction setup** for the dialog box to specify the Crossover setup, and the Mutation setup.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The crossover and mutation operator have different roles: *Crossover* mixes "features" of the parents in a new combination, while *mutation* slightly alters the "features" of the individuals. Both need to be present in a GA. The crossover is a way to discover new combinations while the mutation acts as a local search or fine-tuning step. Mutation also keeps diversity in a population, which is a must for GA.

The crossover operator has two steps. It first alters the variable values of the parents according to a distribution. This tends to produce one child that looks a lot like one parent, and one child that looks a lot like the other parent. Next, some of the variable values of the two children can be exchanged in order to achieve more variation.

For crossover there are four possible parameters.

a. **Individual Crossover Probability** determines, for each pair in the mating pool, the probability that their features will be mixed. Usually, this probability should be close or equal to one. If you set it to less than one, some parents will produce two children which are exact clones of the parents. This means that some children inherit all the features of their parents unchanged.
b. Parents often have multiple variables. If the parent is a candidate for mixing, the **Variable Crossover Probability** determines, for each variable, the probability of mixing. This is usually set high to ensure that most or all variables mix.
c. **Variable Exchange Probability**: After the slight change in the variable values has been made, the crossover operation is also able to exchange the values of the variables between the two children that are being constructed. The Variable Exchange Probability governs the likelihood of exchange of any variable.
d. **Mu** is a general parameter defining the sharpness of the distribution that might be used for the **Variable Crossover Probability**. Mu should be greater than one. There is no theoretical upper limit, but we recommend not exceeding 30.

6. Select one of the four **Crossover types** from the drop-down menu.

   The crossover type selected affects the options available.

Uniform	Individual crossover probability Variable crossover probability
One point	Individual crossover probability
Two point	Individual crossover probability
Simulated binary crossover	Individual crossover probability Variable crossover

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

probability
Variable exchange
probability
Mu

7. Select the **Mutation type** – this can be one of three types, which you select from a drop-down menu:
   - **Uniform Distribution**
   - **Gaussian Distribution**
   - **Polynomial Mutation**
8. For the selected mutation type, set the following parameters:
   - **Uniform Mutation Probability**: If this is more than zero (recommendation is to have still a small probability here), then there will be some children whose features are simply a completely random design (design variables randomly selected over the domain).
   - **Individual Mutation Probability** controls, for each child, the likelihood of a mild mutation.
   - **Variable Mutation Probability**. If the child will be mutated, this probability controls at the variable level the likelihood of a mutation of the variables.
   - **Standard Deviation** is the standard deviation of the selected distribution that is being used for the mutation and it is measured relatively to the optimization-domain.
9. When you have completed the Reproduction setup in the *Genetic Algorithm Optimizer Reproduction Setup* dialog box, click **OK** to close it and return to the *Advanced Genetic Algorithm Optimizer Options* dialog box.
10. In the *Advanced Genetic Algorithm Optimizer Options* dialog box, specify the children as a Number of Individuals.
11. Set the **Pareto Front** value.

    This is the number of the very best individuals (identified relative to the cost function) to keep for future generations.

12. Set the Next Generation parameters. The Next Generation is selected from the Parents, the children, and the Pareto front.
    - **Number of individuals** value field -- specify the number of individuals to survive to form the next generation for the optimizer to use.
    - **Roulette selection** check box -- if checked, this enables the **Selection pressure** value field. This number defines how many times more probable is the selection of the best individual over the worst individual in an elementary spin of the roulette wheel.
13. Click **OK** to accept the settings for the Genetic Algorithm and to close the dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sensitivity Analysis Overview

During a sensitivity analysis, Optimetrics explores the vicinity of the design point to determine the sensitivity of the design to small changes in variables. The variables and their attributes define the design point, the problem around which the sensitivity analysis is performed.

When Optimetrics performs a sensitivity analysis, its goal is to calculate the second-order regression polynomials for all of the design's output parameters. The algorithm first determines an appropriate interval for each variable. The intervals are further sub-divided according to the available number of iterations and variables. If the primary output is not used, the specified initial displacement values define those intervals.

When all of the design calculations are complete, the second-order polynomials are fitted for all the output parameters. Optimetrics then reports the following quantities:

- Regression value at the current variable value
- First derivative of the regression
- Second derivative of the regression

> **Note:** Sensitivity Analysis has been placed in legacy mode because it is no longer under active development. Go to **Tools** > **Options** > **General Options** > **Miscellaneous** and toggle the **Enable Legacy Optimetrics Tools** check box to show or hide these legacy tools.

Selecting a Primary Output

During a sensitivity analysis, the design variations that Optimetrics selects to solve are close to the design point, but not so close that numerical noise (from the finite element mesh) affects the analysis. The algorithm that Optimetrics uses to determine the design variations to solve must be based on only one output parameter and that output parameter's numerical noise. Therefore, if you have defined more than one output parameter, be sure to select **Primary Output** for the output variable on which you want the selection of design variations to be based.

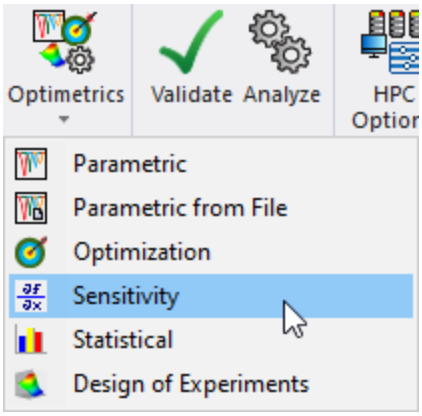
Setting Up a Sensitivity Analysis

Following is the general procedure for setting up a sensitivity analysis. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes. You can create a sensitivity setup before defining variables but all variables must be defined before you start the sensitivity analysis.

1. Before a variable can be included in a sensitivity analysis, you must specify that you intend for it to be used during a sensitivity analysis in the **Design Properties** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **Maxwell** > **Optimetrics Analysis** > **Add Sensitivity** . You can also select the **Simulation** tab in the ribbon, and select **Parametric** from the drop-down menu under the Optimetrics icon:



The **Setup Sensitivity Analysis** dialog box appears.

3. Under the **Calculations** tab, type the maximum number of iterations per variable value that you want the software to perform in the **Max. No. of Iterations/Sensitivity Variable** text box.
4. Set up an output parameter calculation, and select a Primary Output.
5. Specify the value of the design point at which the sensitivity analysis should stop in the **Approximate Error in Primary Output** text box.
6. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization, and the **Initial Displacement (Initial Disp.)** for the analysis.

   You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.

7. In the **General** tab, specify whether Optimetrics should use the results of a previous parametric analysis or perform one as part of the optimization process.

   Checking the Optional **Worst Case Analysis** option does an extreme value analysis that focuses on the upper and lower boundaries of all the analyzed parameters. Some setup is required before performing Worst Case Analysis.

8. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

The following ***optional*** sensitivity analysis setup options can also be used:

- Modify the starting variable value.
- Modify the minimum and maximum values of variables that will be solved.
- Exclude variables from the sensitivity analysis.
- Set the initial displacement.
- Modify the values of fixed variables that are not being modified during the sensitivity analysis.
- Set linear constraints.
- Request that Optimetrics solve a parametric sweep **before** a sensitivity analysis.
- You can also request that Optimetrics solve a parametric sweep **during** a sensitivity analysis.

> **Note:**
>
> Sweeping or using a complex variable is not allowed in any optimetrics setup, including optimization, statistical, sensitivity, and tuning setups.

Setting the Maximum Iterations Per Variable

The **Max. No. of Iterations/Sensitivity Variable** value is the maximum number of design variations that Optimetrics solves per variable during a sensitivity analysis. This value is a stopping criterion; if the maximum number of iterations has been completed, the sensitivity analysis stops. If the maximum number of iterations has not been completed, the sensitivity analysis continues by performing another iteration, that is, by solving another design variation. It performs iterations until the approximate error in primary output value is reached or until Optimetrics cannot proceed as a result of other sensitivity setup constraints, such as when it searches for a variable value that is larger than the maximum value.

To set the maximum number of iterations for a sensitivity analysis:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Under the **Calculations** tab of the *Setup Sensitivity Analysis* dialog box, type a value in the **Max. No. of Iterations/Sensitivity Variable** text box.

Setting Up an Output Parameter

Following is the general procedure for adding an output parameter to a sensitivity setup:

1. Under the **Calculations** tab of the *Setup Sensitivity Analysis* dialog box, click **Setup Calculations** to open the *Add/Edit Calculations* dialog box.
2. In the *Add/Edit Calculations* dialog box, set up output parameter calculations to be evaluated for sensitivity.
3. To modify the solution from which the output parameter is to be extracted, click in the **Solution** column and select from the options in the pop-up list.
4. You can modify the Calculation specified by clicking on the output parameter in the table and selecting **Edit**.
5. For output parameters based on swept variable, you must choose a single value in the **Calculation Range** at which to evaluate the output parameter.
6. If the output parameter is based on a swept variable, in the **Calculation Range** column, set the value of the variable at which the output parameter is to be computed.
7. If you have more than one output parameter, select **Primary Output** if you want Optimetrics to use the output parameter to base its selection of solved design variations.

> **Note:**
>
> During a sensitivity analysis, the design variations that Optimetrics selects to solve are close to the design point, but not so close that numerical noise (from the finite element mesh) affects the analysis. The algorithm that Optimetrics uses to determine the design variations to solve must be based on only one output parameter and that output parameter's numerical noise. If you have defined more than one output parameter, be sure to select **Primary Output** for the output variable on which you want the selection of design variations to be based.

Specifying a Solution Quantity for an Output Parameter

When setting up an output parameter, you must identify the solution quantity on which to base the output parameter. Solution quantities are specified by mathematical expressions that are composed of basic quantities, such as matrix parameters; and output variables.

The *Add/Edit Calculation* dialog box allows you to define the mathematical equation for one or multiple output parameters. To set up an output parameter:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the **Context** section of the dialog box:
    - Select the **Report Type** with a pull-down selection list containing the available types for this design.
    - Select the **Solution** from the drop down selection list. This lists the available setups and sweeps. As a minimum, the **LastAdaptive** solution is available.
    - Select the **Geometry** from the drop down selection list or select none (the default). This modifies the list of quantities available to the ones that apply to the specific geometry.
    - When selecting a geometry, you may also be required to specify a point within the geometry where the calculation is to be performed.
2. The **Output Variables** button opens the Output Variables dialog box allowing you to create special output variables to be used in the output parameter.
3. The **Calculation Expression** field in the **Trace** tab is used to enter the equation to be used for the output parameter. To enter an expression, you may type it directly into the field or use the **Category**, **Quantity**, and **Function** lists as follows:
    - Select the **Category**, these depend on the Solution type and the design. This lets you specify the category of information to be used in the output parameter.
    - Select a **Quantity** from the list. Available quantities depend upon the Solution type, as well as the Geometry and Category selection. Selecting a Quantity automatically enters it into the Calculation Expression field.
    - Select a **Function** to apply to the value in the calculated expression.
    - For swept variables, the **Range Function** button opens the **Set Range Function** dialog to apply functions to the expression that apply over the sweep range.
4. The **Calculation Range** tab applies to swept variables and allows you to specify the range of the sweep over which to apply the calculation.
5. When the desired **Calculation Expression** has been obtained, click **Add Calculation** to add the entry to the calculation table in the Setup Sensitivity Analysis dialog box. You may add multiple entries to the table simply by changing the **Calculation Expression** and using the **Add Calculation** button.
6. To update or edit a selected cost function, enter the desired Calculation Expression and click the **Update Calculation** button.
7. Click **Done** to return to the *Setup Sensitivity Analysis* dialog box.

> **Note:**
>
> The solution quantity you specify must be able to be evaluated to a single, real number.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Calculation Range of an Output Parameter

The calculation range of a solution quantity determines the intrinsic variable value at which the solution quantity is to be extracted. For a sensitivity setup, the calculation range must be a single value. If you specified that the solution quantity be extracted from a frequency sweep solution, by default, Optimetrics uses the starting frequency in the sweep. If you specified that the solution be extracted from the last adaptive solution, Optimetrics uses the adaptive frequency defined in the solution setup.

1. Under the **Calculations** tab of the *Setup Sensitivity Analysis* dialog box, click in the **Calculation Range** column of the table for the calculation to be modified.

   The *Edit Calculation Range* dialog box appears.

2. In the **table**, click **Edit** in the row to be modified.

   If you choose to solve a parametric setup during the sensitivity analysis, the variables swept in that parametric setup are available in the pop-up list dialog box. If you sweep a variable in the parametric setup that is also a sensitivity variable, that variable is excluded from the sensitivity analysis.

   Other examples of available variables include frequency, if you selected an S-parameter solution quantity; and phi or theta, if the solution quantity is a radiated field quantity.

3. Click on the value for the calculation range in the list and dismiss the pop-up dialog box.
4. Click **OK** in the *Edit Calculation Range* dialog box to accept the new value for the intrinsic variable, and return to the *Setup Sensitivity Analysis* dialog box.

Modifying the Starting Variable Value for Sensitivity Analysis

The design point of the sensitivity analysis is the starting value of the sensitivity variable and is usually the first variation to be solved. Optimetrics automatically sets the starting value of a variable to be the current value set for the nominal design. You can modify the design point for each sensitivity setup.

> **Warning:**
>
> Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any optimetric analysis.

1. In the *Setup Sensitivity Analysis* dialog box, click the **Variables** tab.

   All of the variables that were selected for the sensitivity analysis are listed.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Type a new value in the **Starting Value** text box for the value you want to override, and then press **Enter**.

   The **Override** option is now selected. This indicates that the value you entered is to be used for this sensitivity analysis; the current value set for the nominal model will be ignored.

   - Alternatively, you can select the **Override** option first, and then type a new variable value in the **Starting Value** text box.
3. Optionally, click a new unit system in one of the **Units** text boxes.

To revert to the default starting value, clear the **Override** option.

Setting the Min. and Max. Variable Values

For every sensitivity setup, Optimetrics automatically sets the minimum and maximum values that it will consider for a sensitivity variable. Optimetrics sets a variable's minimum value equal to approximately one-half its starting value. (The starting value is the variable's current value set for the nominal design.) Optimetrics sets the variable's maximum value equal to approximately 1.5 times the starting value. During sensitivity analysis, variable values outside this range are not considered.

> **Warning:**
>
> Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any optimetric analysis.

Overriding the Min. and Max. Variable Values for a Single Sensitivity Setup

1. In the *Setup Sensitivity Analysis* dialog box, click the **Variables** tab.

   All of the variables that were selected for sensitivity analysis are listed.
2. Type a new value in the **Min** or **Max** text box for the value you want to override, and then press **Enter**.

   The **Override** option is now selected. This indicates that the value you entered is to be used for this sensitivity analysis; the variable's current **Min** or **Max** value set in the nominal design is ignored.

   - Alternatively, you can select the **Override** option first, and then type a new value in the **Min** or **Max** text box.
3. Optionally, click a new unit system in one of the **Units** text boxes.

   To revert to the default minimum and maximum values, clear the **Override** option.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Changing the Min. and Max. Variable Values for Every Sensitivity Setup

1. Make sure the variable's minimum and maximum values are not being overridden in any sensitivity setup.
2. If the variable is a design variable, do the following: Click **Maxwell**> **Design Properties**.

   If the variable is a project variable, do the following: Click **Project** > **Project Variables**.

   The *Properties* dialog box appears.
3. Select **Sensitivity**.
4. Type a new value in the **Min** or **Max** text box for the value you want to override, and then press **Enter**.

   When Optimetrics solves a sensitivity setup, it does not consider variable values that lie outside of this range.

Setting the Initial Displacement

The initial displacement is the difference in a variable's starting value and the next solved design variation. During the sensitivity analysis, Optimetrics does not consider an initial variable value that is greater than this step size away from the starting variable value.

1. In the *Setup Sensitivity Analysis* dialog box, click the **Variables** tab.
2. Optimetrics displays the **Initial Disp.** column, with default values for each sensitivity variable.
3. In the **Initial Disp.** text box, type the initial displacement value. Optionally, modify the unit system in the **Units** text box.

Solving a Parametric Setup before a Sensitivity Analysis

Solving a parametric setup before a sensitivity setup is useful for guiding Optimetrics in a sensitivity analysis.

To solve a parametric setup before a sensitivity setup:

1. In the *Setup Sensitivity Analysis* dialog box, click the **General** tab.
2. Click the parametric setup you want Optimetrics to solve before the sensitivity setup from the **Parametric Analysis** drop-down menu.

   > **Note:**
   >
   > The parametric setup must include sweep definitions for the sensitivity variables.

3. Select **Solve the parametric sweep before analysis**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the parametric setup has not yet been solved, Optimetrics solves it. Optimetrics uses the results (of the solution calculation you requested under the **Goals** tab of the *Setup Sensitivity* dialog box) to determine the next design variation to solve for the sensitivity analysis.

Solving a Parametric Setup during a Sensitivity Analysis

Solving a parametric setup during a sensitivity analysis is useful when you want Optimetrics to solve every design variation in the parametric setup at each sensitivity analysis iteration. An output parameter goal could then depend on the value of the variable swept in the parametric setup.

To solve a parametric setup during a sensitivity analysis:

1. In the *Setup Sensitivity Analysis* dialog box, click the **General** tab.
2. Click the parametric setup you want Optimetrics to solve during the sensitivity analysis from the **Parametric Analysis** drop-down menu.
3. Select **Solve the parametric sweep during analysis**.

Performing Worst Case Analysis

Two popular worst case analysis techniques can be implemented: extreme value analysis and Monte Carlo analysis.

Some setup is required before performing worst case analysis. First, identify uncertainties in design and create a local or project variable for each of them. Second, determine the variation range of each variable (Min and Max) – its statistical distribution is optional. Third, determine a measurement of performance, especially for extreme value analysis.

1. **Extreme value analysis:**

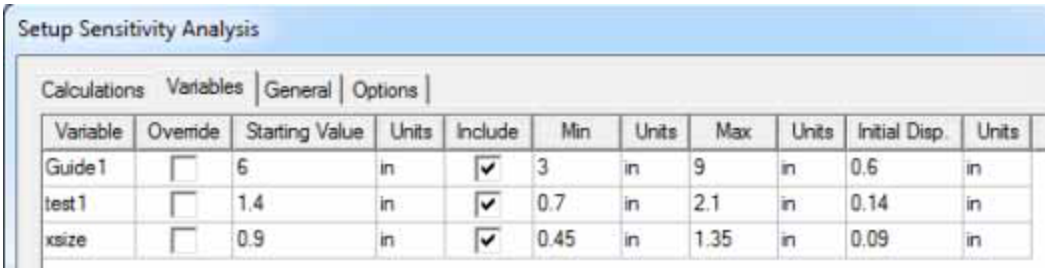
   This is one of the most popular methods to estimate worst-case performance. To start, a sensitivity analysis is performed. The results (sensitivities/first derivative) allow us to pick an extreme value (upper or lower bound) for each variable. The corresponding simulation result is used to predict upper and lower bound of performance. The assumption is that extreme performance is reached at boundary value (note that in certain cases, making such an assumption is not valid).

   To perform an extreme value analysis in Ansys Electronics Desktop, create a new analysis under **Optimetrics** > **Sensitivity**.

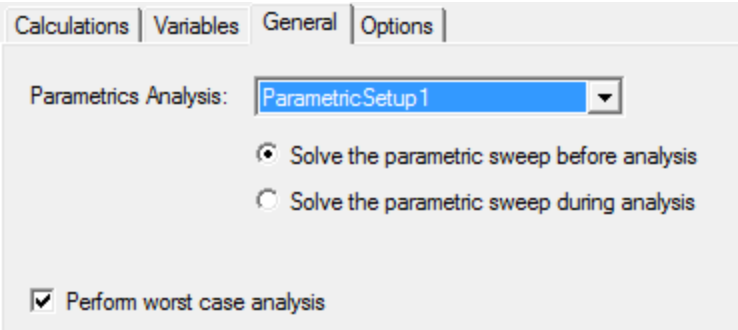
   - Setup:

     First, include variables in a sensitivity analysis. To do so, go to your list of variables, under Sensitivity, specify Min and Max values of each variable, and check 'Include'. The following figure shows an example with three variables included.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Second, follow **Setting Up a Sensitivity Analysis** in the help. During this procedure, set your performance measurement in **Calculations** tab; check all variables in **Variables** tab; check **Perform worst case analysis** in **General** tab.



> **Note:**
>
> Checking **Perform worst case analysis** calculates 1st derivatives for each variable. If we have three variables and for Var1 first derivative is negative, Var2 1st derivative is positive, Var3 1st derivative is positive, then for Worst Case Analysis, we request two more variations:

Var1@minimum, Var2@maximum, Var3@maximun

and Var1@maximum, Var2@minimum, Var3@minimum

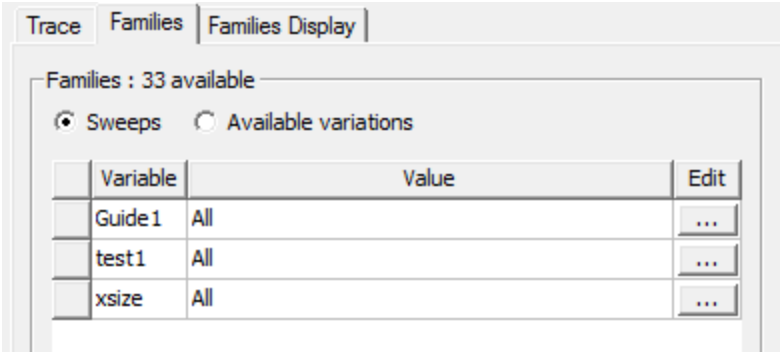
- After setup, analyze your Sensitivity Setup under Optimetrics.
- To view worst-case result (upper bound only):

  Create a Data Table report.

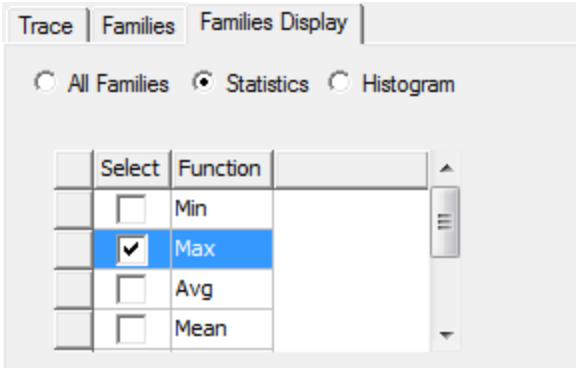
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Under the Context pane in the **Report** dialog box, select matching Solution and Optimetrics setup. Under the **Families** tab, change the setting to 'All' under 'Value' of each variables.



Under the **Families Display** tab, select **Statistics**and then check **Max**.



The associated data table shows the Max values.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

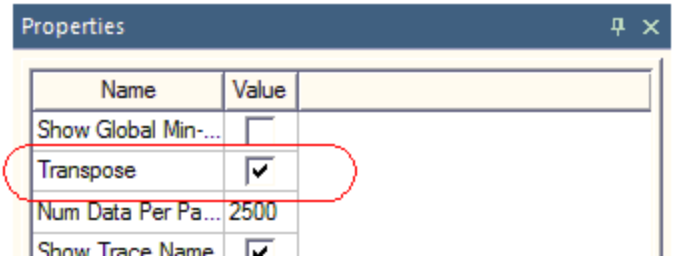
	Freq [GHz]	dB(S(1:1,1:1)) Setup1 : Sweep Max
1	3.000000	-0.000000
2	4.000000	-0.000000
3	5.000000	-0.105323
4	6.000000	-0.248322
5	7.000000	-0.267220
6	8.000000	-0.202039
7	9.000000	-0.298840

To see the corresponding variable values, select 'All Families' under the **Families Display** tab, and locate the max value to see value of variables.

	Freq [GHz]	dB(S(1:1,1:1)) Setup1 : Sweep Guide1='3in' test1='2.1in' xsize='0.45in'
1	3.000000	-0.545922
2	4.000000	-2.300003
3	5.000000	-10.634331
4	6.000000	-4.586505
5	7.000000	-5.950253
6	8.000000	-9.019574
7	9.000000	-3.210097

Tip:

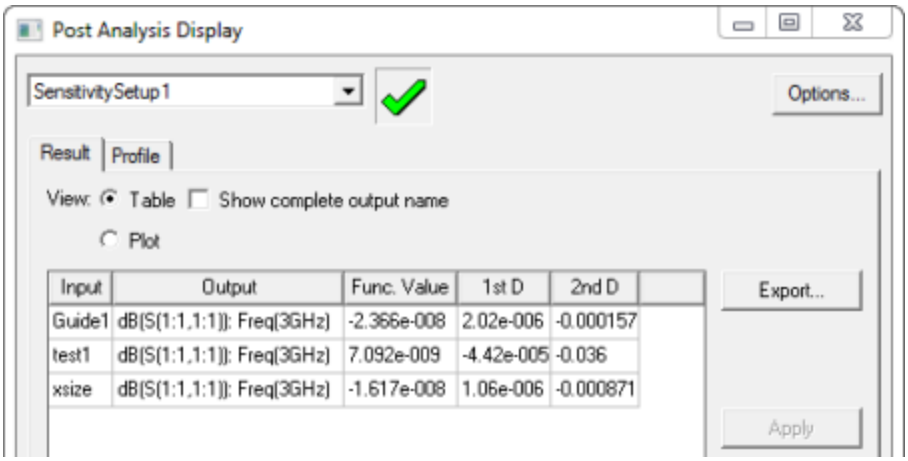
Transpose the table for an alternate view (double-click data table to view the **Properties** window, and on the **Data Table** tab check **Transpose**).



To estimate lower bound of performance:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

First, see the sensitivity of performance to the respect of each variable. (Follow the documentation for Viewing Output Parameter Results for a Sensitivity Analysis). Identify the variables that have major influence over performance.



Second, in your project, manually change these variables to the corresponding bounds: choose Min for positive 1st derivative and Max for negative 1st derivative.

2. **Monte Carlo analysis:**

   This method does not assume a circuit is linear - better accuracy is achieved with more iterations. The cost is computing time and resources.

   To perform Monte Carlo Analysis in Ansys Electronics Desktop, create a new analysis under **Optimetrics> Statistical**. Follow the documentation on Statistical Analysis to set this up. The upper and lower bound of performance can be found on the edge of performance distribution.

   > **Tip:**
   >
   > If distribution of performance is not of interest, set all variables as uniformly distributed.

Viewing Output Parameter Results for a Sensitivity Analysis

To view actual output parameter values versus design point in data table format:

1. In the project tree, right-click the sensitivity setup for which you want to view the parameter results, and select **View Analysis Result**.

   The **Post Analysis Display** dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Under the **Result** tab, select **Table** as the view type, if it is not already selected.

   The following values are listed in table format:

   - The regression value of the output parameter at the design point is listed in the **Func. Value** column.
   - The first derivative of the regression is listed in the **1st D** column.
   - The second derivative of the regression is listed in the **2nd D** column.

3. Click **Apply**.

   The software now points to the selected design variation as the nominal solution and as a result, the design displayed in the **Modeler** window is changed to represent the selected design variation.

   Click **Revert** to return the design in the view window to the original value.

Plotting Output Parameter Results for a Sensitivity Analysis

To plot output parameter results versus sensitivity variable values on a rectangular (xy) plot:

1. In the project tree, right-click the sensitivity setup for which you want to view the output parameter results, and select **View Analysis Result**.

   The **Post Analysis Display** dialog box appears.

2. Under the **Result** tab, select **Plot** as the view type.
3. Select the sensitivity variable with the sweep values you want to plot on the x-axis from the **X** drop-down menu.
4. Select the output parameter results you want to plot on the y-axis from the **Y** drop-down menu.

   The xy plot appears in the **Post Analysis Display** dialog box.

   The plot displays actual output parameter results for each solved design variation. It also displays a parabola that best fits these results. The parabola is a more accurate representation of sensitivity around the design point than any individual solved design variation.

Statistical Analysis Overview

Statistical analysis allows you to explore the effects of random combinations of values of selected variables on selected global or local available analysis results. Therefore, before a variable can be included in a statistical analysis, you must specify that you intend for it to be used during a statistical analysis. For each variable you must specify the type of distribution (Uniform, Gaussian, Lognormal or User Defined) and the corresponding parameters of the selected distribution.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** A statistical analysis is limited to using a maximum of 30 variables.

In addition to specifying the variables to be used in the statistical analysis and the parameters of the chosen distribution, the output quantities of interest also need to be specified. These quantities can be global ones such as previously defined parameters (Force/torque, inductance / capacitance, etc), other named quantities, quantities defined in the field calculator as global (such a domain integral of a certain field quantity) or local (such as field value at a certain location). The calculations to be performed during the statistical analysis are specified during setup, in a manner similar to other types of analysis in Optimetrics.

Following the analysis the statistical distribution of the output quantities can be visualized in histogram format. To access available reports, after the statistical analysis is complete, right click the respective Statistical analysis setup and select **View Analysis Result**.

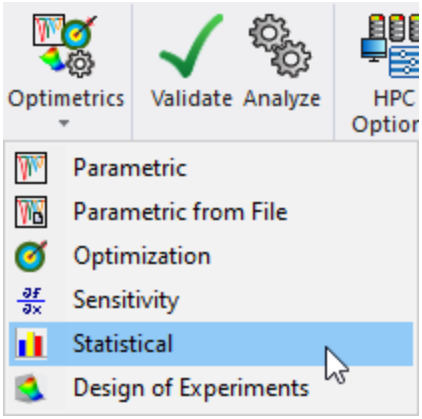
> **Note:**
>
> Statistical Analysis has been placed in legacy mode because it is no longer under active development. Go to **Tools** > **Options** > **General Options** > **Miscellaneous** and toggle the **Enable Legacy Optimetrics Tools** check box to show or hide these legacy tools.

Setting Up a Statistical Analysis

Following is the general procedure for setting up a statistical analysis. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes. You can create a statistical setup before defining variables but all variables must be defined before you start the statistical analysis.

1. Before a variable can be included in a statistical analysis, you must specify that you intend for it to be used during a statistical analysis in the **Properties** dialog box.
2. Click **Maxwell** > **Optimetrics Analysis** > **Add Statistical** , or select the **Simulation** tab of the ribbon and, under the Optimetrics icon, select **Statistical** from the drop-down menu:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Setup Statistical Analysis** dialog box appears.

3. Under the **Calculations** tab, type the maximum number of iterations you want Maxwell to perform in the **Maximum Iterations** text box.
4. If you want to specify an initial seed value, select the **Specify initial seed value** check box and enter a positive value in the text box. Each different seed value creates a new statistical sequence.
5. Specify a solution quantity to evaluate.
6. In the **Calculation** text box, set the value at which the solution quantity is to be computed.
7. Optionally, modify the distribution criteria to be used.
8. The following *optional* statistical analysis setup options can also be used:
    - Modify the starting variable value.
    - Exclude variables from the statistical analysis.
    - Modify the values of fixed variables that are not being modified during the statistical analysis.
    - Request that Optimetrics solve a parametric sweep during a statistical analysis.

    > **Note:**
    >
    > Sweeping or using a complex variable is not allowed in any optimetrics setup, including optimization, statistical, sensitivity, and tuning setups.

9. If you want to save the field solution data for the design variations solved during analysis, select **Save Fields**.

Setting the Maximum Iterations for a Statistical Analysis

The **Maximum Iterations** value is the maximum number of design variations Optimetrics solves during a statistical analysis. This value is a stopping criterion; if the maximum number of

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

iterations has been completed, the analysis stops. If the maximum number of iterations has not been completed, Optimetrics continues by performing another iteration, that is, by solving another design variation.

To set the maximum number of iterations for a statistical analysis:

- Under the **Calculations** tab of the *Setup Sensitivity Analysis* dialog box, type a value in the **Maximum Iterations** text box.

Specifying an Initial Seed Value for a Statistical Analysis

The Initial Seed Value is a positive integer used as the starting point during a statistical analysis. When you enter a new seed value, a new statistical sequence is created.

To set an initial seed value for a statistical analysis:

Under the **Calculations** tab of the **Setup Sensitivity Analysis** window, select the **Specify initial seed value** check box and type a positive value in the text box.

Specifying the Solution Quantity to Evaluate for Statistical Analysis

When you add a statistical setup, you can identify one or more solution quantities to evaluate. The solution quantities are specified by mathematical expressions that are composed of basic quantities. You can see the distribution of the solution quantities in the results.

1. In the **Calculations** tab of the *Setup Statistical Analysis* dialog box, click **Setup Calculations**.

   The **Add/Edit Calculations** dialog box is displayed, allowing you to define one or more mathematical expressions for statistical evaluation.

2. In the **Context** section of the dialog box:
   - Select the **Report Type** with a drop-down selection list containing the available types for this design.
   - Select the **Solution** from the drop-down selection list. This lists the available setups and sweeps. As a minimum, the **LastAdaptive** solution is available.
   - Select the **Geometry** from the drop-down selection list or select none (the default). This modifies the list of quantities available to the ones that apply to the specific geometry.
   - When selecting a geometry, you may also be required to specify a point within the geometry where the calculation is to be performed.
3. The **Output Variables** button opens the *Output Variables* dialog box allowing you to create special output variables to be used in the output parameter.
4. The **Calculation Expression** field in the **Trace** tab is used to enter the equation to be used for the solution quantities. To enter an expression, you may type it directly into the field or use the **Category**, **Quantity**, and **Function** lists as follows:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select the **Category**, these depend on the Solution type and the design. This lets you specify the category of information to be used in the output parameter.
- Select a **Quantity** from the list. Available quantities depend upon the Solution type, as well as the Geometry and Category selection. Selecting a Quantity automatically enters it into the Calculation Expression field.
- Select a **Function** to apply to the value in the calculated expression.
- For swept variables, the Range Function button opens the **Set RangebFunction** dialog to apply functions to the expression that apply overbthe sweep range.

5. The **Calculation Range** tab applies to swept variables and allows you to specify the range of the sweep over which to apply the calculation.
6. When the desired **Calculation Expression** has been obtained, click **Add Calculation** to add the entry to the calculation table in the Setup Statistical Analysis dialog box. You may add multiple entries to the table simply by changing the **Calculated Expression** and using the **Add Calculation** button.
7. To update or edit a selected cost function, enter the desired Calculation Expression and click the **Update Calculation** button.
8. Click **Done** to return to the Setup Statistical Analysis dialog box.

> **Note:**
>
> The solution quantity you specify must be able to be evaluated to a single, real number.

Setting the Solution Quantity's Calculation Range

The calculation range of a solution quantity determines the intrinsic variable value at which the solution quantity is extracted. For a statistical setup, the calculation range must be a single value. For a Driven Modal or Driven Terminal design, if you specified that the solution be extracted from the last adaptive solution, Optimetrics uses the adaptive frequency defined in the solution setup. If you specified that the solution quantity be extracted from a frequency sweep solution, Optimetrics will use the starting frequency in the sweep by default. The calculation range should be set during the setup of the solution quantity for statistical evaluation. In order to modify the calculation range, do the following:

1. Under the **Calculations** tab of the *Setup Statistical Analysis* dialog box, click in the **Calculation Range** column of the table for the calculation to be modified.

   The *Edit Calculation Range* dialog box appears.

2. In the table, click **Edit** in the row to be modified.

   If you choose to solve a parametric setup during the statistical analysis, the variables swept in that parametric setup are available in the pop-up list dialog box. If you sweep a

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

variable in the parametric setup that is also a statistics variable, that variable is excluded from the statistics analysis.

Other examples of available variables include frequency, if you selected an S-parameter solution quantity; and phi or theta, if the solution quantity is a radiated field quantity.

3. Click on the value for the calculation range in the list and dismiss the pop-up dialog box.
4. Click **OK** in the *Edit Calculation Range* dialog box to accept the new value for the intrinsic variable and return to the *Setup Statistical Analysis* dialog box.

Setting the Distribution Criteria

For every statistical setup, Optimetrics automatically sets the distribution criteria to be uniform within a 10% tolerance of the variable's starting value. You can modify the distribution type and criteria for a single statistical setup or for every statistical setup.

Overriding the Distribution Criteria for a Single Statistical Setup

To override the default distribution criteria for a single statistical setup:

1. In the *Setup Statistical Analysis* dialog box, click the **Variables** tab.

   All of the variables that were selected for statistical analysis are listed.

2. Check or clear the **Include** check box for each variable to define the specific variables to be varied in the statistical analysis setup.
3. For each included variable, select **Uniform**, **Gaussian**, **Lognormal**, or **User Defined** in the **Distribution** column for the variable you want to override.

   If you changed the distribution type, the **Override** option is now selected. This indicates that the distribution type you selected is to be used for this optimization analysis; the current distribution type selected for the variable in the nominal design is ignored in this statistical analysis.

   Alternatively, you can select the **Override** option first, and then select a different distribution type in the **Distribution** text box.

4. Optionally, if you want to change the distribution criteria, click in **Distribution Criteria** column for the variable you want to override.

   The *Edit Distribution* dialog box appears.

5. If the distribution type is **Gaussian**, do the following:
   a. Type the lower limit of the distribution in the **Cutoff Probability** text box. This is a value =>0 and < 0.1.
   b. Type the mean value of the distribution in the **Mean** text box.
   c. Type the standard deviation of the distribution in the **Std Dev** text box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The design variations are solved using a Gaussian distribution within the specified mean and standard deviation values.

6. If the distribution type is **Uniform**, Enter a tolerance value in the text box.

   The design variations are solved within the tolerance range of the starting value, using an even distribution.

7. If the distribution type is **Lognormal**, do the following:
   a. Enter the cutoff probability in the **Cutoff Probability** text box.
   b. Enter the sigma value of the distribution in the **Sigma** text box and select a unit from the drop-down menu.
   c. Enter the m value of the distribution in the **M** text box.
   d. Enter the theta value in the Theta text box and select a unit from the drop-down menu.
8. If the distribution type is User Defined, do the following:
   a. Enter the cutoff probability in the **Cutoff Probability** text box.
   b. Click **Edit XY Data** to open the *Edit Datasets* dialog box in which you can select an existing dataset, or create a new one.
9. By default, all variables are set to sample using **Latin Hypercube** sampling. This sampling method provides for greater variability than random sampling by keeping track of chosen samples and guaranteeing that samples cannot be repeated. You may revert to random sampling by clearing the check box in the **Latin Hypercube** column for any desired variable.
10. Click **OK**.

To revert to the default distribution settings, clear the **Override** option.

Changing the Distribution Criteria for Every Statistical Setup

To change the default distribution criteria for every statistical setup:

1. Make sure that the variable's distribution criteria are not being overridden in any statistical setup.
2. If the variable is a design variable, do the following: On **Maxwell** > **Design Properties**, select **Statistics**.

   If the variable is a project variable, do the following: Click **Project** > **Project Variables**.

   The *Properties* dialog box appears.

3. Select **Statistics**.
4. Click in the **Distribution** column for the variable you want to change, and then select **Uniform**, **Gaussian**, **Lognormal**, or **User Defined**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Optionally, if you want to change the distribution criteria, click in the **Distribution Criteria** column for the variable you want to change.

   If the distribution type is **Gaussian**, the *Gaussian Distribution* dialog box appears. If the distribution type is **Uniform**, the *Uniform Distribution* dialog box appears.

6. If the distribution type is **Gaussian**, do the following:
   a. Type the lower limit of the distribution in the **Cutoff Probability** text box. This is a value =>0 and < 0.1.
   b. Type the mean value of the distribution in the **Mean** text box.
   c. Type the standard deviation of the distribution in the **Std Dev** text box.

   The design variations are solved using a Gaussian distribution within the specified mean and standard deviation values.

7. If the distribution type is **Uniform**, do the following:
   a. Type a cutoff probability value in the **Cutoff Probability** text box.
   b. Type mean and tolerance values in the corresponding text boxes.

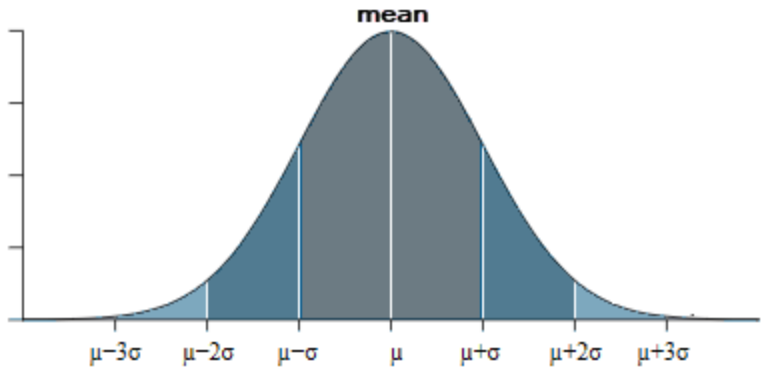
   The design variations are solved within the tolerance range of the starting value, using an even distribution.

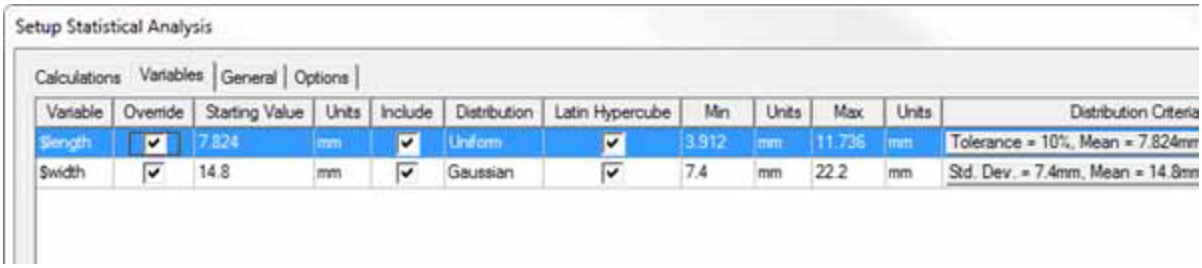
8. If the distribution type is **Lognormal**, do the following:
   a. Type a cutoff probability value in the **Cutoff Probability** text box.
   b. Type values for Sigma, M, and Theta in the corresponding text boxes.
9. If the distribution type is **User Defined**, do the following:
   a. Type a cutoff probability value in the **Cutoff Probability** text box.
   b. Click **Edit XY Data** to open the *Edit Dataset* dialog box.
   c. Either type or import the X and Y data values for the distribution in the *Edit Dataset* dialog box.
10. Click **OK**.

Statistical Cutoff Probability

The cutoff probability values affects the Gaussian distribution criteria. This is a value =>0 and < 0.1. The design variations are solved using a Gaussian distribution using a lower limit cutoff probability and specified mean and standard deviation values.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Uniform distributions such as **length** variable above use only the **Tolerance** value and do not have a cutoff probability.

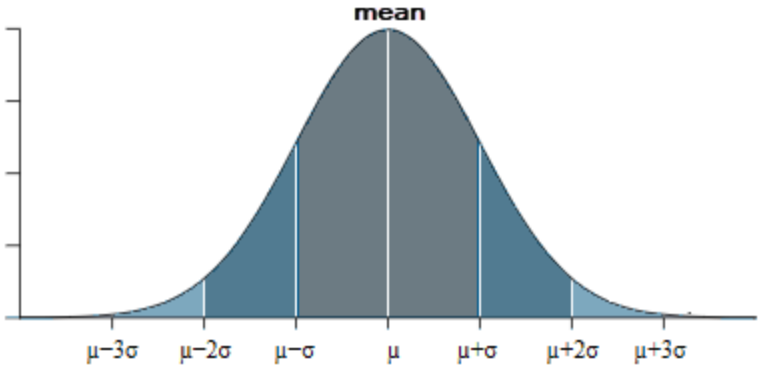
Edit Distribution

When setting the distribution type for a variable, you have the option of changing the distribution parameters from the default values.

1. If the distribution type is **Gaussian**, do the following:
    a. Type the lower limit of the distribution in the **Cutoff Probability** text box. This is a value =>0 and < 0.1.
    b. Type the mean value of the distribution in the **Mean** text box.
    c. Type the standard deviation of the distribution in the **Std Dev** text box.

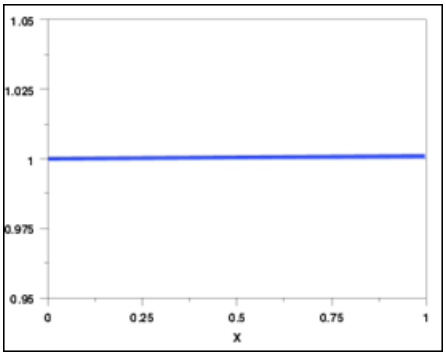
    The design variations are solved using a Gaussian distribution within the specified mean and standard deviation values.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. If the distribution type is **Uniform**, do the following:
    a. Type the lower limit of the distribution in the **Cutoff Probability** text box.
    b. Type the mean value of the distribution in the **Mean** text box.
    c. Enter the tolerance in the **Tolerance** text box.

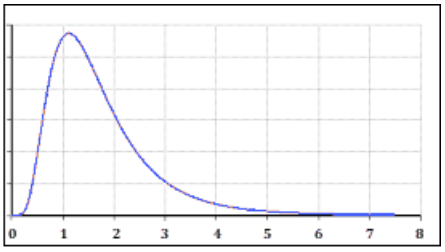
    The design variations are solved within the tolerance range of the starting value, using an even distribution.



3. If the distribution type is **Lognormal**, do the following:
    a. Type the lower limit of the distribution in the **Cutoff Probability** text box.
    b. Enter the shape parameter of the distribution in the **Sigma** text box.
    c. Enter the scale parameter in the **M** text box. The scale parameter should be set to 1 for the standard lognormal distribution.
    d. Enter the location parameter value for **Theta** in the text box. The value for a standard lognormal distribution is 0.

    The design variations are solved with a logarithmic distribution using the shape, scale and location parameters provided.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. If the distribution type is **User Defined**, do the following:
    a. Type the lower limit of the distribution in the **Cutoff Probability** text box.
    b. Select the **Edit XY Data** button to manually define the data distribution using datasets.

Modifying the Starting Variable Value for Statistical Analysis

A variable's starting value is the first value that is solved during the statistical analysis. Optimetrics automatically sets the starting value of a variable to be the current value set for the nominal design. You can modify this value for each statistical setup.

> **Warning:**
>
> Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any optimetric analysis.

1. In the *Setup Statistical Analysis* dialog box, click the **Variables** tab.

   All of the variables selected for the statistical analysis are listed.

2. Type a new value in the **Starting Value** text box for the value you want to override, and then press **Enter**.

   The **Override** option is now selected. This indicates that the value you entered is to be used for this statistical analysis; the current value set for the nominal model will be ignored.

   Alternatively, you can select the **Override** option first, and then type a new variable value in the **Starting Value** text box.

3. Optionally, click a new unit system in one of the **Units** text boxes.

To revert to the default starting value, clear the **Override** option.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solving a Parametric Setup During a Statistical Analysis

Solving a parametric setup during a statistical analysis is useful when you want Optimetrics to solve every design variation in the parametric setup at each statistical analysis iteration.

To solve a parametric setup during a statistical analysis:

1. In the *Setup Statistical Analysis* dialog box, click the **General** tab.
2. Click the parametric setup you want Optimetrics to solve during the statistical analysis from the **Parametric Analysis** drop-down menu.
3. Select **Solve the parametric sweep during analysis**.

Viewing Distribution Results for a Statistical Analysis

1. In the project tree, right-click the statistical setup for which you want to view the distribution results calculated for the solution quantities, and select **View Analysis Result**.

   The **Post Analysis Display** dialog box appears.

2. Select the statistical setup with the results you want to view from the drop-down menu at the top of the dialog box.
3. To view the results in tabular form, select **Table** as the view type.

   The distribution results for the selected solution quantities are listed in table format for each solved design variation.

4. Optionally, click a design variation in the table, and then click **Apply** (at the far right side of the dialog box).

   The design displayed in the **Modeler** window is changed to represent the selected design variation.

5. To view the results in graphic format, select **Plot** as the view type.
6. Type the number of bins you want to plot on the x-axis.
7. Select the solution quantity for which you want to plot distribution results on the y-axis from the **Y** drop-down menu.

   A histogram plot appears in the **Post Analysis Display** dialog box. It displays the distribution of the selected solution quantity.

8. Optionally, click a design variation in the table, and then click **Apply** (at the far right side of the dialog box).

   The software now points to the selected design variation as the nominal solution and as a result, the design displayed in the **Modeler** window changes to represent the selected design variation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Click **Revert** to return the design in the view window to the original value.

Plotting Distribution Results for a Statistical Analysis

1. In the project tree, right-click the statistical setup for which you want to view the distribution results calculated for the solution quantities, and select **View Analysis Result**.

   The **Post Analysis Display** dialog box appears.

2. Select the statistical setup with the results you want to view from the drop-down menu at the top of the dialog box.
3. If it is not already selected, select **Plot** as the view type.
4. Type the number of bins you want to plot on the x-axis.
5. Select the solution quantity for which you want to plot distribution results on the y-axis from the **Y** drop-down menu.

   A histogram plot appears in the **Post Analysis Display** dialog box. It displays the distribution of the selected solution quantity.

Using Design of Experiments

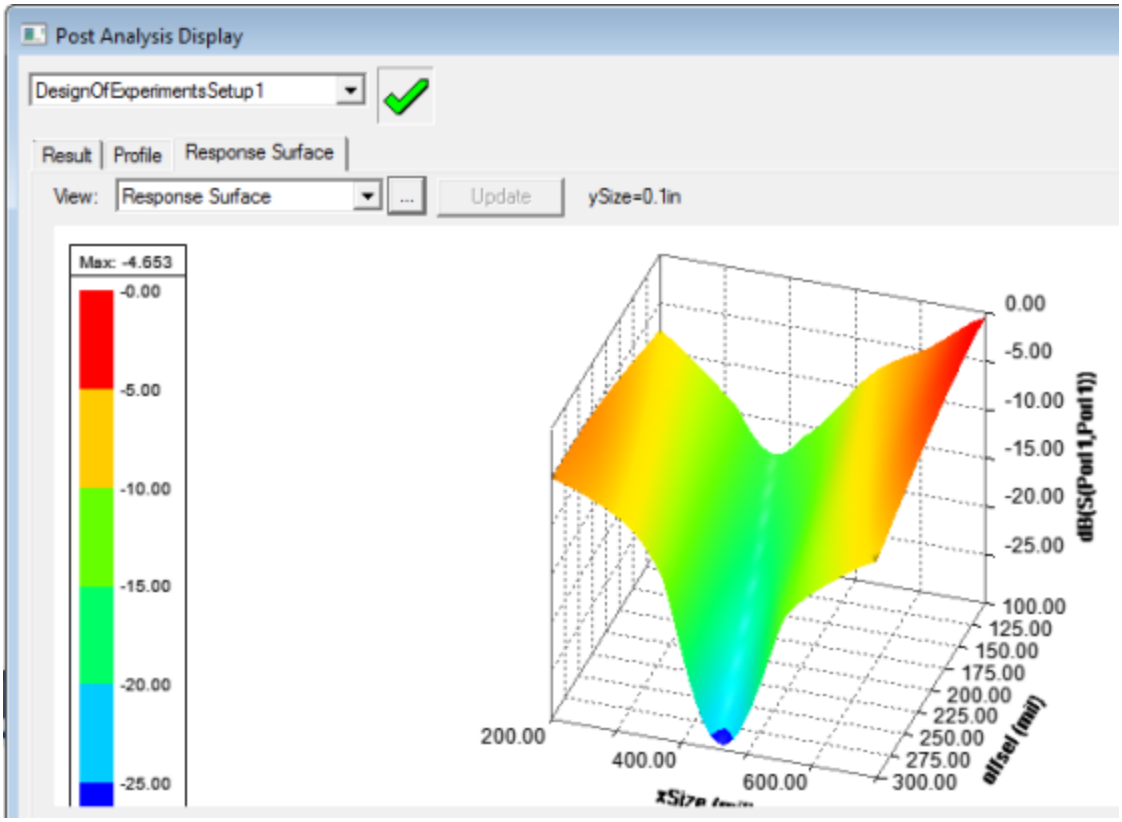
Design of Experiments (DOE) is a technique used to scientifically determine the location of sampling points and is included as part of the Response Surface, Goal Driven Optimization, and Analysis systems. Design of Experiments plus a mathematical approximation of output parameters lets you

- reduce the number of simulations
- interactively explore the design space before running optimization

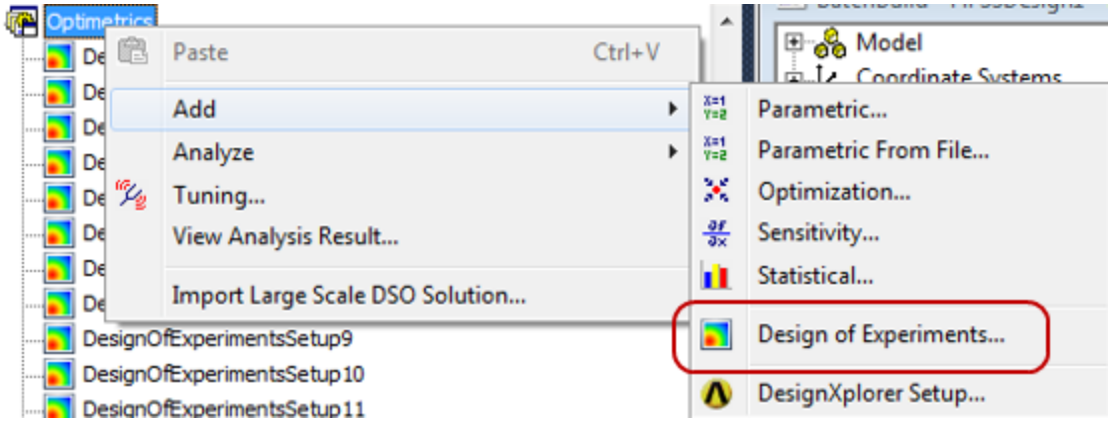
Design of Experiments describes the relationship between the design variables and the performance of the product by using Design of Experiments (DOE), combined with response surfaces. DOE and response surfaces provide all of the information required to achieve Simulation Driven Product Development. Once the variation of the performance with respect to the design variables is known, it becomes easy to understand and identify all changes required to meet the requirements for the product.

The goal is to create a response surface by interpolating through calculated points (a best curve fit). For each design, you can create a response surface for each output parameter. Once the response surfaces are created, you can share the information can in easily understandable terms: curves, surfaces, sensitivities, etc. They can be used at any time during the development of the product without requiring additional simulations to test a new configuration.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

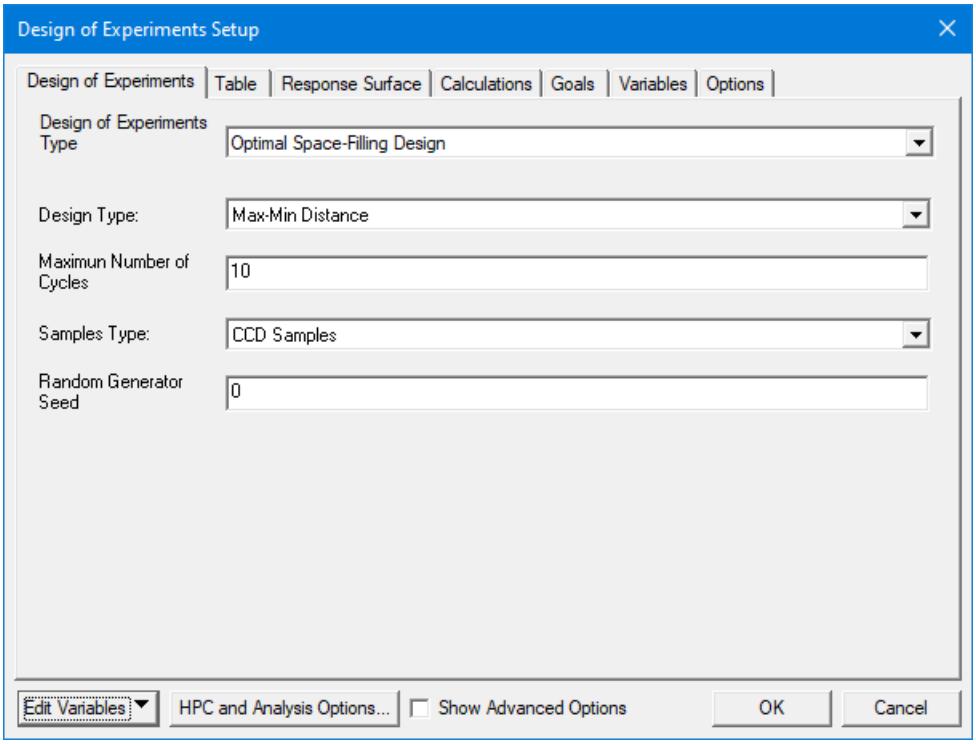


The Design of Experiments feature is integrated inside Electronics Desktop. Combined with Electronics Desktop‘s distributed solve feature, you can build the response surfaces from the DOE variation table much faster.



Selecting **Design of Experiments** under **Optimization** opens a window with several tabs:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the Design of Experiments setup, you select the DOE type, select the Response Surface, specify goals, view and include variables.

There are a wide range of DOE algorithms or methods available in engineering literature. These techniques all have one common characteristic: they try to locate the sampling points such that the space of random input parameters is explored in the most efficient way, or obtain the required information with a minimum of sampling points. Sample points in efficient locations only reduce the required number of sampling points and increases the accuracy of the response surface generated. For more information on the available types of DOE, see Design of Experiments Types.

Once you have set up your input parameters, you can update the DOE, which submits the generated design points to the analysis system for solution. Design points are solved simultaneously if the analysis system is set up to do so; sequentially, if not. After the solution is complete, you can update the Response Surface cell, which generates response surfaces for each output parameter based on the data in the generated design points.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Requirements and recommendations regarding the number of input parameters vary according to DOE type. For more information, see Number of Input Parameters for DOE Types.

If you change the Design of Experiments type after doing an initial analysis and preview the Design of Experiments Table, any design points generated for the new algorithm that are the same as design points solved for a previous algorithm will appear as up-to-date. Only the design points that are different from any previously submitted design points need to be solved.

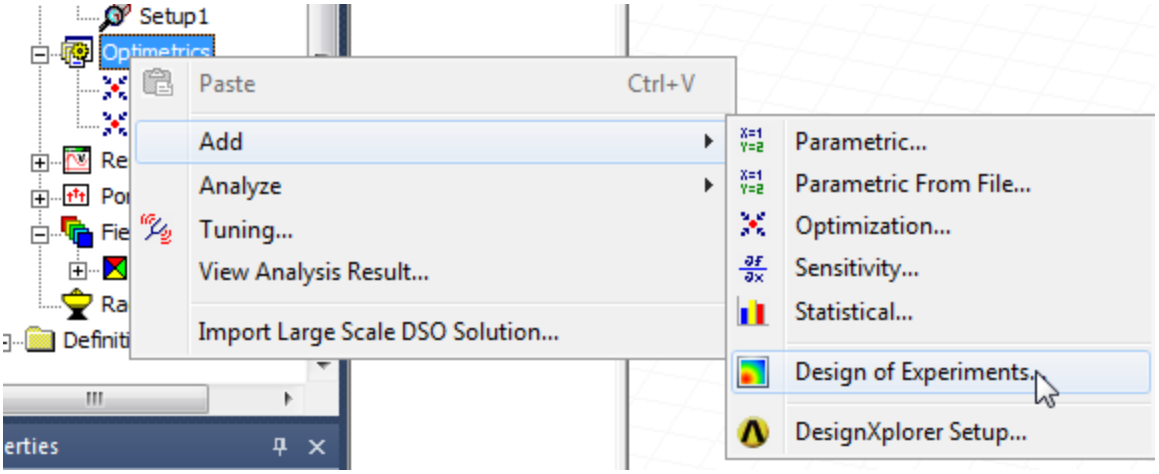
You should set up your DOE Properties before generating your DOE Design Point matrix. The following topics describe setting up and solving your Design of Experiments, and viewing the results.

Setting Up Design of Experiments in Ansys Electronics Desktop

The process for setting up a Design of Experiments analysis in Ansys Electronics Desktop follows:

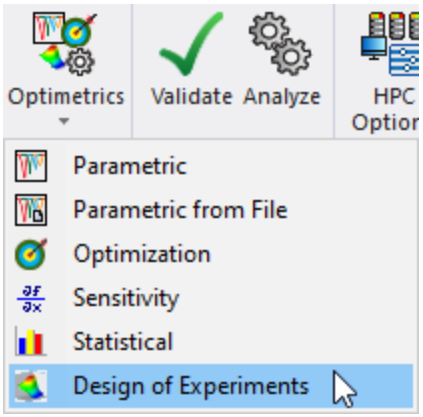
Add the Design of Experiments Setup

Right-click the Optimetrics icon in the Project tree and select **Add** > **Design of Experiments**. You can also use **Maxwell** > **Optimetrics Analysis** > **Add Design of Experiments**.

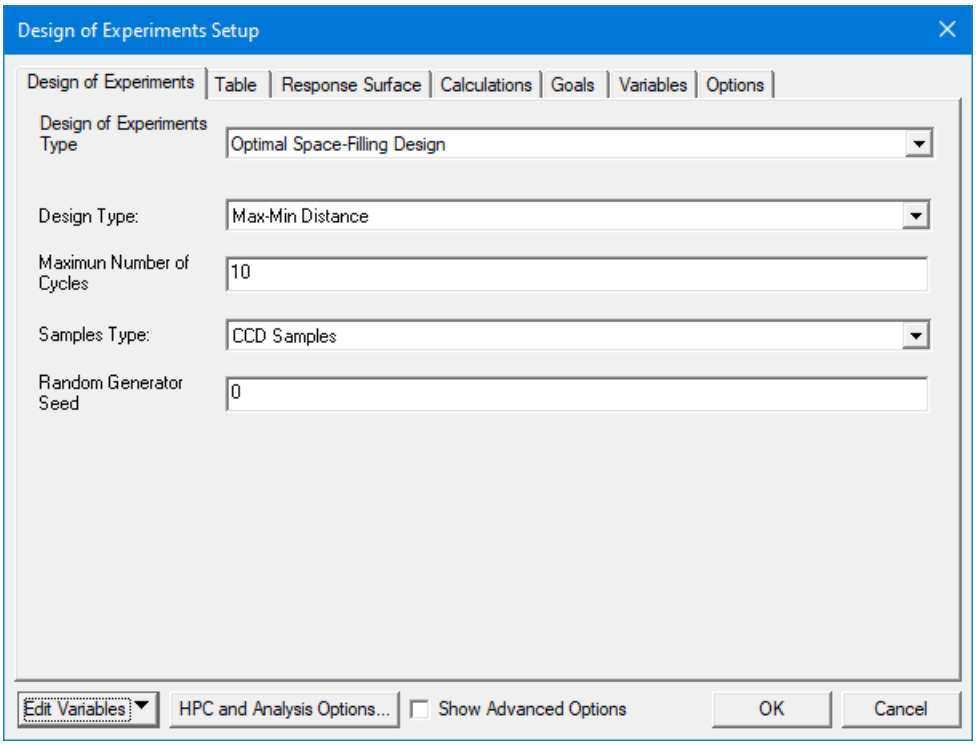


You can also select the **Simulation** tab and under the Optimetrics icon, select Design of Experiments from the drop-down menu:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Design of Experiments Setup** window appears.



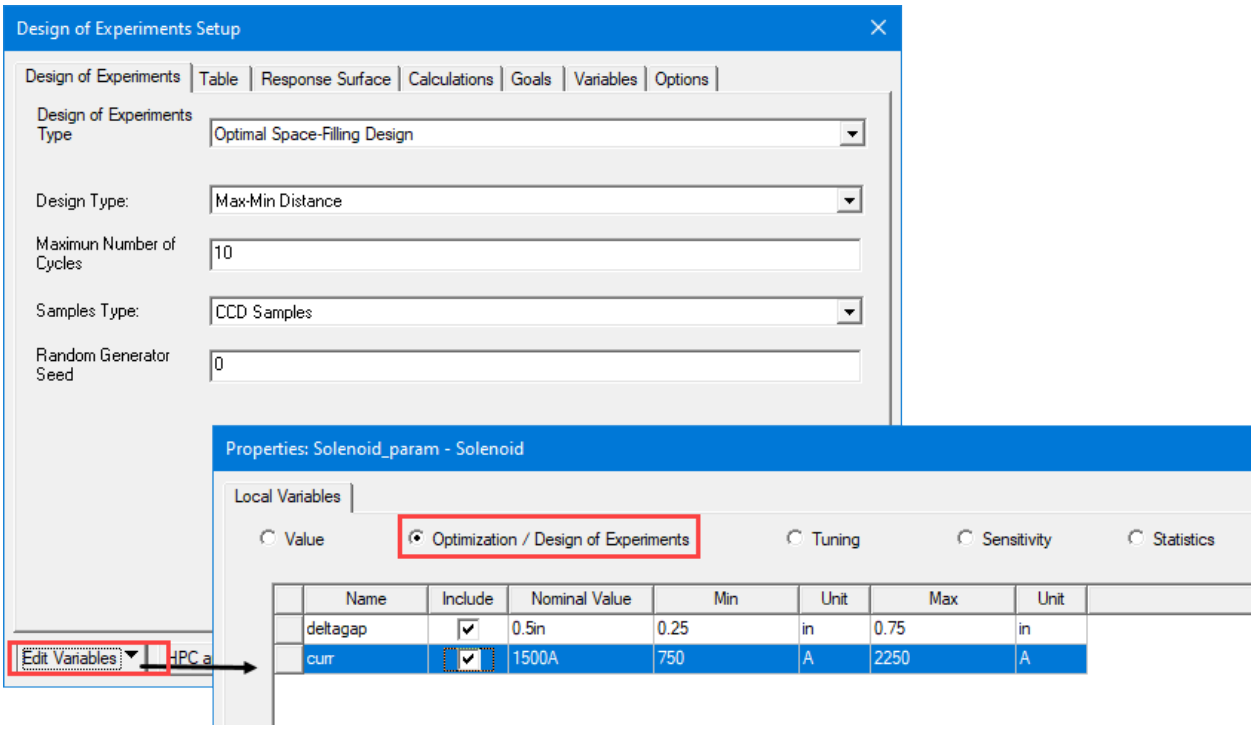
It has tabs for

- Design of Experiments, where you specify the sampling points and associated parameters.
- Table, which shows the sampling points defined by the Design of Experiments settings you provide. If you select Custom and the Design of Experiments type, you can import data files as well as add or edit rows. You can also Export files of DOE sampling points you have defined by any method.

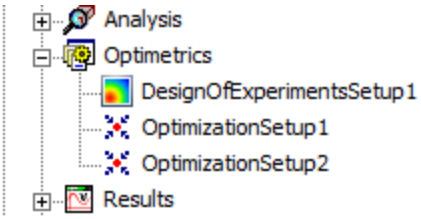
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Response Surface, which specifies the Response surface type, and Refinement parameters.
- Calculations, which accesses Optimetrics calculations.
- Goals, including cost functions calculations and norm type.
- Variables, previously defined for Optimization/Design of Experiments as Project or Design variables, and whether to include them, treat as discrete, whether to Use Manufacturable Variables, and the available Levels.
- Options, to Save Fields and Mesh, and whether to copy geometrically equivalent meshes.

You must define local or project variables as Optimization /Design of Experiments variables for the Design of Experiments setup to include the variable.

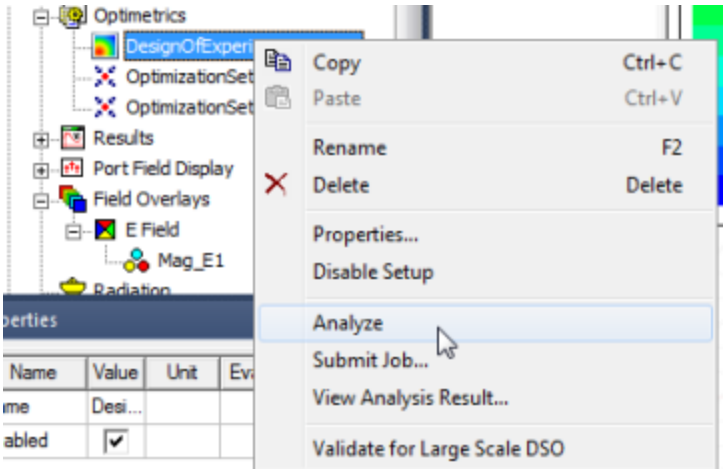


Once you have set parameters and click **OK**, the Design of Experiments setup appears under the Optimetrics icon in the Project tree.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

From here you can right-click the DesignOfExperiments setup from the shortcut menu to run **Analyze**, **Submit Job...** or **Validate for Large Scale DSO**.

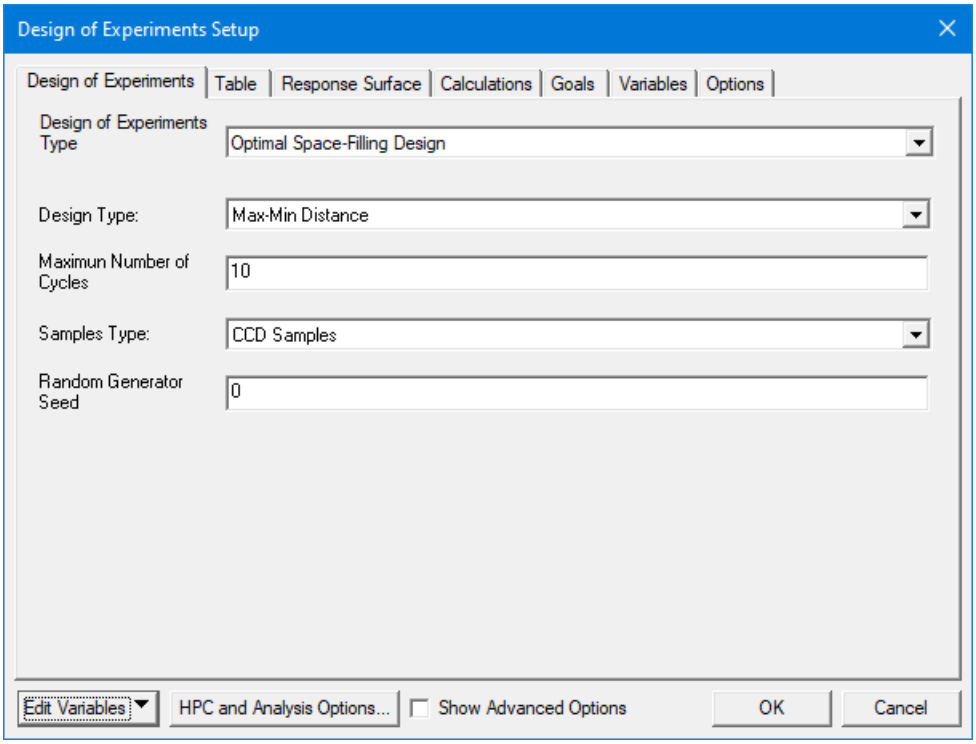


Stand-alone and distributed simulation will have the same behavior as the parametric setup.

Design of Experiments Tab

The Design of Experiments tab in the DOE Setup includes selections for defining the sampling points that define your experiment. Each selection for Design of Experiments type has a different set of associated parameters so the appearance of the dialog changes to show the parameters for your selection.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The goal in Design of Experiments is to determine the smallest sufficient set of points required to calculate a response surface. Therefore, you choose the type depending on the parametric problem and targeted response surface. The number of points depends on the number of input parameters, or is user defined

The Design of Experiments Types available in the Desktop include the following:

Design of Experiments Types	Brief Description (see links for more details)
Optimal Space Filling (Default)	An optimized Latin Hypercube Sampling maximizing distance between experiments. Several design type criteria are available: • Max-Min Distance • Centered L2 • Maximum Entropy Sampling type determines the number of samples in the design: • CCD samples (Central Composite Designs are five level factorial designs that are suitable for calibrating the quadratic response model) • Linear model samples

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Design of Experiments Types	Brief Description (see links for more details)
	• Pure quadratic model samples • Full quadratic model samples • User-Defined samples You also specify a Maximum Number of Cycles and a Random Generator Seed.
Central Composite Design	Several design types available: • Face-Centered • Rotatable • VIF-Optimality • G-Optimality • Auto-defined You can also choose a Standard or Enhanced Template.
Box-Behnken	Avoids critical configurations in the corner of the design space. Maximum number of input parameters is 12.
Custom	Lets you customize a DOE matrix, by editing values, adding or removing samples, and/or importing samples from a CSV file. Selecting Custom enables an Import button and the **Table** tab, as well as buttons to **Add** editable rows or **Delete** selected rows. If you previously solved the DOE using one of the other algorithms, those design points are retained and you can add new design points to the table. You can also import and export design points into the custom DOE Table from the Parameter Set.
Latin Hypercube Sampling	Statistical design where no two experiments share input parameters of the same value. The samples Type can be • CCD Samples • Linear Model Samples • Pure Quadratic Model Samples • Full Quadratic Model Samples • User-Defined Samples, for which you also specify the Number of Samples. For each samples type you also specify a Random Generator Seed.

The Table tab provides a preview view of the design points defined by your selections.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Optimal Space Filling Design (OSF)

The goal in Design of Experiments is to determine the smallest sufficient set of points required to calculate a response surface. Therefore, you choose the type depending on the parametric problem and targeted response surface. The number of points depends on the number of input parameters, or is user defined.

Optimal Space-Filling Design (OSF) creates optimal space filling Design of Experiments (DOE) plans according to some specified criteria. Essentially, OSF is a Latin Hypercube Sampling Design (LHS) that is extended with post-processing. It is initialized as an LHS and then optimized several times, remaining a valid LHS (without points sharing rows or columns) while achieving a more uniform space distribution of points (maximizing the distance between points).

To offset the noise associated with physical experimentation, classical DOE types such as CCD focus on parameter settings near the perimeter of the design region. Because computer simulation is not quite as subject to noise, though, the Optimal Space-Filling (OSF) design is able to distribute the design parameters equally throughout the design space with the objective of gaining the maximum insight into the design with the fewest number of points. This advantage makes it appropriate when a more complex meta-modeling technique such as Kriging, Non-Parametric Regression, or Neural Networks is used.

OSF shares some of the same disadvantages as LHS, though to a lesser degree. Possible disadvantages of an OSF design are:

- When the CCD Samples sample type is selected, a maximum of 20 input parameters is supported.
- Extremes, such as the corners of the design space, are not necessarily covered.
- The selection of too few design points can result in a lower quality of response prediction.

The following properties are available for the OSF DOE type.

- **Design Type**: The following choices are available:
  - **Max-Min Distance** (default): Maximizes the minimum distance between any two points. This strategy ensures that no two points are too close to each other. For a small size of sampling (N), the Max-Min Distance design generally lies on the exterior of the design space and fill in the interior as N becomes larger. Generally, this is the faster algorithm.
  - **Centered L2**: Minimizes the centered L2-discrepancy measure. The discrepancy measure corresponds to the difference between the empirical distribution of the sampling points and the uniform distribution. This means that the centered L2 yields a uniform sampling. This design type is computationally faster than the **Maximum Entropy** type.
  - **Maximum Entropy**: Maximizes the determinant of the covariance matrix of the sampling points to minimize uncertainty in unobserved locations. This option often

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

provides better results for highly correlated design spaces. However, its cost increases non-linearly with the number of input parameters and the number of samples to be generated. Thus, it is recommended only for small parametric problems.

- **Maximum Number of Cycles**: Determines the number of optimization loops the algorithm needs, which in turns determines the discrepancy of the DOE. The optimization is essentially combinatorial, so a large number of cycles slows down the process. However, this makes the discrepancy of the DOE smaller. For practical purposes, 10 cycles is generally good for up to 20 variables. The value must be greater than 0. The default is 10.
- **Samples Type**: Determines the number of DOE points the algorithm should generate. This option is suggested if you have some advanced knowledge about the nature of the metamodel. The following choices are available:
  - **CCD Samples** (default): Supports a maximum of 20 inputs. Generates the same number of samples a CCD DOE would generate for the same number of inputs. You can use this to generate a space filling design that has the same cost as a corresponding CCD design.
  - **Linear Model Samples**: Generates the number of samples as needed for a linear metamodel.
  - **Pure Quadratic Model Samples**: Generates the number of samples as needed for a pure quadratic metamodel (no cross terms).
  - **Full Quadratic Samples**: Generates the number of samples needed to generate a full quadratic model.
  - **User-Defined Samples**: Specify the desired number of samples.
- **Random Generator Seed**: Set the value used to initialize the random number generator invoked internally by the LHS algorithm. Although the generation of a starting point is random, the seed value consistently results in a specific LHS. This property allows you to generate different samplings by changing the value or regenerate the same sampling by keeping the same value. The default is 0.
- **Number of Samples**: Enabled when **Samples Type** is set to **User-Defined Samples**. Specifies the default number of samples. The default is 10.

Central Composite Design (CCD)

The goal in Design of Experiments is to determine the smallest sufficient set of points required to calculate a response surface. Therefore, you choose the type depending on the parametric problem and targeted response surface. The number of points depends on the number of input parameters, or is user defined.

Central Composite Design (CCD) provides a screening set to determine the overall trends of the metamodel to better guide the choice of options in Optimal Space-Filling Design. The CCD DOE type supports a maximum of 20 input parameters.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following properties are available for the CCD DOE type.

- **Design Type**: By specifying the **Design Type** for CCD, you can help to improve the response surface fit for DOE studies. For each CCD type, the alpha value is defined as the location of the sampling point that accounts for all quadratic main effects. The following CCD design types are available:
  - **Face-Centered**: A three-level design with no rotatability. The alpha value equals 1.0. A Template Type setting automatically appears, with Standard and Enhanced options. Choose Enhanced for a possible better fit for the response surfaces.
  - **Rotatable**: A five-level design that includes rotatability. The alpha value is calculated based on the number of input variables and a fraction of the factorial part. A design with rotatability has the same variance of the fitted value regardless of the direction from the center point.
  - **VIF-Optimality**: A five-level design in which the alpha value is calculated by minimizing a measure of non-orthogonality known as the Variance Inflation Factor (VIF). The more highly correlated the input variable with one or more terms in a regression model, the higher the VIF.
  - **G-Optimality**: Minimizes a measure of the expected error in a prediction and minimizes the largest expected variance of prediction over the region of interest.
  - **Auto-Defined**: Design exploration automatically selects the Design Type based on the number of input variables. Use of this option is recommended for most cases as it automatically switches between the G-Optimality if the number of input variables is 5 or VIF-Optimality otherwise.
    However, you can use the Rotatable design if the default option does not provide good values for the Goodness of Fit from the response surface plots. Additionally, you can use the Enhanced template if the default Standard template does not fit the response surfaces well.
- **Template Type**: Enabled for the Rotatable and Face-Centered design types. The following options are available:
  - **Standard**
  - **Enhanced**: Choose this option for a possible better fit for the response surfaces

Box Behnken Design (CCD)

The goal in Design of Experiments is to determine the smallest sufficient set of points required to calculate a response surface. Therefore, you choose the type depending on the parametric problem and targeted response surface. The number of points depends on the number of input parameters, or is user defined.

A **Box-Behnken Design** is a three-level quadratic design that does not contain fractional factorial design. The sample combinations are treated in such a way that they are located at

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

midpoints of edges formed by any two factors. The design is rotatable (or in cases, nearly rotatable).

One advantage of a Box-Behnken design is that it requires fewer design points than a full factorial CCD and generally requires fewer design points than a fractional factorial CCD. Additionally, a Box-Behnken Design avoids extremes, allowing you to work around extreme factor combinations. Consider using the Box-Behnken Design DOE type if your project has parametric extremes (for example, has extreme parameter values in corners that are difficult to build). Because the Box-Behnken DOE doesn't have corners and does not combine parametric extremes, it can reduce the risk of update failures.

Possible disadvantages of a Box-Behnken design are

- Prediction at the corners of the design space is poor and that there are only three levels per parameter.
- A maximum of twelve input parameters is supported.

No additional properties are available for the Box-Behnken Design DOE type.

Custom DOE Type

The goal in Design of Experiments is to determine the smallest sufficient set of points required to calculate a response surface. Therefore, you choose the type depending on the parametric problem and targeted response surface. The number of points depends on the number of input parameters, or is user defined.

The **Custom DOE** type allows for definition of a custom DOE Table. You can manually add new design points, entering the input and (optionally) output parameter values directly into the table. If you previously solved the DOE using one of the other algorithms, those design points are retained and you can add new design points to the table. You can also import and export design points into the custom DOE Table from the Parameter Set.

You can change the edition mode of the DOE table to edit the output parameter values. You can also copy and paste data and import data from a CSV file by right-clicking and selecting Import Design Points.

Latin Hypercube Sampling

The goal in Design of Experiments is to determine the smallest sufficient set of points required to calculate a response surface. Therefore, you choose the type depending on the parametric problem and targeted response surface. The number of points depends on the number of input parameters, or is user defined.

In the **Latin Hypercube Sampling** Design DOE type, the DOE is generated by the LHS algorithm, an advanced form of the Monte Carlo sampling method that avoids clustering samples. In a Latin Hypercube Sampling, the points are randomly generated in a square grid

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

across the design space, but no two points share the same value. This means that no point shares a row or a column of the grid with any other point.

Possible disadvantages of an LHS design are

- When the CCD Samples sample type is selected, a maximum of 20 input parameters is supported. For more information, see Number of Input Parameters for DOE Types.
- Extremes, such as the corners of the design space, are not necessarily covered. Additionally, the selection of too few design points can result in a lower quality of response prediction.

> **Note:**
>
> The Optimal Space-Filling Design DOE type is an LHS design that is extended with post-processing.

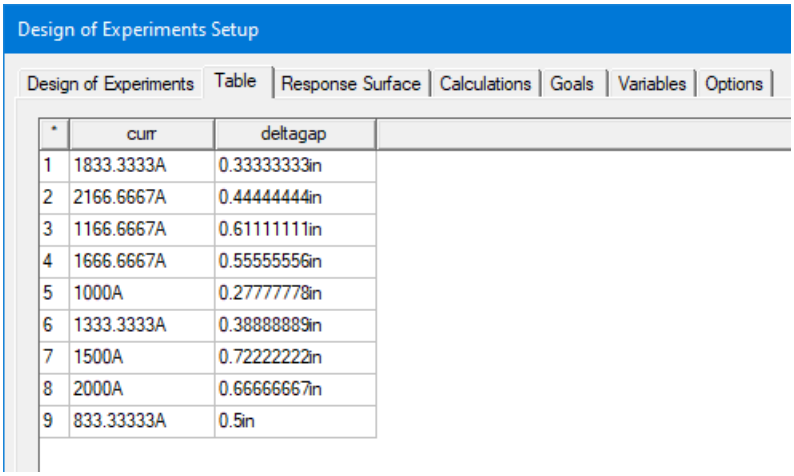
The following properties are available for the LHS DOE type:

- **Samples Type**: Determines the number of DOE points the algorithm should generate. This option is suggested if you have some advanced knowledge about the nature of the metamodel. The following choices are available:
  - **CCD Samples** (default): Supports a maximum of 20 inputs. Generates the same number of samples a CCD DOE would generate for the same number of inputs. You can use this to generate a space filling design that has the same cost as a corresponding CCD design.
  - **Linear Model Samples**: Generates the number of samples as needed for a linear metamodel.
  - **Pure Quadratic Model Samples**: Generates the number of samples as needed for a pure quadratic metamodel (no cross terms).
  - **Full Quadratic Samples**: Generates the number of samples needed to generate a full quadratic model.
  - **User-Defined Samples**: Specify the desired number of samples.
- **Random Generator Seed**: Set the value used to initialize the random number generator invoked internally by the LHS algorithm. Although the generation of a starting point is random, the seed value consistently results in a specific LHS. This property allows you to generate different LHS samplings (by changing the value) or to regenerate the same LHS sampling (by keeping the same value). The default is 0.
- **Number of Samples**: Enabled when **Samples Type** is set to **User-Defined Samples**. Specifies the default number of samples. The default is 10.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

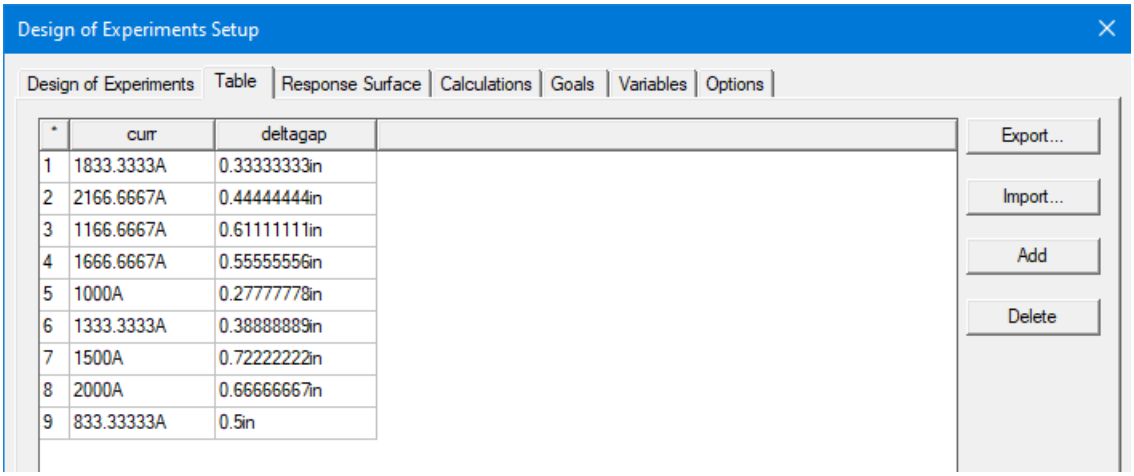
Table Tab for Design of Experiments

The **Table** tab for the **Design of Experiments Setup** window displays a preview of the design points designed by your selections on the **Design of Experiments** tab. There is one column for each defined variable. The **Export** button lets you create a file of the table in a format you specify.



*	curr	deltagap
1	1833.3333A	0.33333333in
2	2166.6667A	0.44444444in
3	1166.6667A	0.61111111in
4	1666.6667A	0.55555556in
5	1000A	0.27777778in
6	1333.3333A	0.38888889in
7	1500A	0.72222222in
8	2000A	0.66666667in
9	833.33333A	0.5in

If you have specified Custom as the Design of Experiments Type, the table is editable, and the **Table** tab includes buttons for **Add** and **Delete** rows. All rows are editable. You can add new rows by entering values in the * row of the table. You enter values in the input parameter columns. Once you have entered a value in one column in the * row, the row is added to the table and the values for the remaining input parameters are set to the initial values of the parameters. You can then edit that row in the table and change any of the other input parameter values if needed. Output parameter values are then calculated when the design is solved updated.



*	curr	deltagap
1	1833.3333A	0.33333333in
2	2166.6667A	0.44444444in
3	1166.6667A	0.61111111in
4	1666.6667A	0.55555556in
5	1000A	0.27777778in
6	1333.3333A	0.38888889in
7	1500A	0.72222222in
8	2000A	0.66666667in
9	833.33333A	0.5in

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Depending on the context, the tables are read-only and filled automatically, or they are partially or completely editable. The background color of a cell indicates if it is editable or not:

- A gray background indicates a read-only cell
- A white background indicates an editable cell

Output parameter values calculated from a simulation (a design point update) are displayed in black text.

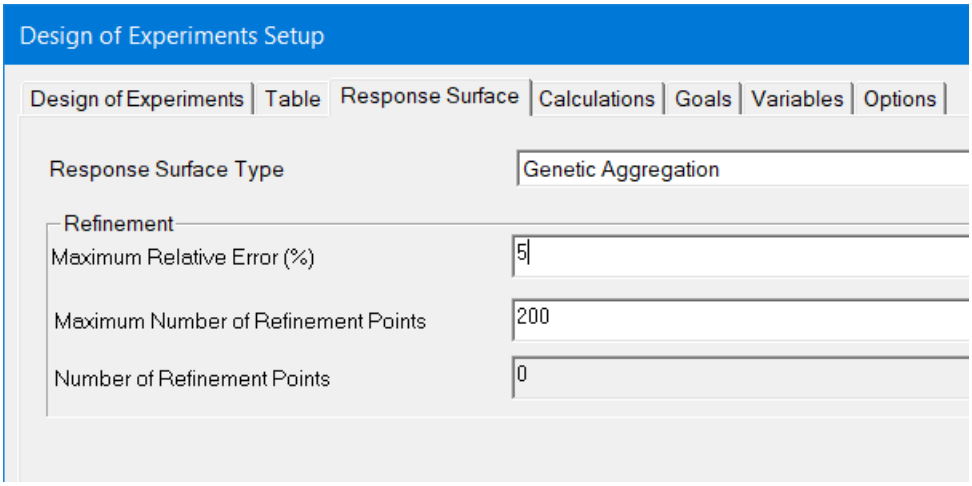
The Custom Table view also includes an Import button. Import and Export files can be

- Comma delimited data files, (*.csv)
- Tab delimited data files, (*.tab)
- Ansoft Plot data files, (*.dat)
- Post Processor format data files, (*.txt)

The Table updates automatically when you change your selections on the **Design of Experiments** tab.

Response Surface Tab for Design of Experiments

The Response Surface tab for the **Design of Experiments Setup** window lets you select the Response Surface type as **Genetic Aggregation** or **Standard Response Surface**.



The Response Surface type selection specifies the refinement applied to the initial Design of Experiments. The Genetic Aggregation setting finds the best possible response surface for each output automatically by combining

- Metamodels
- Settings

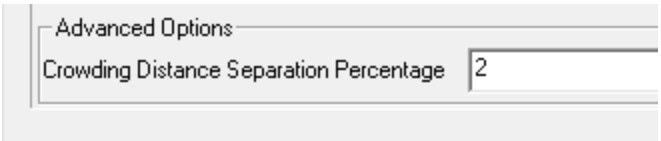
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Kernel Variation
- Polynomial Regression

For each output, a fitness factor works to minimize error, including cross-validation errors. The automatic refinement adds design points to the DOE until the response surface accuracy meets user requirements. You can specify requirements for

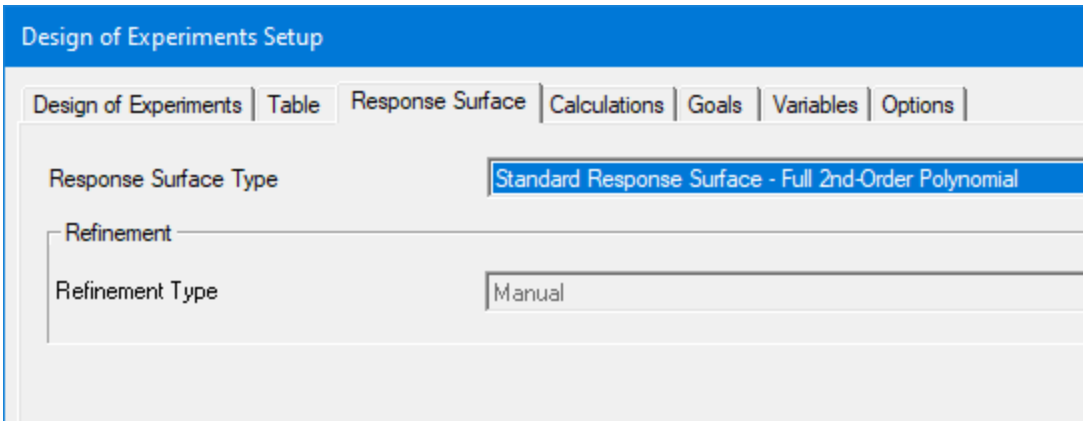
- **Maximum Relative Error %**: This apply to all output calculations. Empty this field to turn off Auto refinement. (Note: Workbench does not have this field.)
- **Maximum Number or Refinement Points**: Determines the maximum number of refinement points that can be generated for use with the Genetic Aggregation algorithm.
- **Number of Refinement Points**: Read-only property indicating the number of existing refinement points.

If you check **Show Advanced Options** at the bottom of the **Setup** window, you can also specify **Crowding Distance Separation Percentage**:



This setting defines the minimum allowable distance between new refinement points; it is implemented as a constraint in the search for refinement points.

You do have the option of selecting **Standard Response Surface- Full 2nd Order Polynomial**:

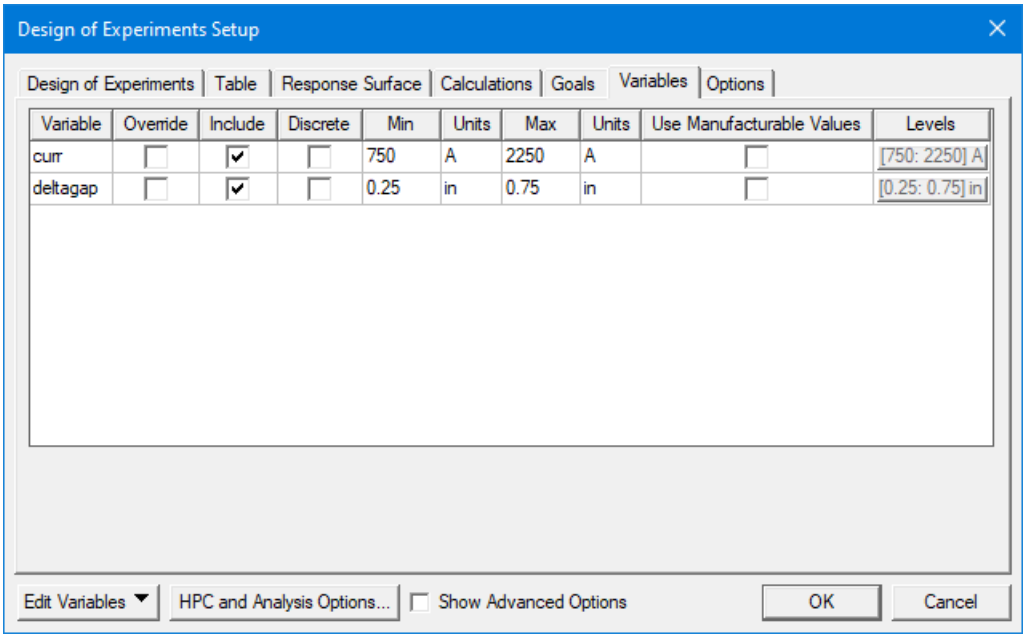


After you have completed an analysis, you can view the generated plot.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

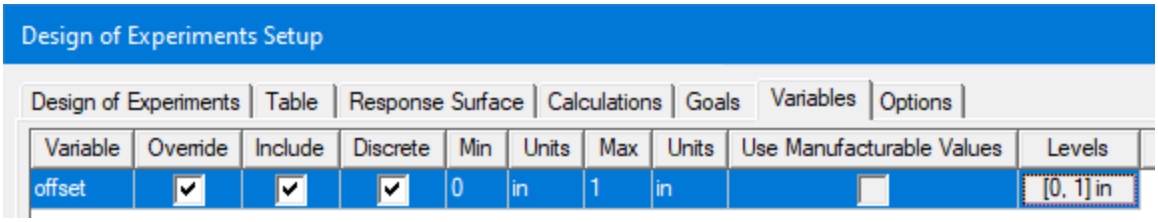
Variables Tab for Design of Experiments

The Variables tab for the **Design of Experiments Setup** window displays list of the variables defined for the Design and Project as Optimetrics/Design of Experiments variables.



The columns list the Variable names, the current Min and Max Values and Units, and provides options for the following:

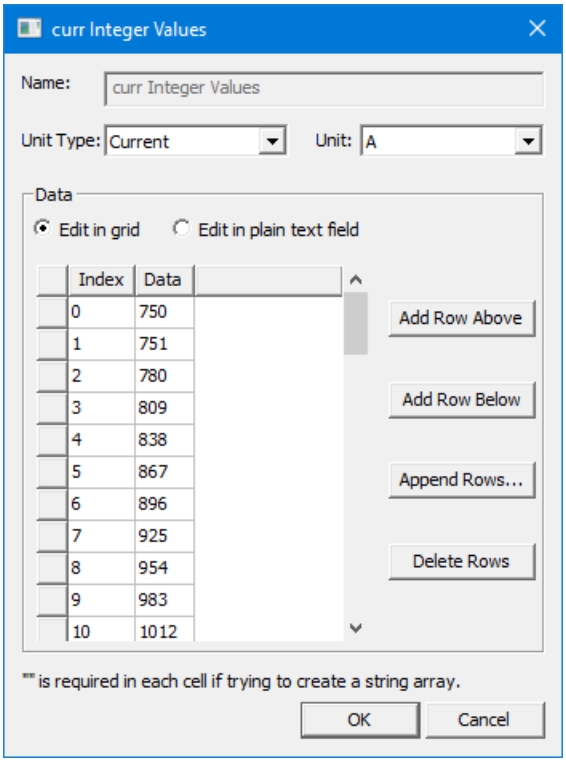
- **Override** – check this to override the current design value. Unchecking this causes a dialog to appear asking you to confirm the return to the design value.
- **Include** – whether to include the variable in an analysis.
- **Discrete** – **Discrete Variables** physically represent different configurations or states of the model. When check box in the Discrete column is checked, the button in the **Levels** column will be enabled and **Use Manufacturable Values** is disabled.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Levels show the variable values. For continuous variable, Levels is determined by the min/max columns. For Integer variables Levels can be a subset of the min/max range. The discrete values can be bounded by a min/man range and/or manufacturable values.

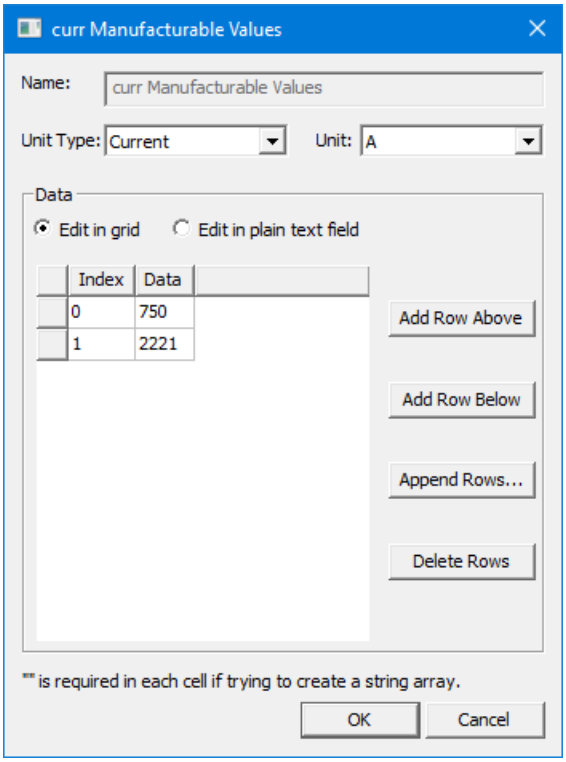
Click the enabled **Levels** button for the row to edit the Discrete values. An edit dialog box for the variable appears:



For continuous variables (with **Discrete** unchecked), the values physically vary in a continuous manner between a lower and an upper bound (min/max) defined by the user. With continuous variables, you can also check **Use Manufacturable Values**.

- When you enable the check box in the "**Use Manufacturable Values**" column the button in the "**Levels**" column is enabled. Manufacturable Values represent real-world manufacturing or production constraints. The min max values for the variable will be upper and lower constraints on the available manufacturable values bounded by the min/max range. If you check **Use Manufacturable values**, you edit the Levels to define the Manufacturable values. Click the Levels button to edit the values. An edit dialog for the variable appears.

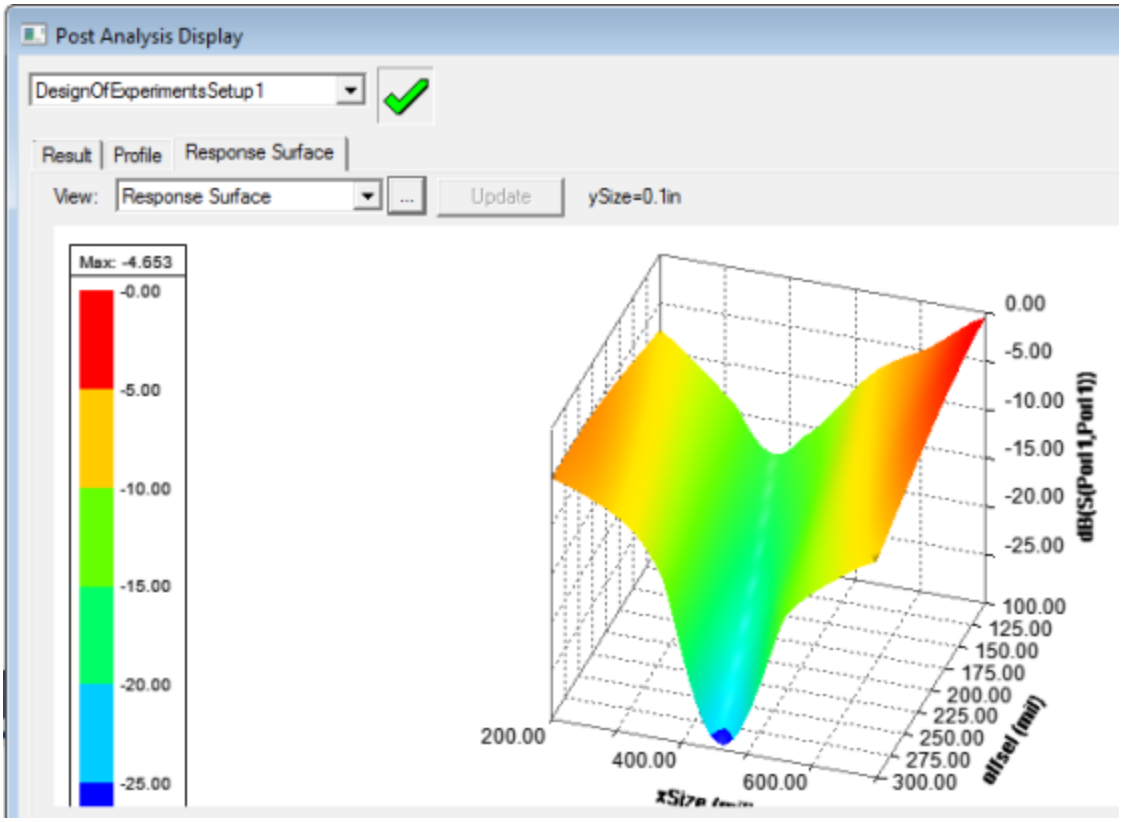
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Viewing Analysis Result for Design of Experiments

After the Analysis has competed, you can right-click the Design of Experiments setup in the **Project tree** and click **View Analysis Result** on the shortcut menu. This opens a dialog that includes a **Result** tab that lists the variations and variable values, a solution **Profile** tab with start, stop, time elapsed, and machines used, as well as a **Response Surface** tab. When you include more than one variable in the setup, response surface view is available. You can choose any two variables as the X, Y axis, and choose an output calculation as the Z axis, by clicking the ellipsis [...] button next to the view list box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



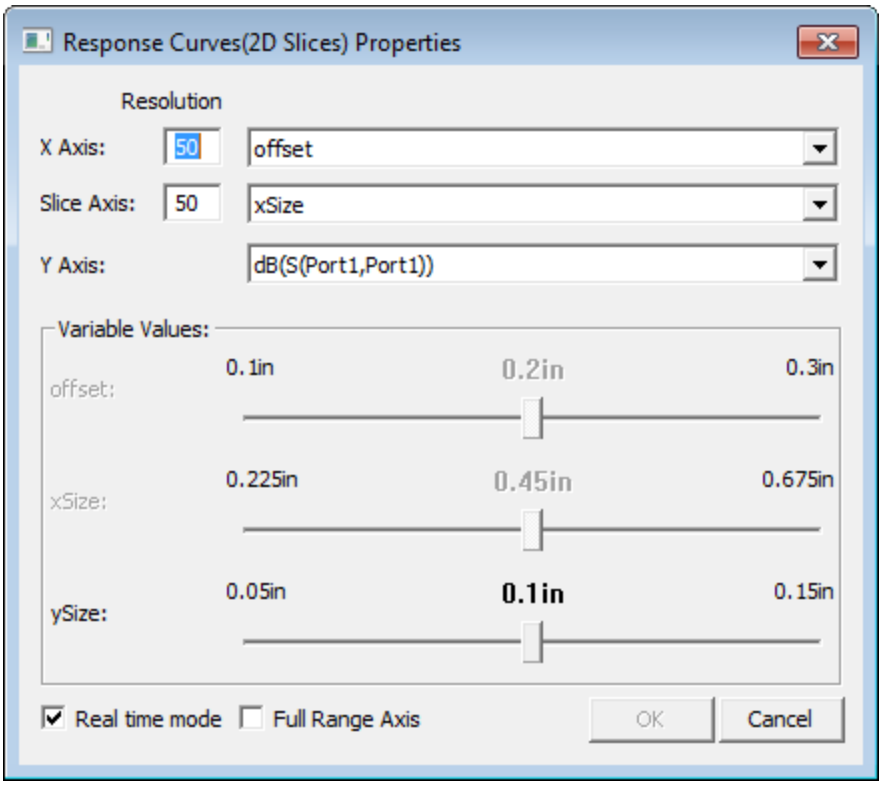
From the **Response Surface** tab, the “View” list box lets you select all available views of the selected response-surface-setup.

- Min Max Search
- Refinement points table
- Response points table
- Verification points table
- Goodness of Fit
- Response Curve
- Response Curve (2D Slices)
- Response Surface
- Local Sensitivity Charts
- Local Sensitivity Curves

The **Update** push button is disabled when the response surface is up to date. After the setup, if you modify a verification point or refinement points, it is enabled. Click this button to regenerate the response-surface with new settings. It may start new simulations if any of the design points in the DOE, refinement points, or verification points has not been solved.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clicking the ellipsis button [...] by the View for Response Surface opens a dialog box that lets you adjust the variables selected and the values applied.



Tuning a Response Curve

For the X Axis and Y Axis, you can specify a Resolution, and the variable to use. For the Z Axis you can select the Cost or calculation. For variables not selected for the X and Y axis, a slider is enabled that lets you adjust the value to see the effect on the response curve plot. You can enable or disable **Real time mode** by using the check box at the lower left.

Full Range Axis check box

When it is checked all Axes are set to their maximum ranges, and the ranges won't be changed while tuning unless you change the axis variable.

When it is unchecked, the Y(2D)/Z(3D) axis range is auto updated to fit the curve/surface.

Exporting Response Curve Data

You can Export the response curve data as a table in the following formats:

- Comma delimited data files, (*.csv)
- Tab delimited data files, (*.tab)
- Ansys Plot data files, (*.dat)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

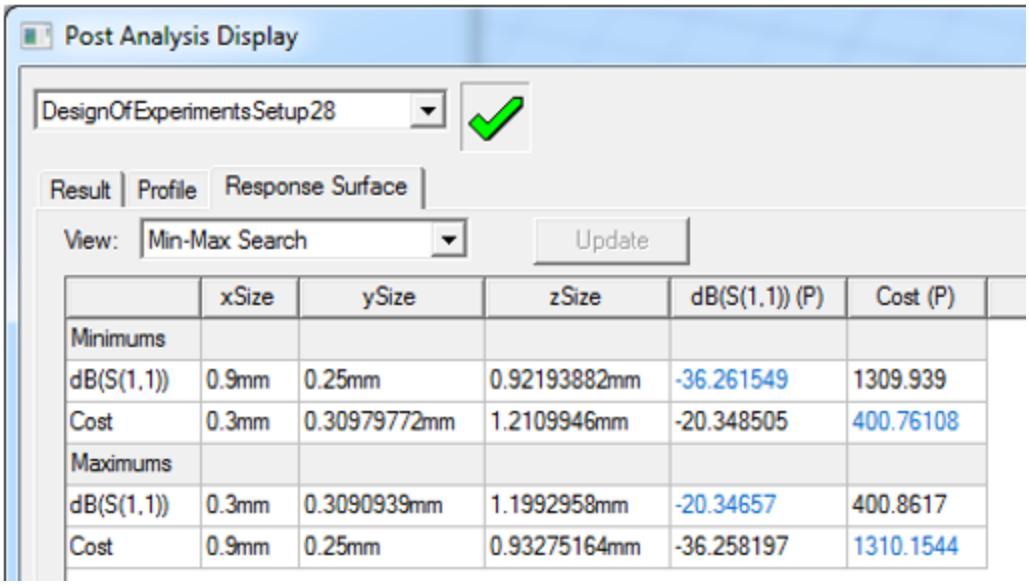
- Post Processor format data files, (*.txt)
- Ansys Report Data *.rdat files

Exported files can be imported into a report.

Min-Max Search View for Design of Experiments Result

From the **Response Surface** tab of the Design of Experiments **Post Analysis Display** dialog box, the **View** list box lets you select all available views of the selected response-surface-setup.

The **Min-Max Search** view examines the entire output parameter space from a response surface to approximate the minimum and maximum values of each output parameter. When you select a Min/Max row **Apply** is enabled, and you can then apply the selected variation variable values to the variables' nominal values.



	xSize	ySize	zSize	dB(S(1,1)) (P)	Cost (P)
Minimums					
dB(S(1,1))	0.9mm	0.25mm	0.92193882mm	-36.261549	1309.939
Cost	0.3mm	0.30979772mm	1.2109946mm	-20.348505	400.76108
Maximums					
dB(S(1,1))	0.3mm	0.3090939mm	1.1992958mm	-20.34657	400.8617
Cost	0.9mm	0.25mm	0.93275164mm	-36.258197	1310.1544

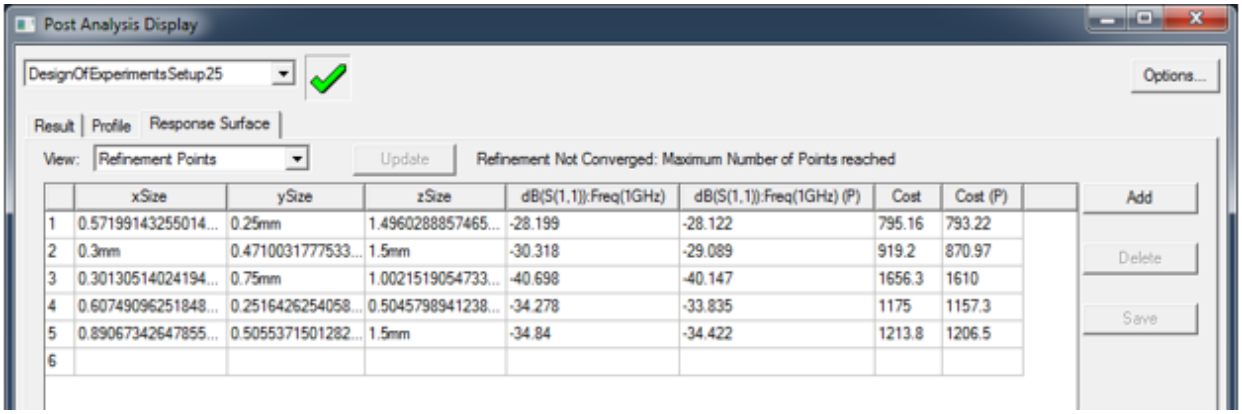
You can Export the Table in the following formats:

- Comma delimited data files, (*.csv)
- Tab delimited data files, (*.tab)
- Ansoft Plot data files, (*.dat)
- Post Processor format data files (*.txt)

Refinement Points Table

From the Response Surface tab, the "View" list box lets you select all available views of the selected response-surface-setup. All refinement points are shown in the Refinement Points table:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



	xSize	ySize	zSize	dB(S(1,1)):Freq(1GHz)	dB(S(1,1)):Freq(1GHz) (P)	Cost	Cost (P)
1	0.57199143255014...	0.25mm	1.4960288857465...	-28.199	-28.122	795.16	793.22
2	0.3mm	0.4710031777533...	1.5mm	-30.318	-29.089	919.2	870.97
3	0.30130514024194...	0.75mm	1.0021519054733...	-40.698	-40.147	1656.3	1610
4	0.60749096251848...	0.2516426254058...	0.5045798941238...	-34.278	-33.835	1175	1157.3
5	0.89067342647855...	0.5055371501282...	1.5mm	-34.84	-34.422	1213.8	1206.5
6							

Refinement points are points added to your model to enrich and improve your response surface. They can either be generated automatically with the response surface update or added manually, as described in **Performing a Manual Refinement**. As with design points, DesignXplorer must perform a design point update (a "real solve") in order to obtain the output parameters for the refinement points.

Upon update, the refinement points are used to build the response surface and are taken into account for the generation of verification points. Along with DOE points, refinement points are also used as "learning points" for Goodness of Fit calculations.

Performing a Manual Refinement

Manual refinement is a way to force the response surface to take into account points of your choice, in addition to the points already in the Design of Experiments. You can insert a refinement point in the Refinement Points table, and you do not need to solve for an initial response surface (without the refinement point) before updating it with your manual refinement. Manual refinement is available for all response surface types except for Sparse Grid.

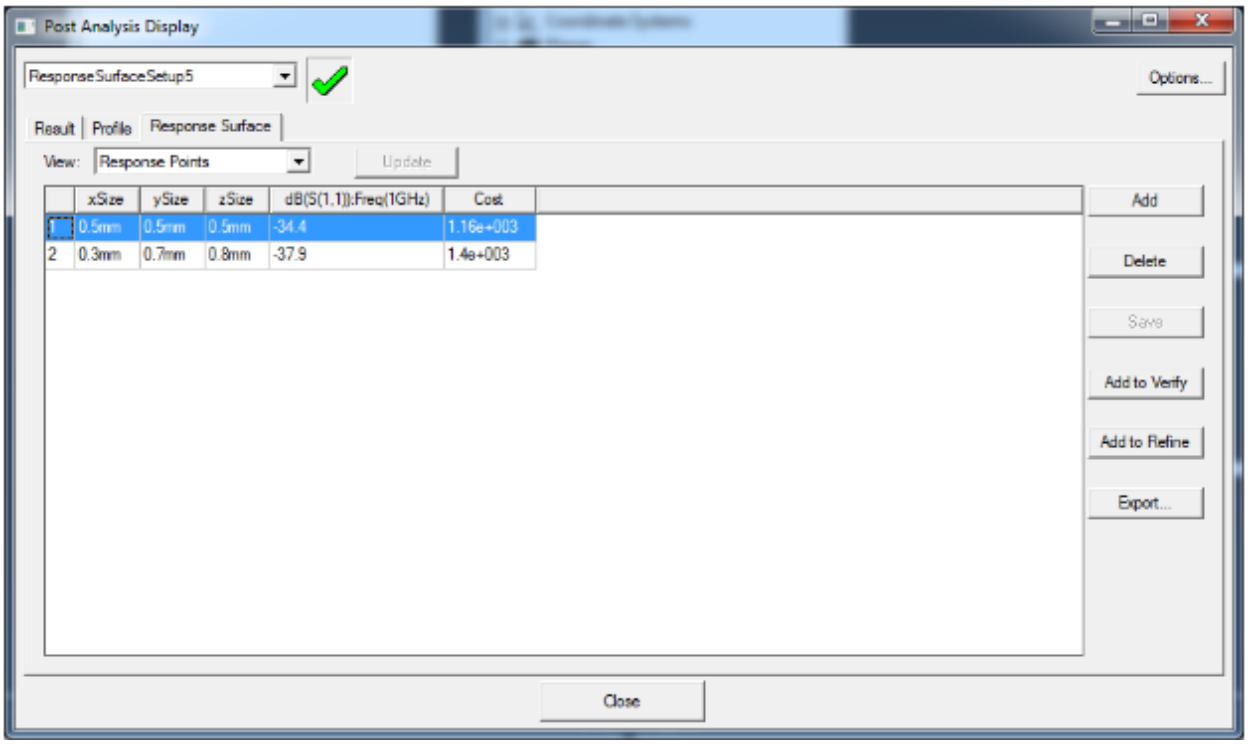
You can add, delete, or modify refinement points by clicking the **Add** or **Delete** buttons, or by modifying point values directly in the grid.

Manual Refinement Point can be inserted from the Response point table and Verification Points Table.

Response Points Table

From the Response Surface tab, the **View** list box lets you select all available views of the selected response-surface-setup. All Response points are shown in the Response Points table:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A response point is defined by a snapshot of variable values where output calculation values were calculated in Ansys DesignXplorer from a response surface. As such, the output calculation(or cost) values are approximate and calculated from response surfaces.

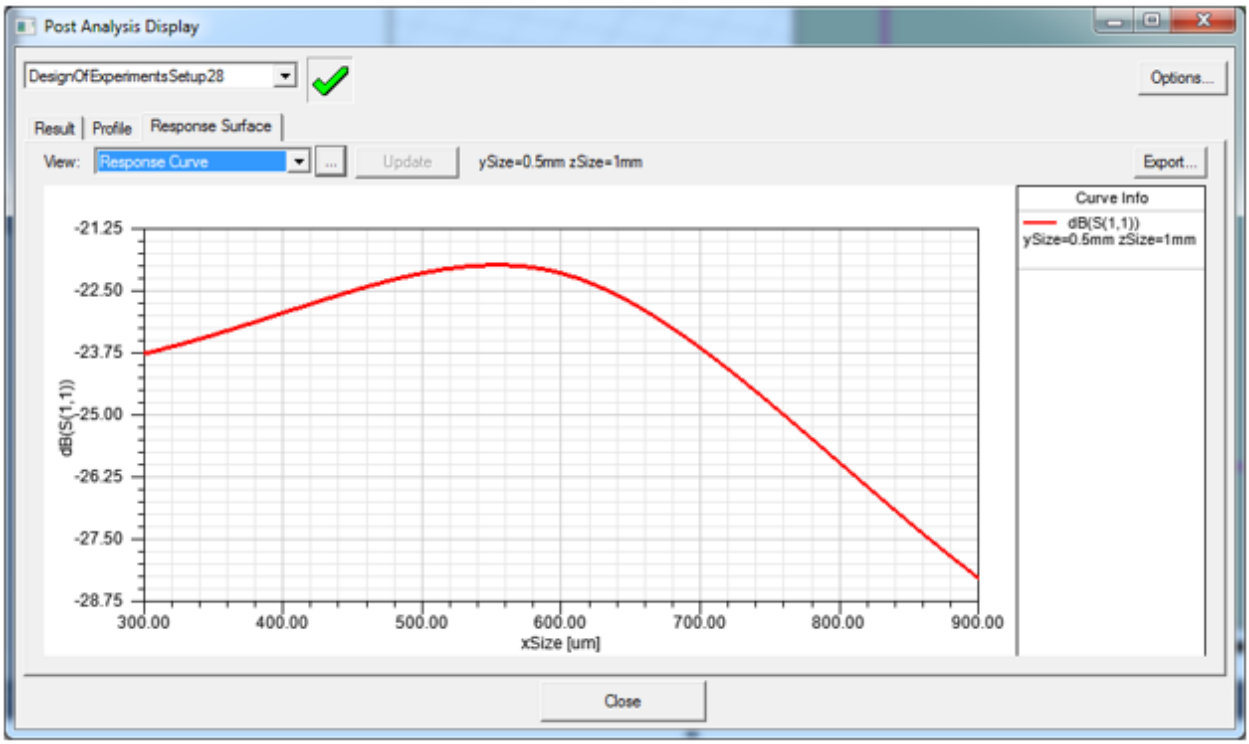
You can add, delete, save, or export response points by using the command buttons in the dialog or you can modify response points manually by modifying point values directly in the grid.

Click **Add to Verify** or **Add to Refine** buttons to insert the selected response point to the verification table or refinement table.

Response Curve

When Continuous Variable is chosen as the X-axis, and Continuous XY plot will be shown.

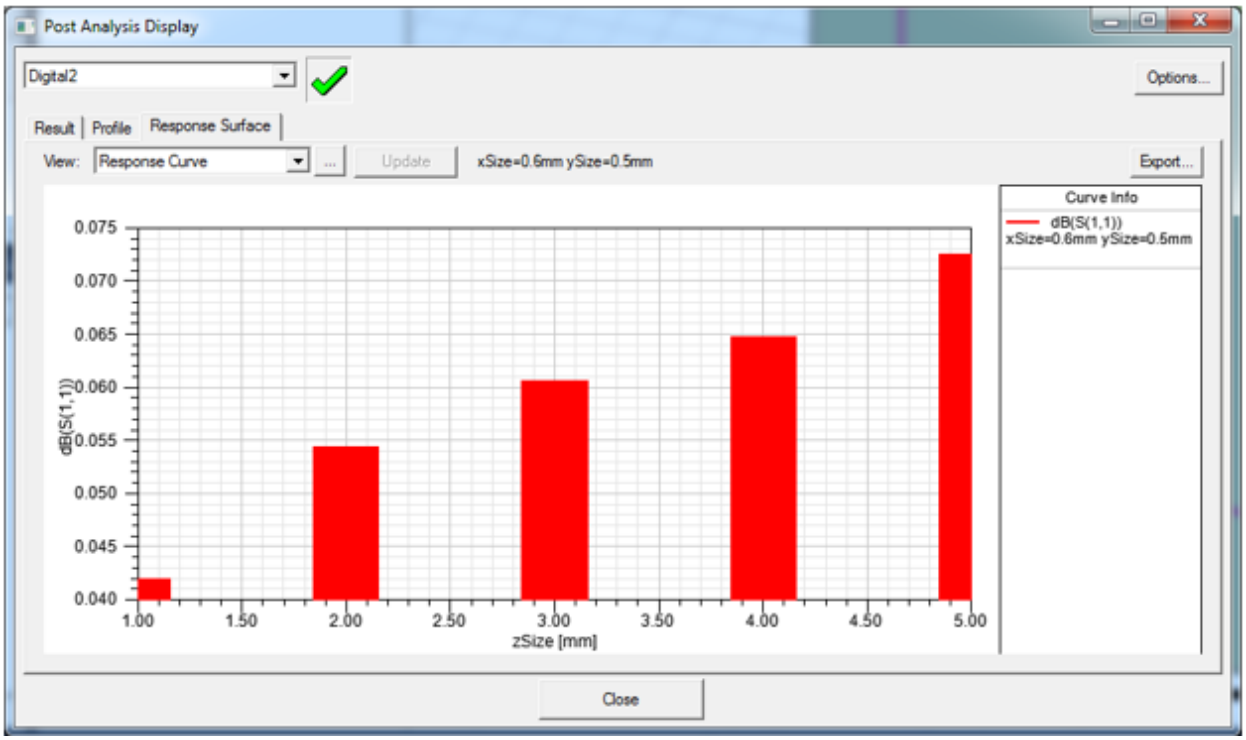
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Discrete Variable

If you choose Discrete Variable as the X-axis, a bar chart plot is shown:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Tuning a Response Curve

For the X Axis and Y Axis, you can specify a Resolution, and the variable to use. For the Z Axis you can select the Cost or calculation. For variables not selected for the X and Y axis, a slider is enabled that lets you adjust the value to see the effect on the response curve plot. You can enable or disable **Real time mode** by using the check box at the lower left.

Full Range Axis check box

When it is checked all Axes are set to their maximum ranges, and the ranges won't be changed while tuning unless you change the axis variable.

When it is unchecked, the Y(2D)/Z(3D) axis range is auto updated to fit the curve/surface.

Accumulate Response Curve

It always be unchecked when the tuning dialog started, and won't restore it last check/uncheck state. When it is checked, it will retaining the existing curves, and add new curve to the plot. Uncheck the check box won't clear the accumulated curves, just stop to accumulate new curve. When the Axis variable is changed, all accumulated curves will be cleared.

Exporting Response Curve Data

You can Export the response curve data as a table in the following formats:

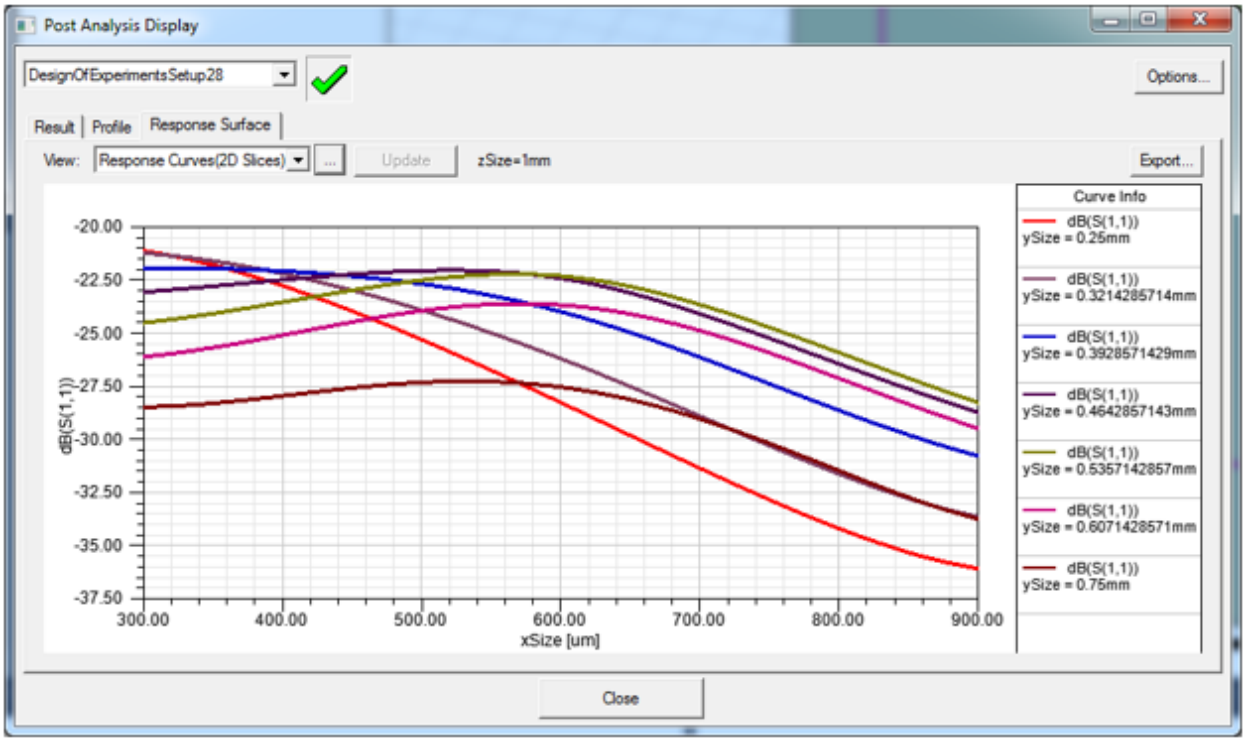
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Comma delimited data files, (*.csv)
- Tab delimited data files, (*.tab)
- Ansys Plot data files, (*.dat)
- Post Processor format data files, (*.txt)
- Ansys Report Data *.rdat files

Exported files can be imported into a report.

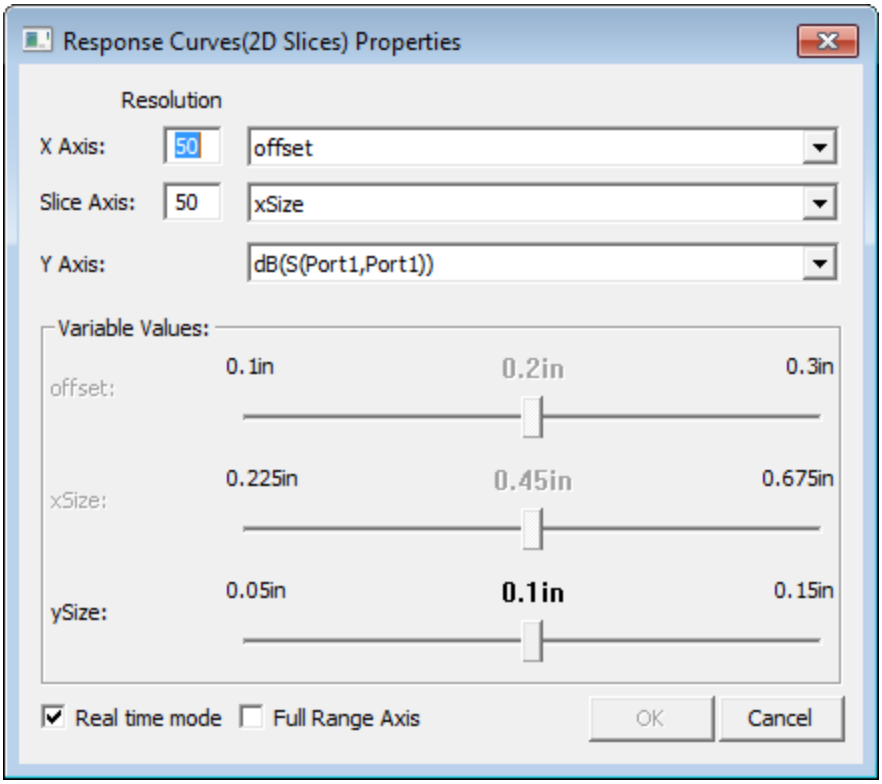
Response Curve (2D Slices)

If you have included more than one variable in the setup, the Response Curves (2D) view is available:



Clicking the ellipsis button [...] by the View for Response Surface opens a dialog that lets you adjust the variables selected and the values applied.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Tuning a Response Curve

For the X Axis and Y Axis, you can specify a Resolution, and the variable to use. For the Z Axis you can select the Cost or calculation. For variables not selected for the X and Y axis, a slider is enabled that lets you adjust the value to see the effect on the response curve plot. You can enable or disable **Real time mode** by using the check box at the lower left.

Full Range Axis check box

When it is checked all Axes are set to their maximum ranges, and the ranges won't be changed while tuning unless you change the axis variable.

When it is unchecked, the Y(2D)/Z(3D) axis range is auto updated to fit the curve/surface.

Exporting Response Curve Data

You can Export the response curve data as a table in the following formats:

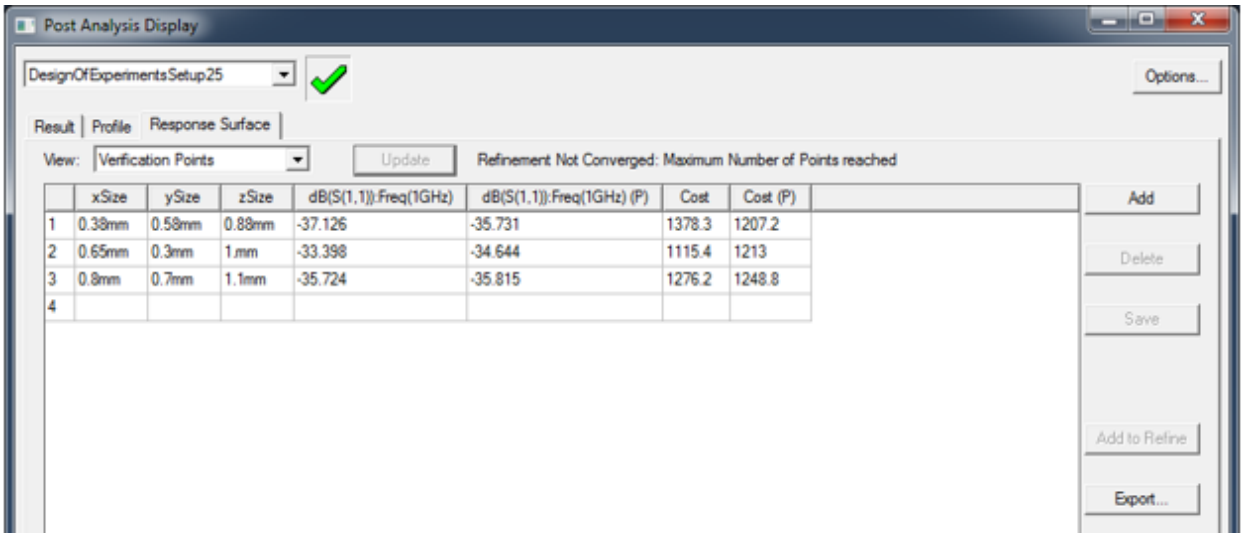
- Comma delimited data files, (*.csv)
- Tab delimited data files, (*.tab)
- Ansys Plot data files, (*.dat)
- Post Processor format data files, (*.txt)
- Ansys Report Data *.rdat files

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Exported files can be imported into a report.

Verification Points Table

From the Response Surface tab, the **View** list box lets you select all available views of the selected response-surface-setup. All Verification points are shown in the Verification Points table:



Verification points enable you to verify that the response surface accurately approximates the output parameter values; they compare the predicted and observed values of the output parameters.

You can add, delete and modify Verification points manually:

- Same as add/delete/modify refinement points.
- Insert from Response Points Table.

Click **Add to Refine** button to insert the selected response point to the refinement table.

A design point update (that is, a "real solve") calculates each verification point. These verification point results are then compared with the response surface predictions and the difference is calculated.

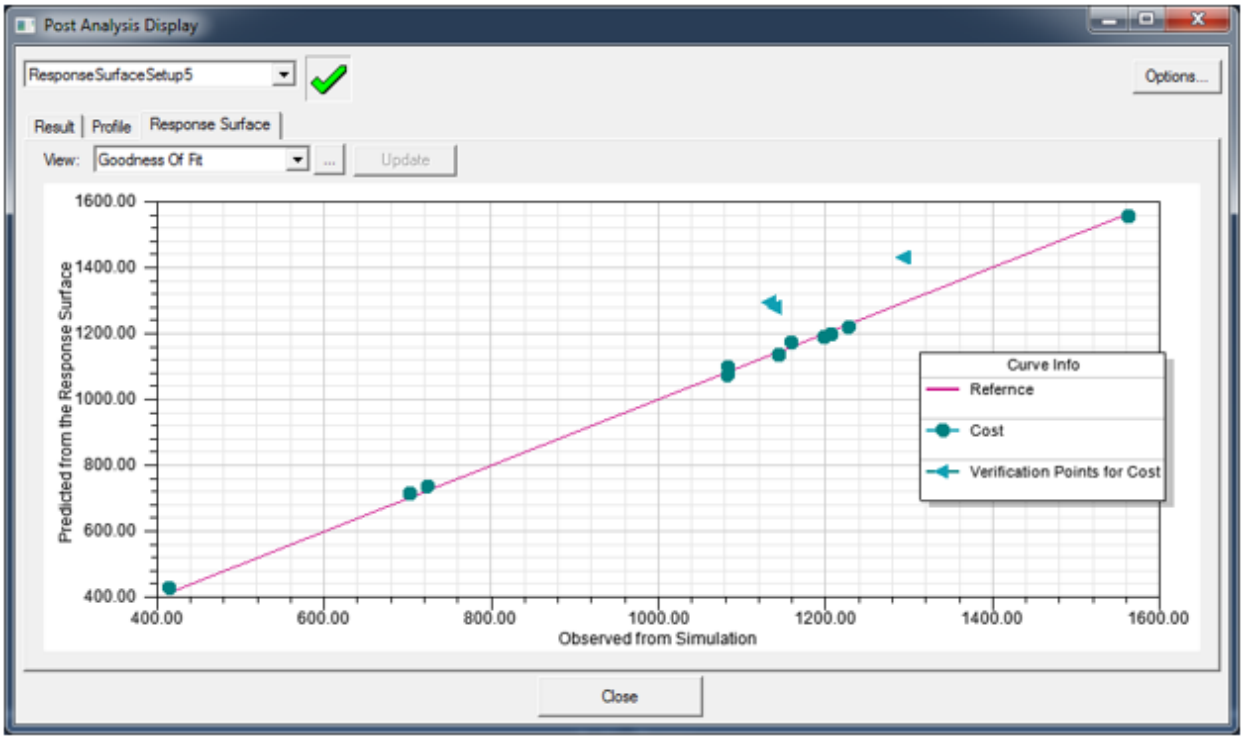
Verification points are useful in validating any type of response surface. In particular, however, you should always use verification points to validate the accuracy of interpolated response surfaces, such as Kriging or Sparse Grid.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Goodness of Fit (Predicted vs Observed Chart)

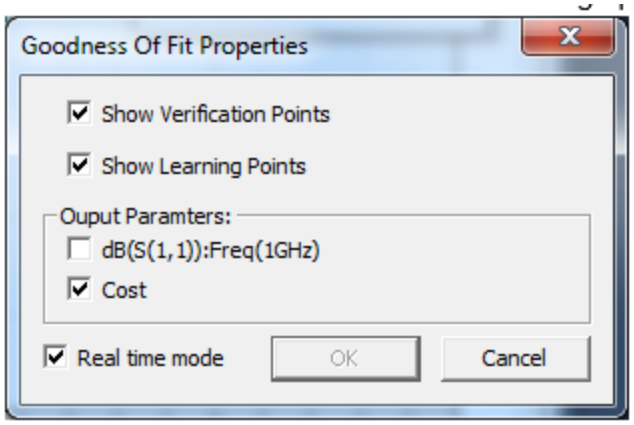
From the Response Surface tab, the **View** list box lets you select all available views of the selected response-surface-setup.Response surfaces are built from design points in the Design of Experiments (DOE) and refinement points (collectively, called "learning points"). The Goodness of Fit calculations compare the response surface outputs with the DOE results used to create them.

The closer the points are to the diagonal line, the better the response surface fits the points.



You can view Goodness of Fit information for any of the output parameters in a response surface. To do so click the ellipsis button [...] button to bring up this dialog box:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

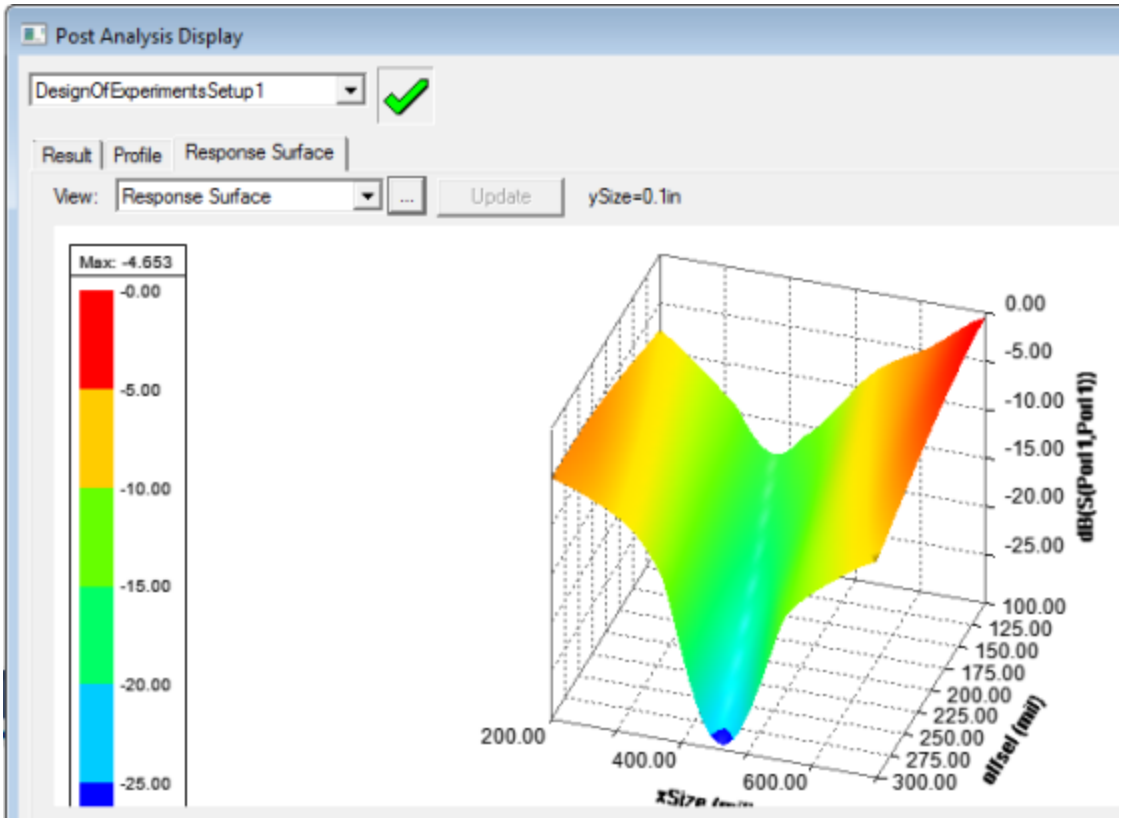


Response Surface Results

When more than one variable is included in the setup, the Response Surface view is available. You can choose any two variables as the X, Y axis, and choose an output calculation as the Z axis, by clicking the "..." button next to the view list box. From the **Response Surface** tab of the Design of Experiments **Post Analysis Display** dialog box, the "View" list box lets you select all available views of the selected response-surface-setup.

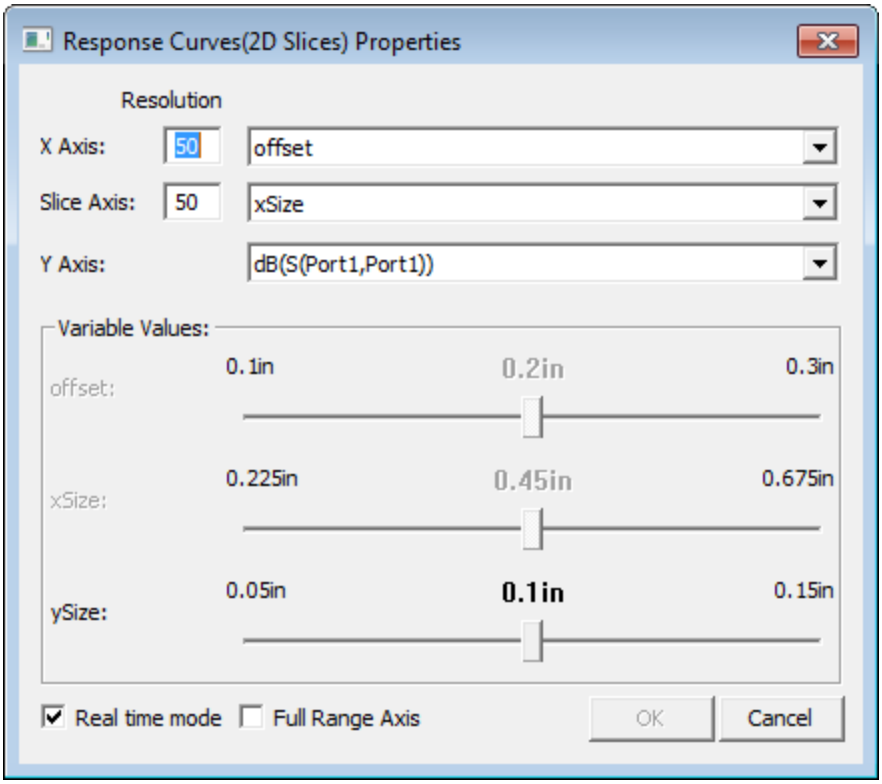
Response surfaces are functions of varying natures in which the output parameters are described in terms of the input parameters. Built from the Design of Experiments (DOE), they quickly provide the approximated values of the output parameters throughout the design space without having to perform a complete solution. The accuracy of a response surface depends on several factors: the complexity of the variations of the solution, the number of points in the original DOE, and the response surface type. Once a response surface has been generated, you can create and manage response points and charts. These postprocessing tools help you to understand how each output parameter is driven by input parameters and how you can modify your design to improve its performance.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Clicking the ellipsis button [...] by the View for Response Surface opens a dialog that lets you adjust the variables selected and the values applied.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Tuning a Response Surface

For the X Axis and Y Axis, you can specify a Resolution, and the variable to use. For the Z Axis you can select the Cost or calculation. For variables not selected for the X and Y axis, a slider is enabled that lets you adjust the value to see the effect on the response surface plot. You can enable or disable **Real time mode** by using the check box at the lower left.

Full Range Axis check box

When it is checked, all Axes are set to their maximum ranges, and the ranges won't be changed while tuning unless you change the axis variable.

When it is unchecked, the Y(2D)/Z(3D) axis range is auto updated to fit the curve/surface.

Exporting Response Surface Data

You can Export the Response surface data as a table in the following formats:

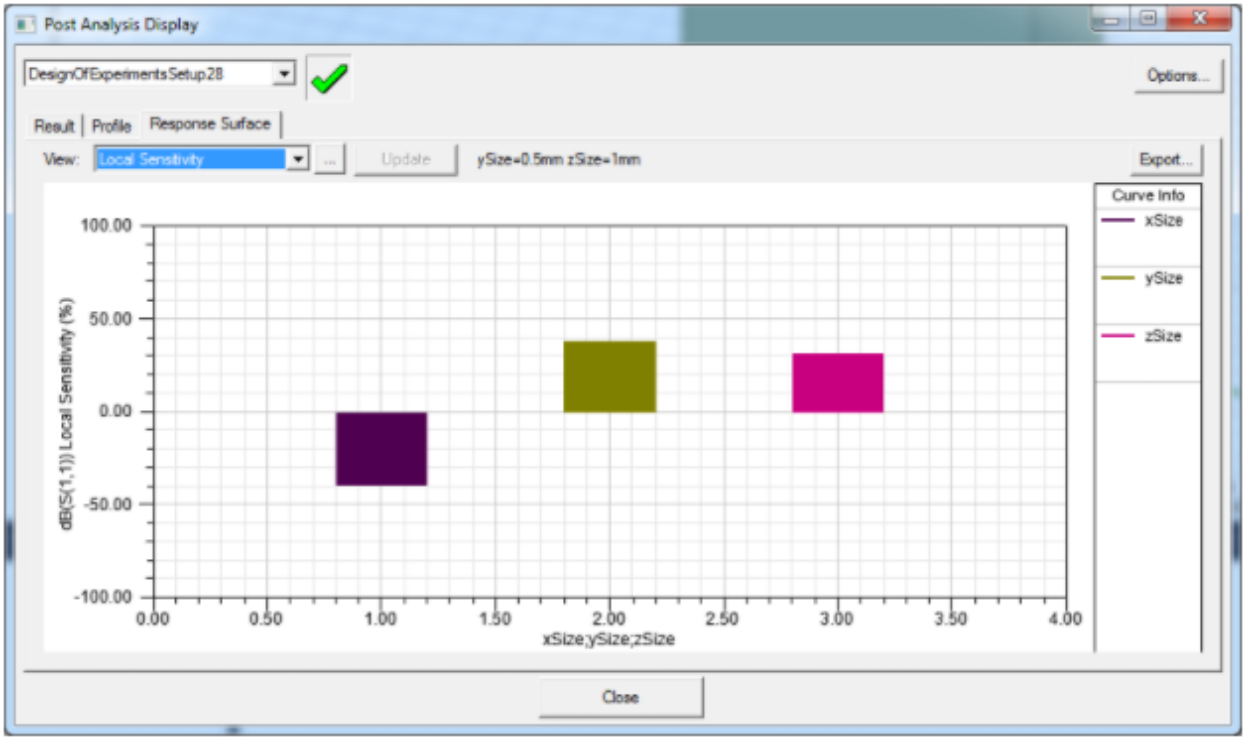
- Comma delimited data files, (*.csv)
- Tab delimited data files, (*.tab)
- Ansoft Plot data files, (*.dat)
- Post Processor format data files (*.txt)

Exported files can be imported into a report.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

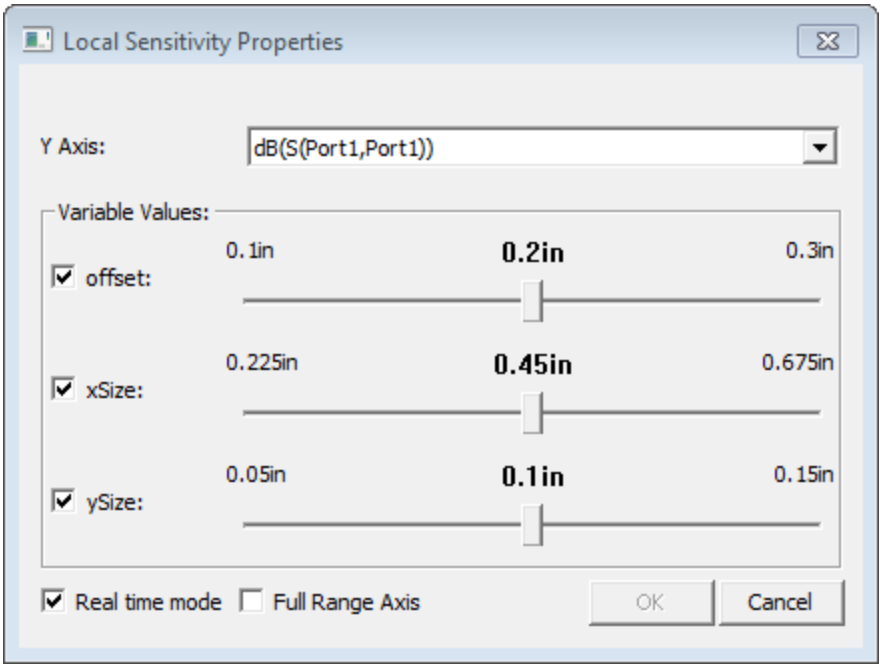
Local Sensitivity Charts

From the **Response Surface** tab, the **View** list box lets you select all available views of the selected response-surface-setup. Local Sensitivity Charts allow you to graphically view the impact that changing each input parameter has on the output parameters. The Local Sensitivity chart can be a powerful exploration tool. For each output, it allows you to see the weight of the different input. This chart calculates the change of the output based on the change of each input independently, at the current value of each input parameter in the project.



Clicking the ellipsis button [...] opens the **Local Sensitivity Properties** dialog that lets you adjust the variables selected and the values applied.

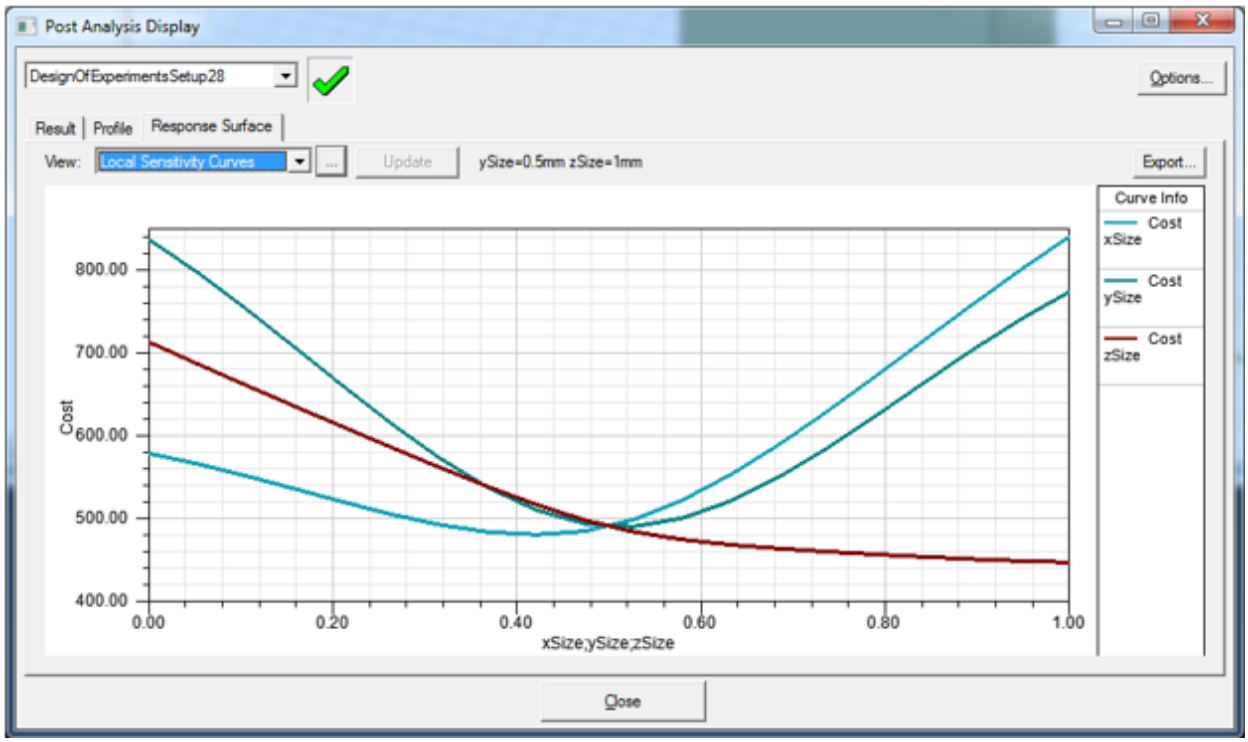
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Local Sensitivity Curves

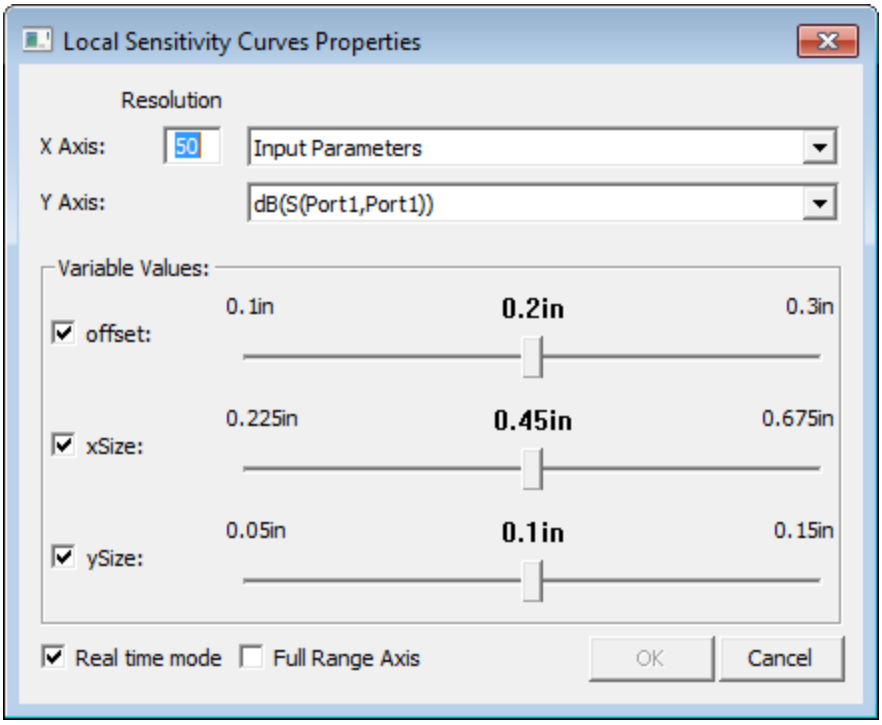
From the **Response Surface** tab, the **View** list box lets you select all available views of the selected response-surface-setup. Local Sensitivity Curves allow you to graphically view the impact that changing each input parameter has on the output parameters. The Local Sensitivity Curves chart helps you to focus your analysis by allowing you to view independent parameter variations within the standard Local Sensitivity chart. This multi-curve chart provides a means of viewing the impact of each input parameter on specific outputs, given the current values of the other parameters. The Local Sensitivities Curves chart shows individual local sensitivities, with a separate curve to represent the impact of each input on one or two outputs.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Clicking the ellipsis button [...] opens the **Local Sensitivity Properties** dialog box that lets you adjust the variables selected and the values applied.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



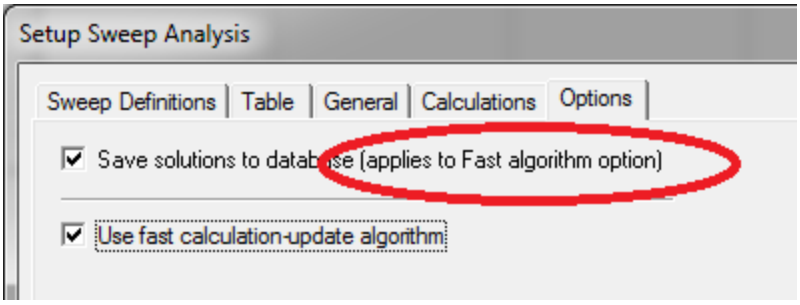
Using the Fast Calculation-Update Algorithm

A fast calculation-update algorithm is available to speed up Optimetrics and report updates during Optimetrics analyses. The fast calculation-update algorithm will generate the same Optimetrics results, only faster, and is available for all Optimetrics analyses *except* a Tuning analysis. By default, the fast calculation-update option is automatically enabled whenever it is applicable, but you can configure it manually using the **Options** tab of the applicable Optimetrics *Setup* dialog box. To enable the fast calculation-update algorithm:

1. From the menu bar, click your product and then **Circuit** > **Optimetrics Analysis** > **Add** <*Optimetrics Type*>.

   A typical dialog box to *Setup Sweep Analysis* is shown below with the **Options** tab selected.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

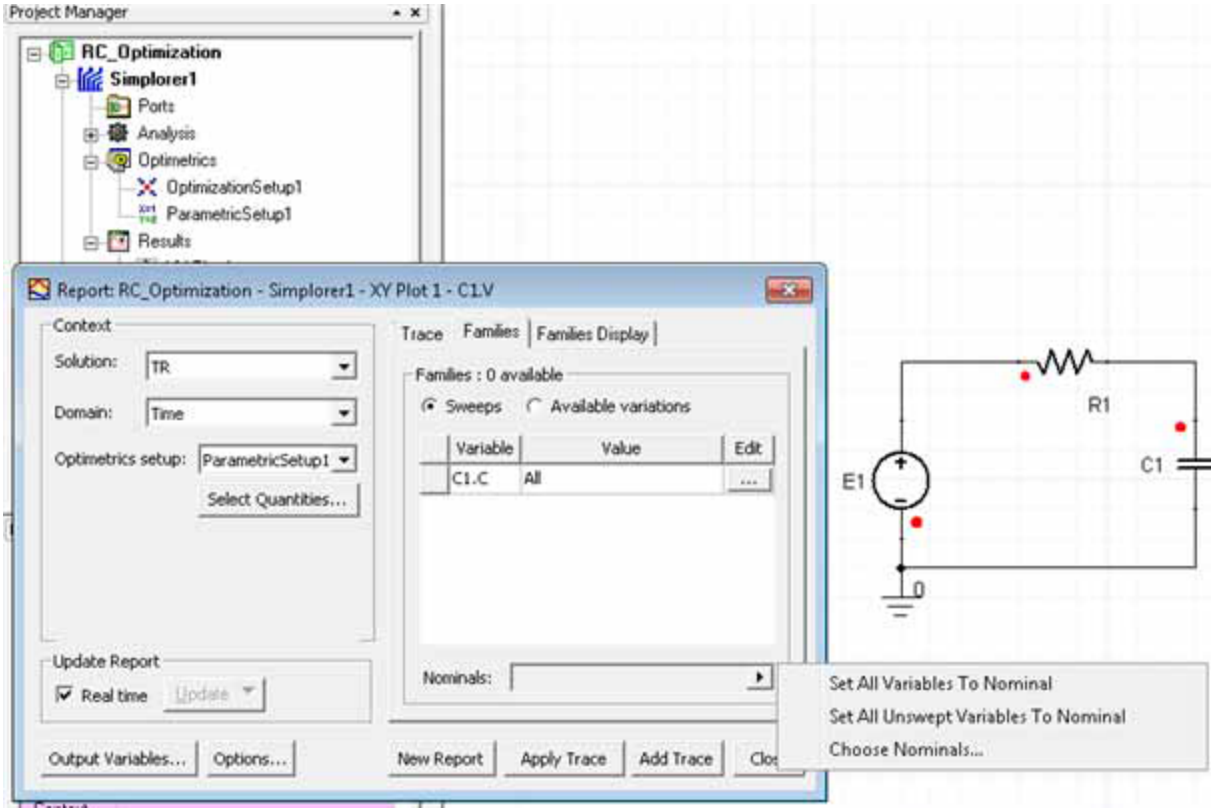


The *Options* tab is the same for all optimization type setups.

2. Select **Use fast calculation-update algorithm** to enable use of the algorithm. (See also, *Fast Calculation-Update Algorithm Limitations*.)

   When the fast calculation-update algorithm is enabled:

   - You can enable some reports to be updated automatically during the optimetrics analysis if you **Set All Variables To Nominal** in the Report/Trace setup dialog box:

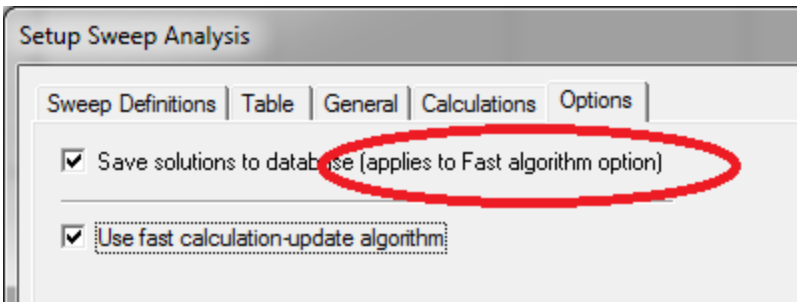


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You can see each trace (overwriting the previous), by setting the **Optimetrics setup** in the Report dialog box to **None** in addition to having all variables set to Nominal. At the end of the analysis, the user will see the last calculated value.

3. If you have enabled **Use fast calculation-update algorithm** and want to save the solution data for every solved design variation in the Optimetrics analysis, select **Save solutions to database** as shown below. Selecting this option has no effect without enabling **Use fast calculation-update algorithm**.

> **Warning:**
>
> Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.



- When the **Save solutions to database** option is checked, a plot with traces based on the Optimetrics Setup just run can be updated through a menu command. Right-click the desired report under *Results* in the Project Manager and select **Update Report**) to show results as appropriate (including a Family of results, if chosen).
- Without the **Save solutions to database** option checked, you can examine analysis data in the *Post Analysis Display* dialog box, which is available by right-clicking the Optimetrics Setup and choosing **View Analysis Result**.

Fast Calculation-Update Algorithm Limitations

The fast calculation-update algorithm **cannot** be used if **any** optimetrics calculation uses

- **Project/Design variables** – If the project/design variable is ***not*** swept in the Optimetrics analysis, and you would like to use it in the expression: You can create an output variable for the Project/Design variable. Assign the value of the Project/Design variable to the output variable and use the output variable in the expression instead of a constant numerical value.
- **More than one range function** – For example, when range function is not the outermost function or when range function takes arguments.
- **More than one calculation range**, or the calculation range is not for primary sweep.

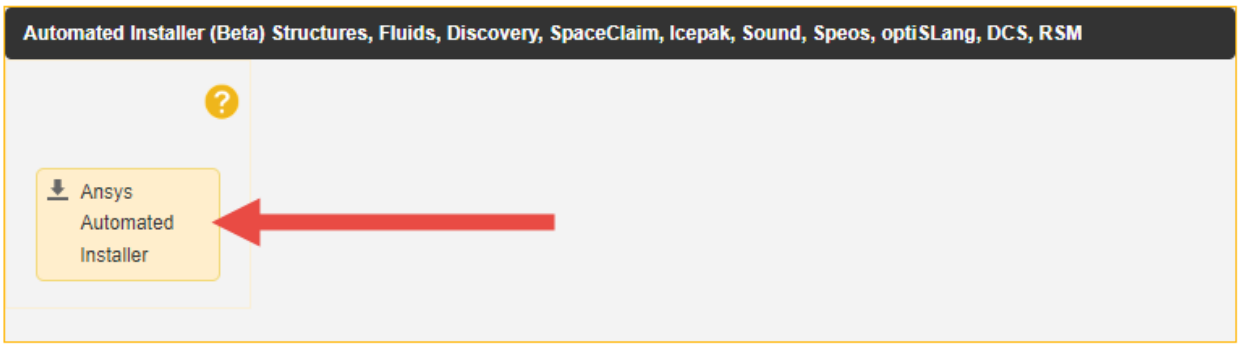
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Similarly, when the fast calculation-update algorithm is enabled, a trace cannot be updated during an analysis if the trace expression uses

- **Project/Design variables**. (However, you can use the same workaround described above.)
- **Any range function.**

optiSLang Integration with Electronics Desktop

optiSLang is a tool for graphical programming, process integration, and automation that can be integrated with Ansys Electronics Desktop (AEDT) for Optimetrics analysis. It is available as part of the Ansys Automated Installer package on the Ansys Customer Portal.



Comprehensive online help for optiSLang is available from the optiSLang Help menu, or at:

https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v252/en/opti_ug/opti_ug_intro.html

This section covers only its integration with Ansys Electronics Desktop.

optiSLang in AEDT allows AEDT and optiSLang to stay in continuous interaction by exchanging API commands between each other. optiSLang algorithms can be executed without exiting AEDT because optiSLang setups follow Optimetrics norms and have an explicit association with a specific analysis setup. AEDT stays continuously open while optiSLang algorithms demand set after set of designs.

Each optiSLang setup resides in one specific design of an AEDT project and is linked to one specific analysis setup. When an optiSLang setup is executed within AEDT, optiSLang runs in the background in batch mode and executes the algorithm that the optiSLang setup represents. optiSLang administers an optiSLang project file (*.opf) and a normal optiSLang project directory (*.opd), stored relative to the AEDT project.

The **standalone optiSLang GUI** allows for further editing of the optiSLang project structure, with each analysis setup represented as a single Ansys EDT node:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

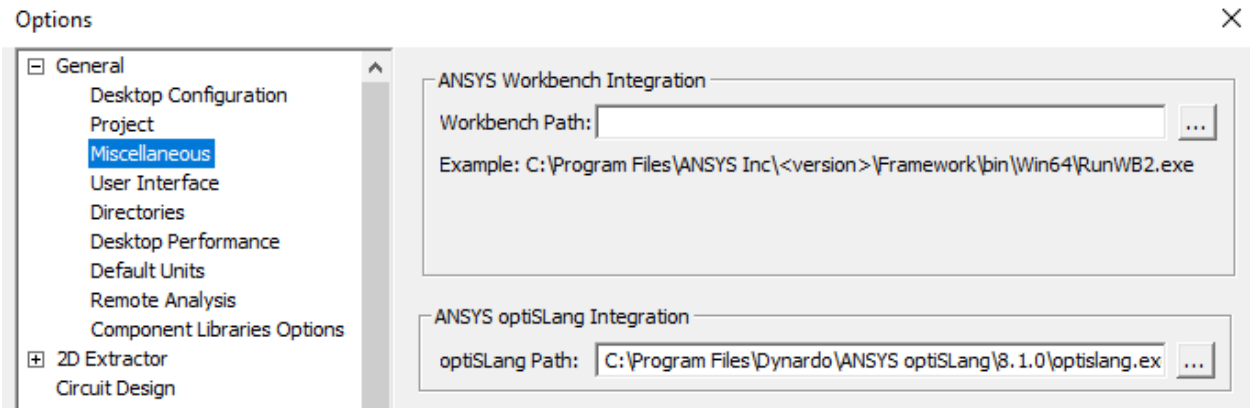
If the original Ansys EDT node and its copies are all kept in the optiSLang Setup run mode, they will remain linked to the specific optiSLang setup existing in the referenced AEDT project. The optiSLang project can contain and execute multiple algorithm systems, which means that the linked optiSLang setup in the AEDT project represents a connector, not an algorithm, in that case.

The topics in this section cover:

- optiSLang User Workflow
- Creating an optiSLang Setup in AEDT
- Solving an optiSLang Setup in AEDT
- optiSLang Menu Options
- Viewing optiSLang Postprocessing Results

Prerequisites

Before working with optiSLang in AEDT, specify the path to the optiSLang installation using **Tools** > **General Options** > **General** > **Miscellaneous**.



Additionally, ensure that the following are true:

- The project is solved
- Parameters exist (See: Parametrization for optiSLang Integration)
- Results reports exist (See: Results and Reports for optiSLang Integration)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

optiSLang User Workflow

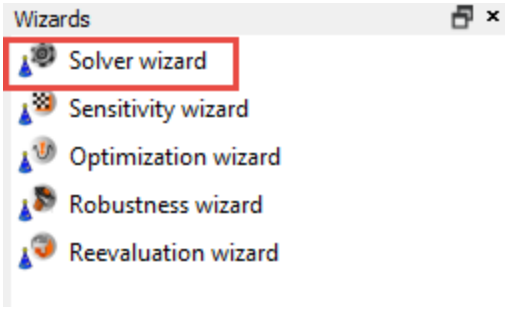
This topic explores the general workflow for creating an optiSLang project from Ansys Electronics Desktop (AEDT).

1. Prepare the AEDT project by ensuring:
   - Parameters exist.

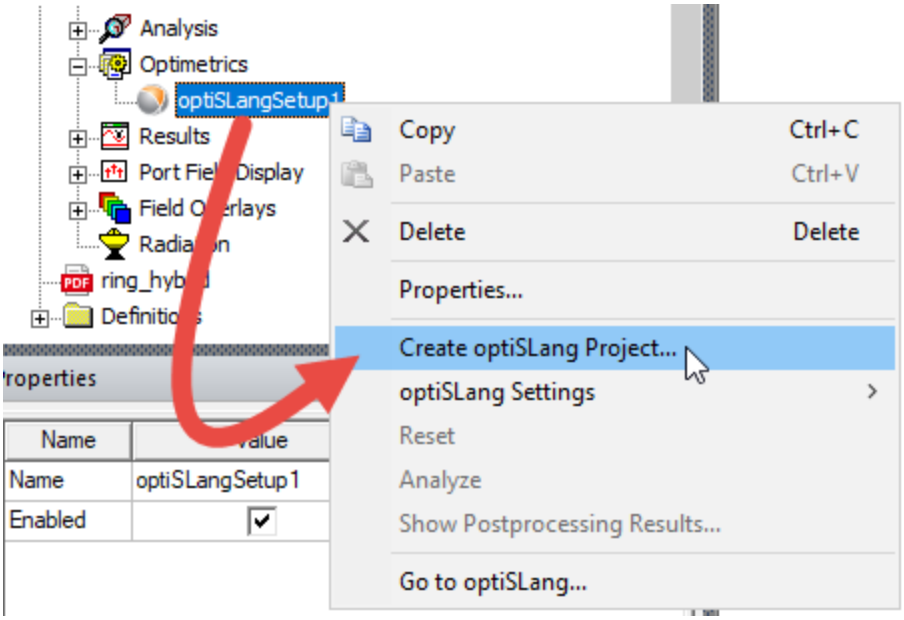
     > **Note:**
     >
     > optiSLang scans parameters at both the project and design level. Hidden parameters are also scanned. See: Parametrization for optiSLang Integration.

   - Project is solved.
   - Results reports exist and are prepared for the specific use case
     - For a standard use case, set all variables to nominal.
     - For a use case with a user-defined sweep, display all variations.

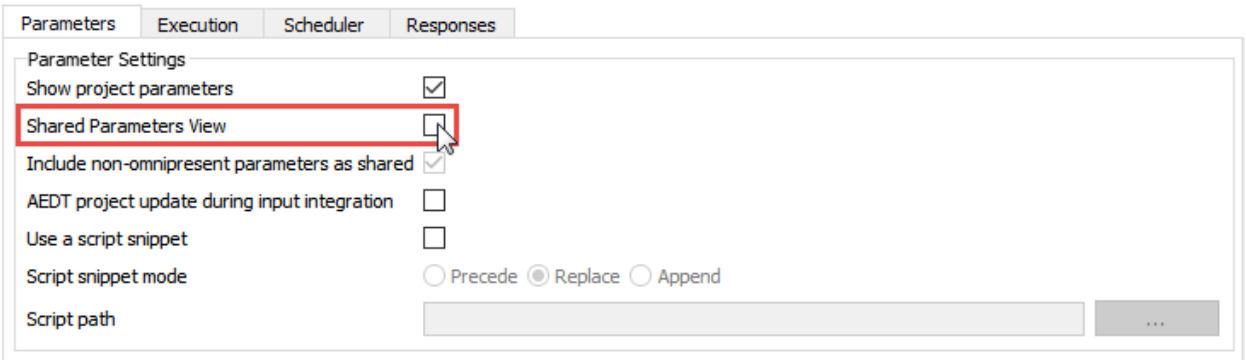
     See: Results and Reports for optiSLang Integration.
2. Invoke the Solver Wizard in optiSLang or create an optiSLang project directly from AEDT.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Create a one-node setup or a two-node setup.
    - A one-node setup registers both parameters and responses. This is the type of setup created by the wizard.
    - A two-node setup registers parameters in the first node and responses in the second. This is a more complicated setup that requires cloning the integration node, setting function switches, and setting absolute or relative paths. Consult the optiSLang help for more information.
4. View results in either tabular or tree view.
5. Enable **Shared Parameters View**. By default, parameters from different models are treated separately and a prefix based on the model name ensure uniqueness of names. In shared mode, sub-models are kept in sync and prefixes are unnecessary.



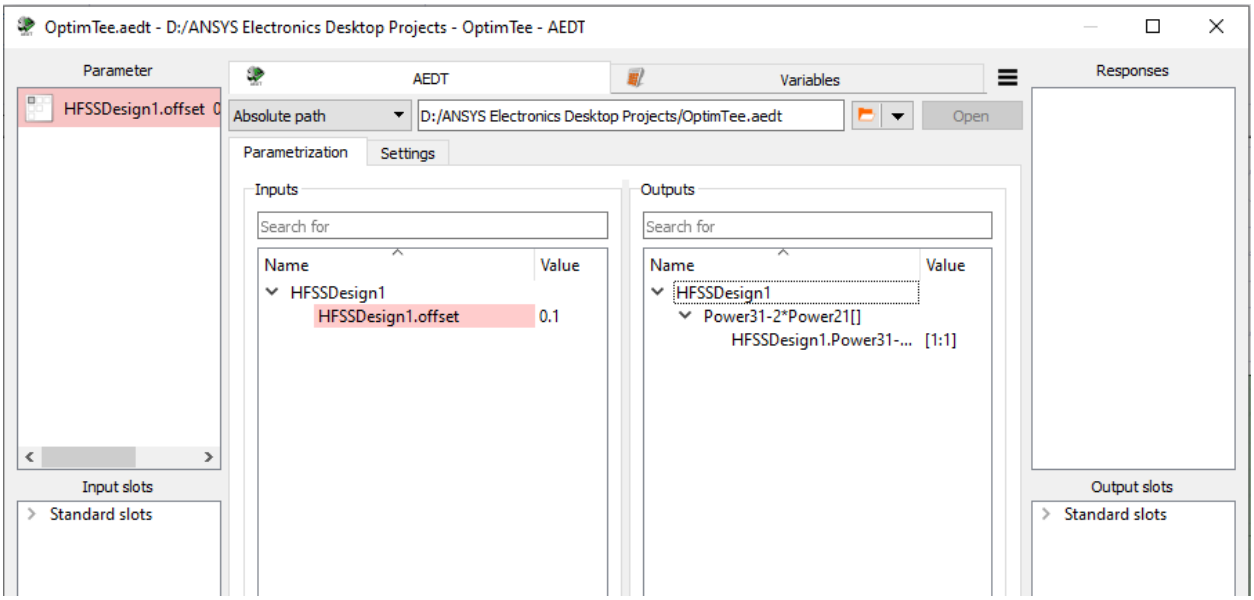
6. Reload the AEDT project in optiSLang to see the effects of changed settings.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parametrization for optiSLang Integration

Certain Ansys Electronic Desktop data must exist before optiSLang can use it.

Variables act as **Inputs** in optiSLang, while simulation results (reports) act as **Outputs**.



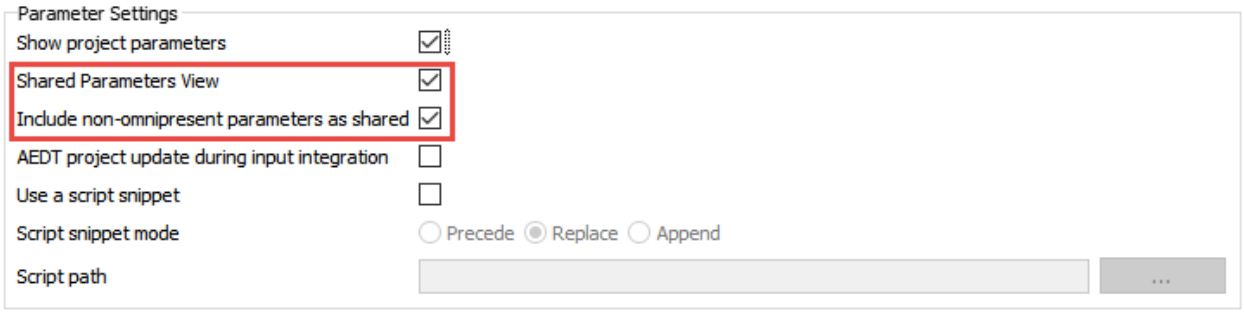
Inputs

When integrated with Ansys Electronics Desktop, optiSLang automatically detects project variables and design variables for every design in the project. Note that:

- Dependent parameters are skipped.
- Any parameter marked as "hidden" is always included.

Users decide whether to treat parameters from different models in the project as independent or communal (kept synchronized). In optiSLang, the Ansys Electronics Desktop integration has a **Shared Parameters View** with an additional **Include non-omnipresent parameters as shared** option.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In Ansys Electronics Desktop, project variables can be used to steer model parameters.

It is best to leverage features on both sides of the optiSLang-Electronics Desktop integration for managing overlapping (that is, partially synced) parameter sets. optiSLang's standard parametrization management tools offer the parameter type Dependent for aligning or otherwise controlling parameters.

In the context of bundling optiSLang-demanded designs for simultaneous execution as DSO jobs, remember that DSO design variation tables are hosted in the Optimetrics section of each design, and that there is no Optimetrics branch at the project level. So, even varying only project variables will still yield separate DSO tables and thus separate jobs for each model in the project.

A project walker mechanism inquires a given Electronics Desktop project and stores the generated project structure info in a JSON file for optiSLang. If the same reference project is linked for use in a different Electronics Desktop node even from a different optiSLang project, the stored project inquiry results can be recalled and the parametrization can be instantly displayed. This mechanism spares users wait time for the project inquiry call.

Outputs

The framework for result data transfer from Ansys Electronics Desktop to optiSLang is very simple: each report trace shows up in optiSLang response listings as a signal object. You can register the signals directly or apply math functions offered by optiSLang's calculator with each integration node. The transfer vessels are CSV files.

Even such plots as Smith charts or 3D polar plots can be collected as groups of signal objects using the uniform CSV file export function offered in Ansys Electronics Desktop.

Parameterization with optiSLang Analysis Goals in Mind

The parametrization scheme should connect to the goals of the optiSLang analysis.

Bad parametrization might hide the design optimum from the best optimizer or destroy the connection of a robustness sampling with reality. Thoughtfully improved parametrization can

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

achieve simulation goals with a fraction of the designs that would be needed for a task within the most simple ad hoc parametrization scheme.

The algorithms being applied in a specific scenario should also guide your parametrization scheme. For example, it makes sense to give a robustness sampling algorithm the chance to examine a parameter space where the simulation can represent the relevant physical effects happening in the real world. An optimizer algorithm, meanwhile, needs the chance to hit feasible designs frequently.

Some tips:

- Think ahead to devise a stable and meaningful parametrization
- Be attentive to the kinds of errors encountered by failing designs
- Be quick to eliminate the origins of errors that appear most frequently in error logs
- Learn to use optiSLang post-processing as a diagnostic toolbox
- Apply feedback and lessons learned to your parametrization schemes

Results and Reports for optiSLang Integration

AEDT can export simulation reports as CSV files that can be read in optiSLang.

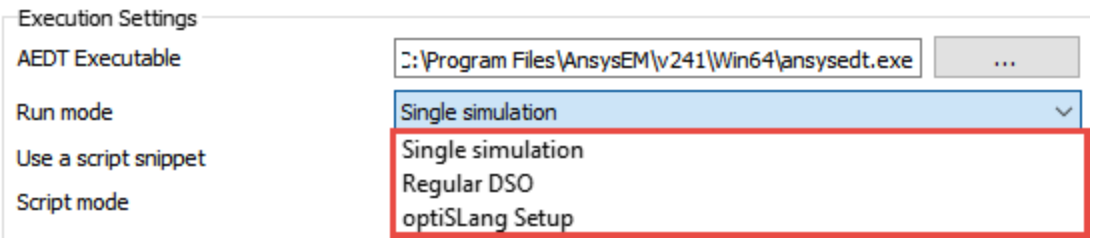
Additionally, all Optimetrics setups in AEDT (including optiSLang setups) have a **Calculations** tab, allowing for additional results. See: Setting up Calculations for Optimetrics.

When the scripted AEDT integration machinery within optiSLang exports reports, it is just as if a user had exported a CSV file from within the AEDT UI. Any reports that can be exported as CSV (rectangular plots, data tables, polar plots, etc.) are sent in CSV format, and the reports' traces can be registered as responses in optiSLang. As a general rule, each data column of a CSV file is exposed as a signal trace.

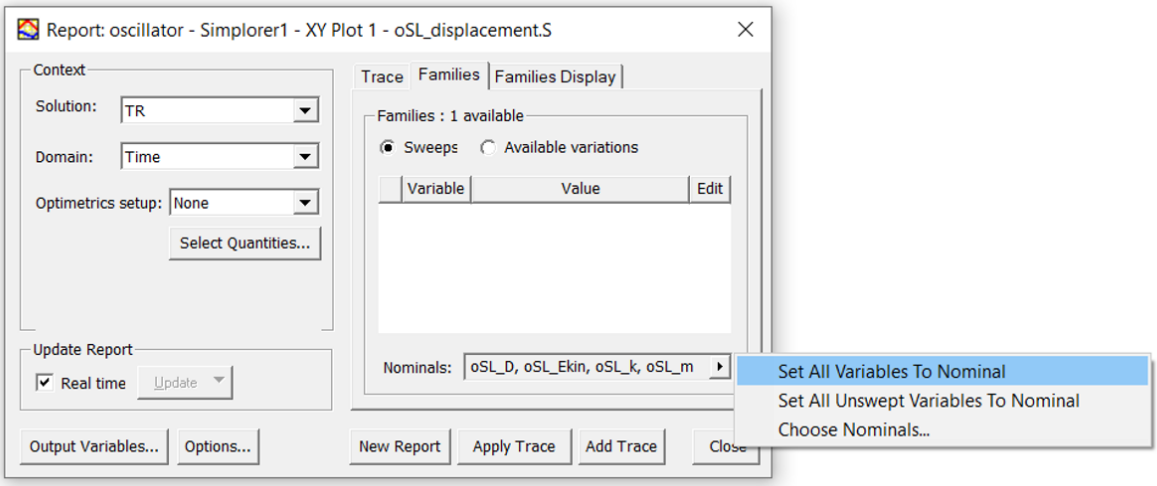
> **Note:**
>
> For 3D Polar Plots, this can cause a small disadvantage by yielding, for example, 180 signal objects instead of a matrix. However, the simplicity of the routine achieves good performance and robustness, and allows the AEDT integration node in optiSLang to cover a large array of use cases.

Below, find considerations for setting up reports for use in optiSLang's different run modes.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



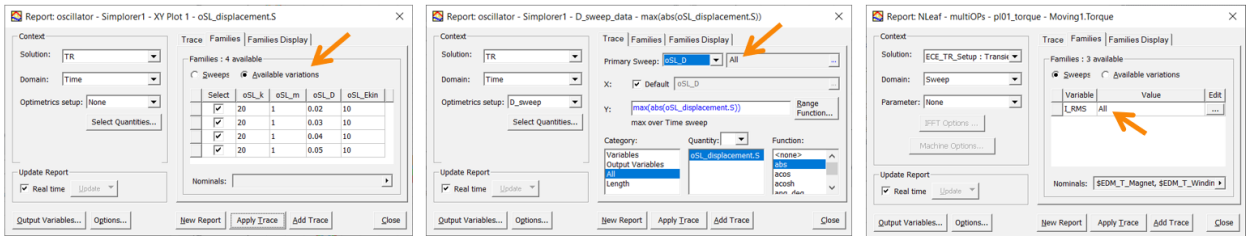
In optiSLang's **Single Simulation** run mode, optiSLang modifies the parameters of the nominal design and triggers the solution process. Therefore, reports used in this way should be configured to display the nominal design if they are to contain traces after modification and solution update. To ensure all report parameters are set to nominal, open the report settings and select the **Families** tab. From the **Nominals** field, select **Set All Variables to Nominal**.



In optiSLang's **Regular DSO** mode, the result export script cycles over all design variations of the Optimetrics setup created by previous script commands for holding the set of design variations demanded by optiSLang. In the loop, it applies the next design to be displayed as the nominal design, forces a refresh of all reports, and exports all reports. This loop is repeated for all design variations and the CSV files are stored each time in the corresponding design folder. Therefore, reports used in this mode should also be configured to display the nominal design. To ensure all report parameters are set to nominal, open the report settings and select the **Families** tab. From the **Nominals** field, select **Set All Variables to Nominal**.

In optiSLang's **optiSLang Setup** mode, the software solves a pre-existing Optimetrics setup (either serially or as a DSO job). This is commonly used to represent a parametric sweep or a prescribed set of operating points. In these use cases, reports should contain all data from all design variations to support a single export report call. To ensure the reports are adequate, open the report settings and enable all available variations:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

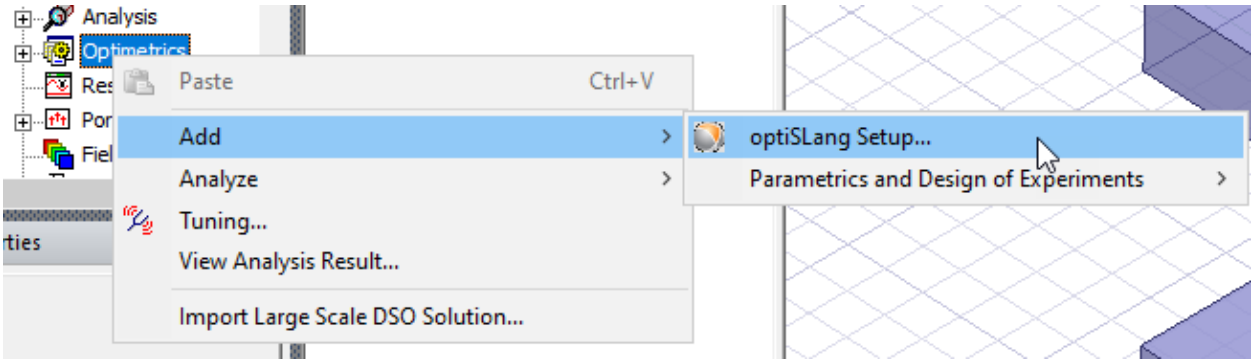


Creating an optiSLang Setup in AEDT

optiSLang setups are defined similarly to Optimetrics setups.

Create an optiSLang setup one of two ways:

- From the **Project Manager**, right-click **Optimetrics** and select **Add** > **optiSLang Setup**.
- Click **Maxwell** > **Optimetrics Analysis** > **Add optiSLang Setup**.

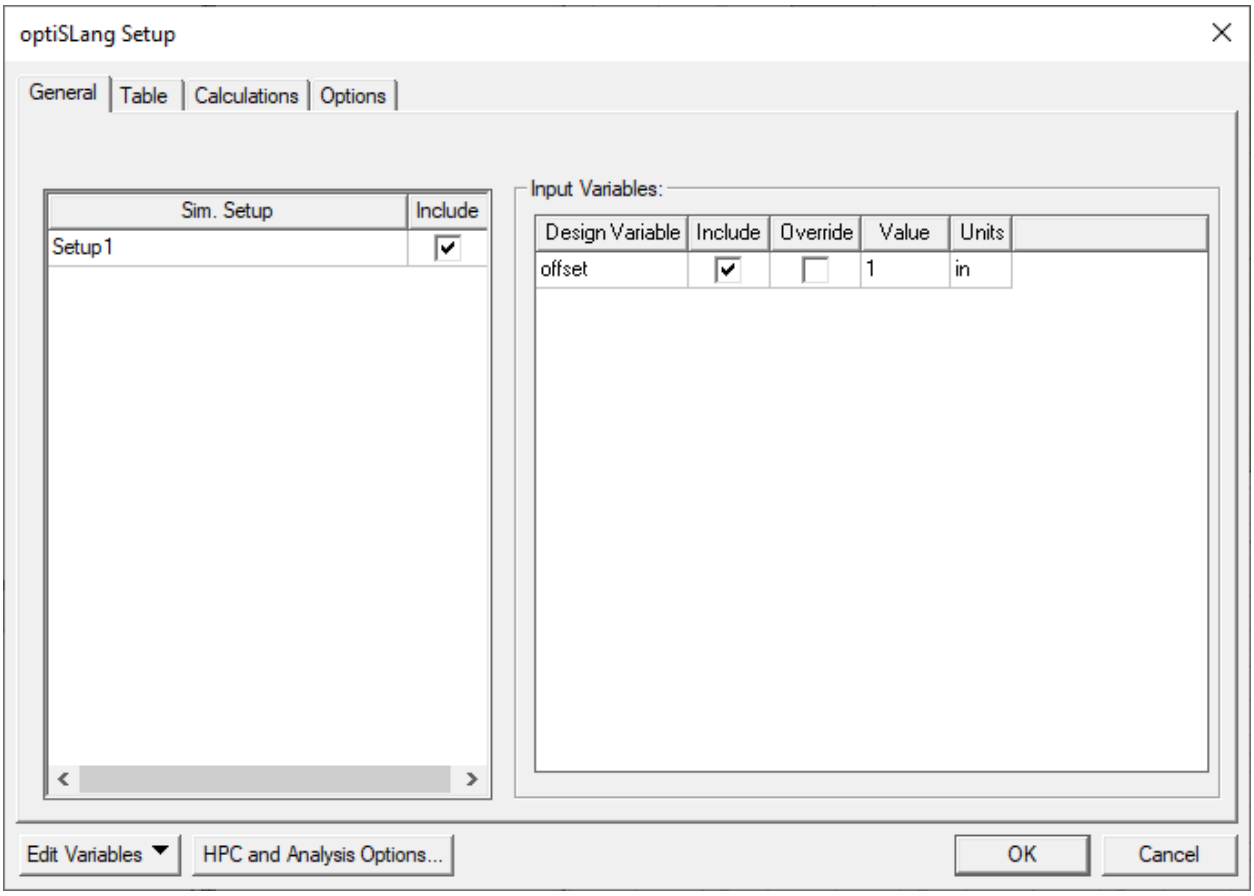


The **optiSLang Setup** window appears, with content based on the current design.

This window contains several tabs:

- **General** – lists solution setups and design variables associated with the current design, and allows you to specify which setups and variables to include, as well as to override a variable's value.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Table** – lists the values of the any variable(s) selected for inclusion.
- **Calculations** – allows for the setup of optiSLang calculations, similar to Optimetrics or Parametric setup. See: Setting up Calculations below.
- **Options** – lists mesh options. See: Setting Options below.

Each tab of the window also provides access to Edit Variables and HPC and Analysis Options.

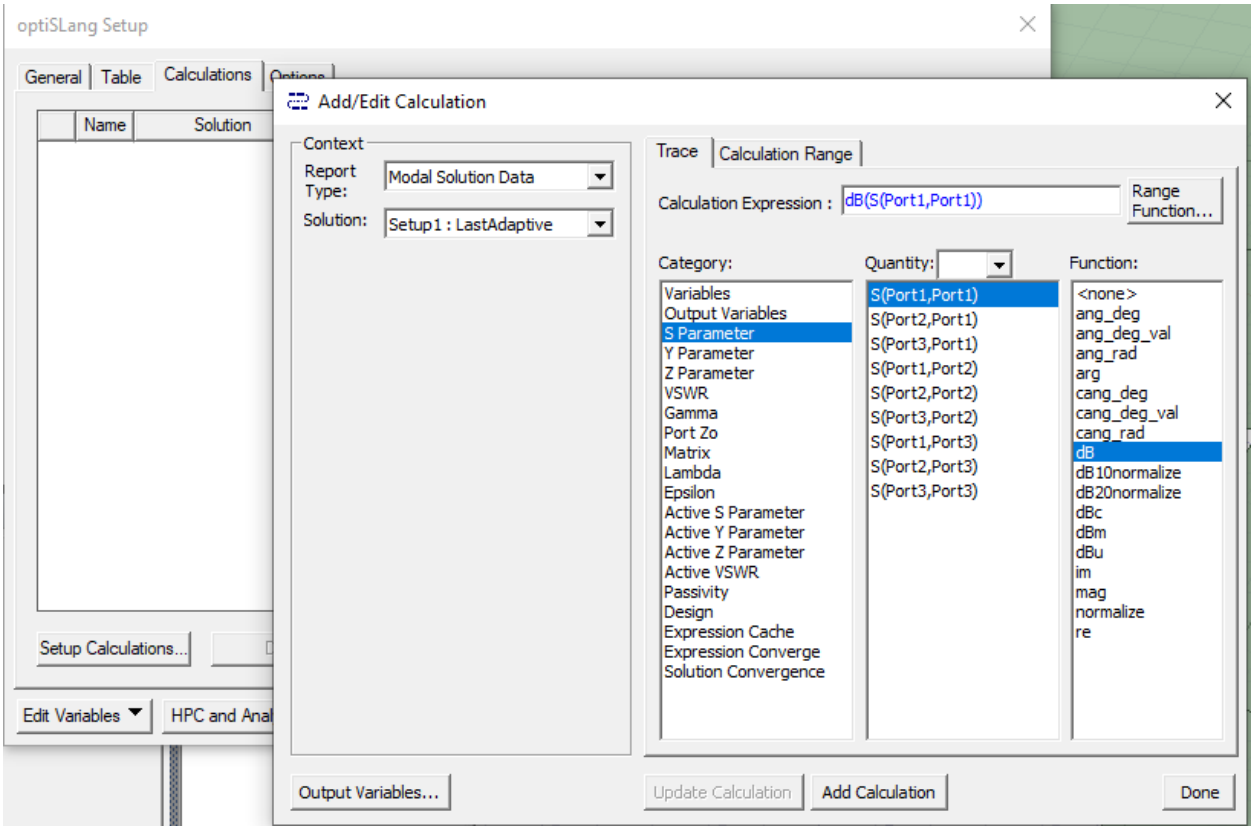
Setting up Calculations

Initially, the **Calculations** tab is empty.

To set up a calculation, click **Setup Calculations**.

The **Add/Edit Calculation** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This window contains **Context**, **Category**, **Quantity**, and **Function** fields that can be used to build an optimization calculation expression, just as for other optimization or parametric calculations.

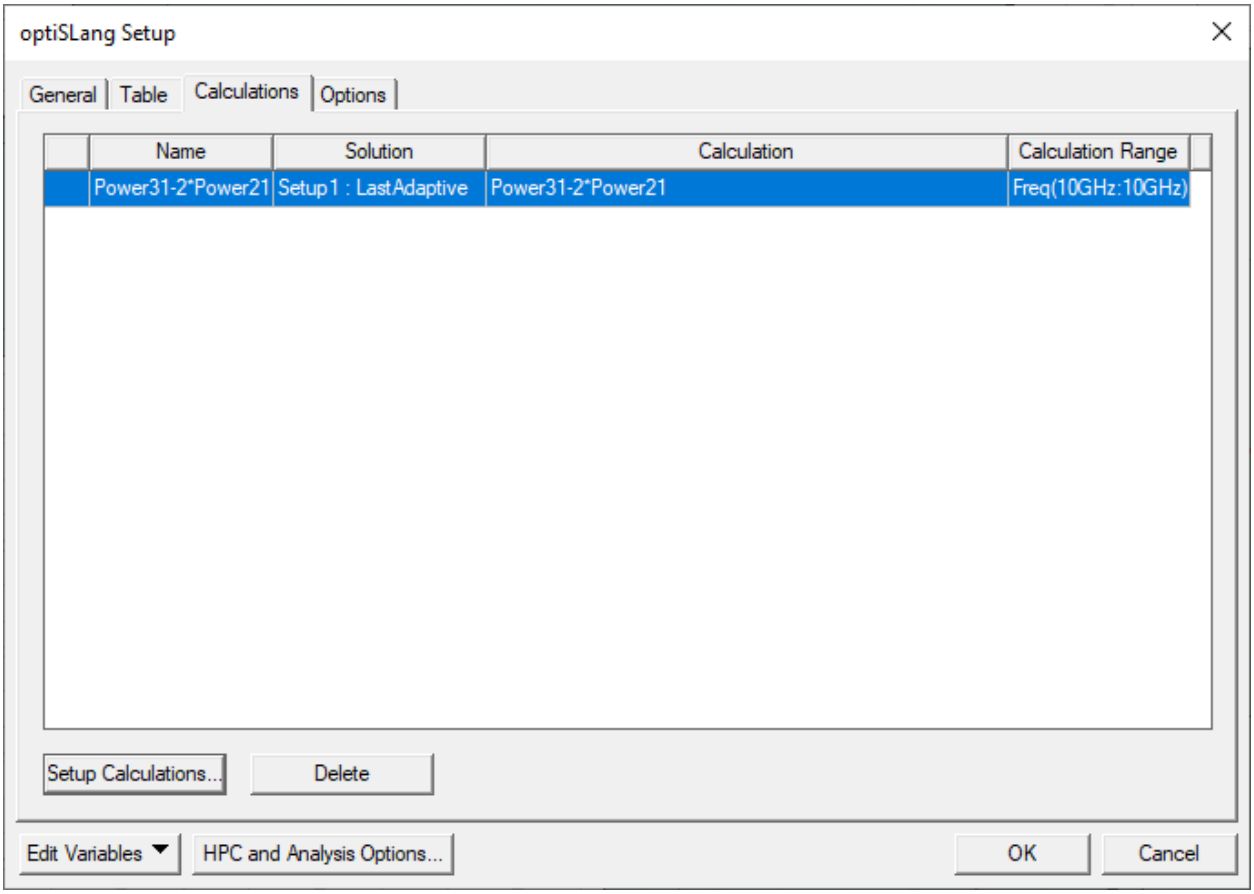
See: Setting Up Calculations for Optimetrics.

Once you have created an expression, click **Add Calculation** to add it to the **optiSLang** setup.

Click **Done** to close the **Add/Edit** window and return to the **Calculations** tab.

The new calculation is listed:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



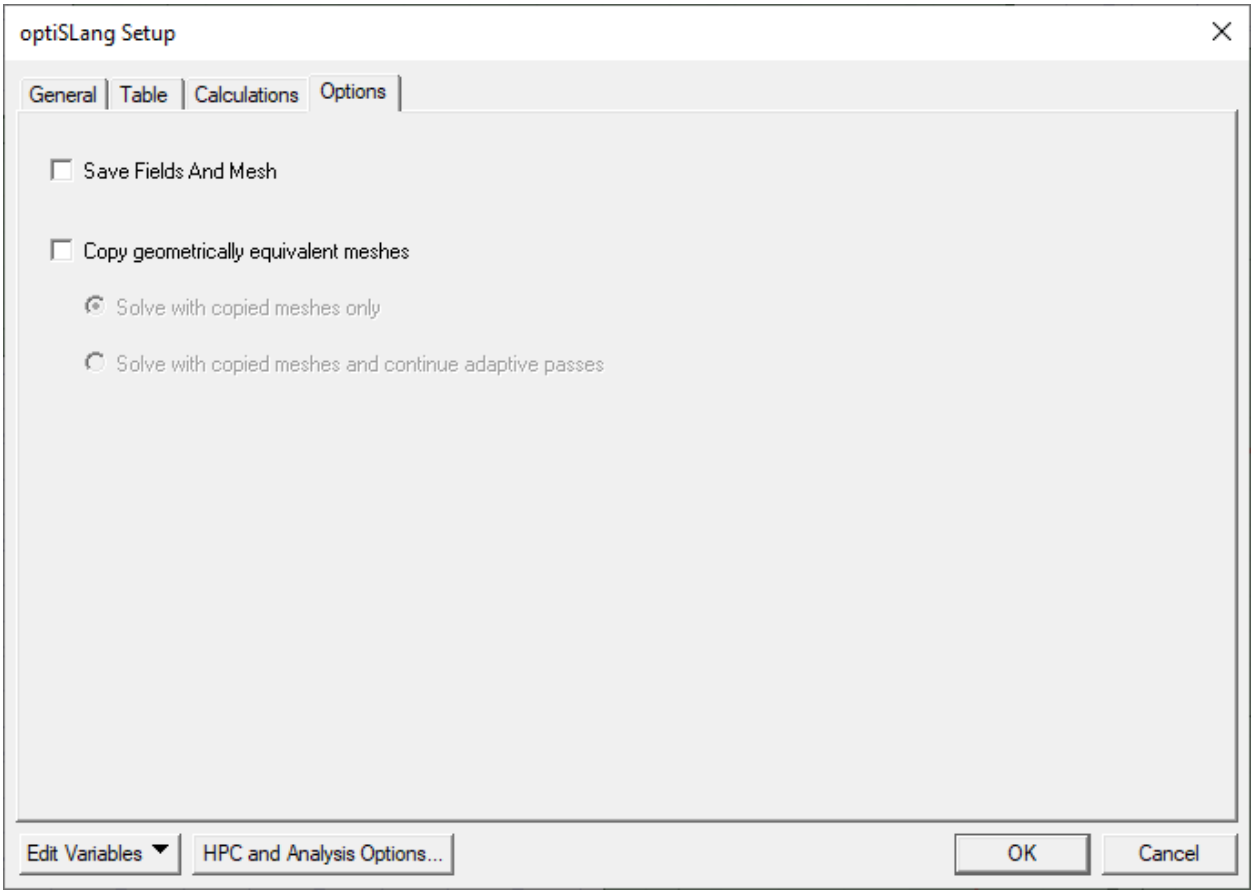
Setting Options

The **Options** tab contains options determining whether to **Save Fields and Mesh** and whether to **Copy geometrically equivalent meshes**.

In order to preserve disk space, by default AEDT does not save field solution data for every solved design variation. **Save Fields and Mesh** overrides this behavior.

Copy geometrically equivalent meshes directs AEDT to copy a mesh that was calculated for one sweep variation for reuse on a geometrically equivalent sweep variation. See: Copying Meshes in Optimetrics Sweeps.

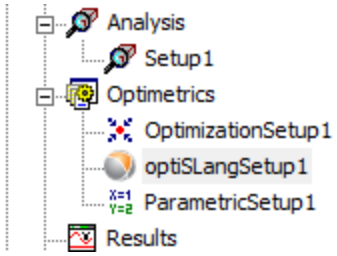
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Finishing Setup

Click **OK** when you have completed the setup.

The optiSLang setup appears in the **Project Manager**, under **Optimetrics**:



Example optiSLang Setups

The following are example scenarios illustrating different considerations that might be taken when preparing an optiSLang setup. See: Solving an optiSLang Setup.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Settings in these examples are handled both in AEDT and in optiSLang.

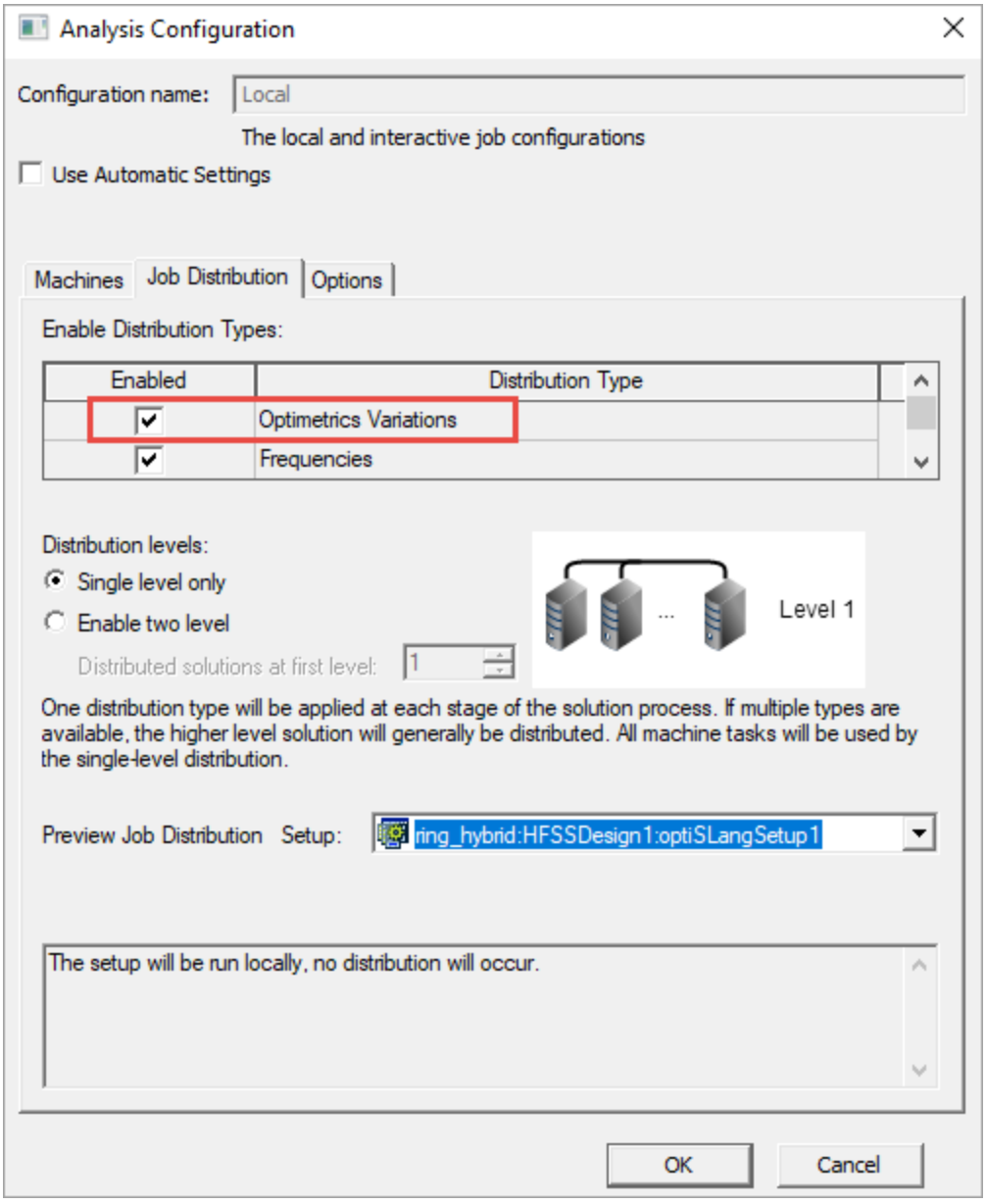
Example 1: Solving a User-created, Pre-existing optiSLang Setup

Recall that optiSLang setups are handled as Optimetrics setups in AEDT.

By preparing an optiSLang setup in an AEDT project, you can create regular DSO jobs even though the AEDT integration node is left in the single simulation run mode. In every design folder, the clone of the reference project contains the optiSLang setup and its solution is triggered under the **Analyze All** umbrella. Depending on the HPC settings active in AEDT for the model type in question, this yields optiSLang DSO jobs.

A key setting is the check box for enabling **Optimetrics Variations** in the **Job Distribution** tab of the **Analysis Configuration** window:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

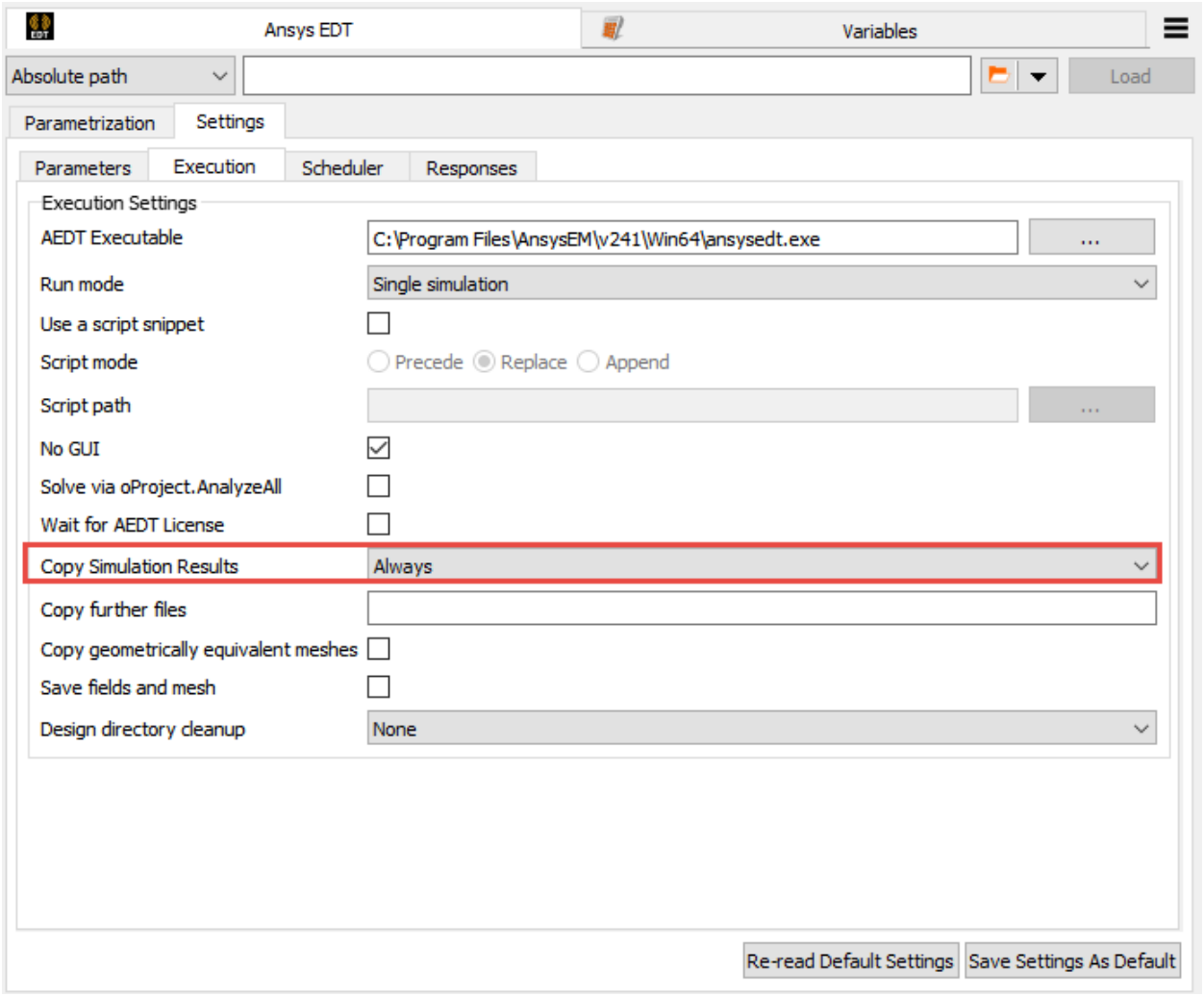
The **Use Automatic Settings** check box must be disabled to view this tab.

The working points being solved completely independently (simultanization) enables nice speed-up degrees for all designs within the optiSLang setup.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example 2: Increasing Performance by Avoiding Wasteful Solution of the Nominal Design

In Example 1, performance drag can exist if the analysis setup of the nominal design is also solved in every optiSLang design folder in series with a desired Optimetrics setup. Keeping the optiSLang setting **Copy Simulation Results** on the value **Always** can avoid redundant solution of the nominal design at the cost of the time for copying the *.aedtresults folder of the reference design repeatedly into every design folder.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example 3: Avoiding Redundant Computations while Exploiting AEDT Features around Postprocessing Variables

In optiSLang, an AEDT node can be used to copy a reference project along with its associated *.aedtresults directory of solution data. When this node is used for evaluating different combinations of postprocessing variables, AEDT does not call any solver and that the evaluation time per design is accordingly fast. The only drag consists in loading a fresh instance of the AEDT program for every new design.

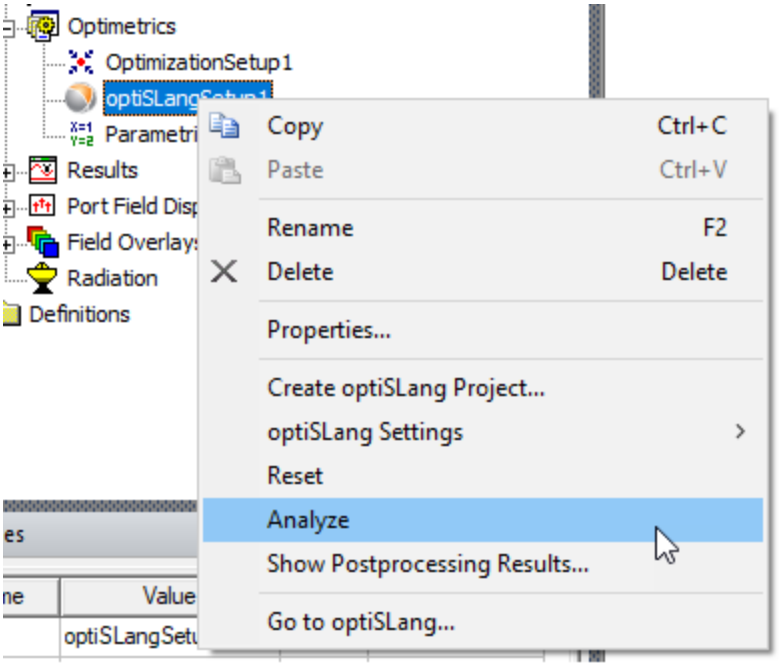
Example 4: Parametrization of an AEDT Design for optiSLang Integration

Consider a 2D Maxwell model of a permanent magnet synchronous motor. For design optimization, the magnet positions and angles can be the crucial parameters and due to symmetry a quarter model can represent the entire machine. For questions of robustness, reliability, tolerances, and achieving specs, it might be necessary to parametrize eccentricity and ellipticity. If magnet quality scatter is important, this might be another aspect requiring a full 360 degree model because it would be drastically unrealistic to assume that 16 worst-case magnets can find their way into one single machine. See: Parametrization for optiSLang Integration.

Solving an optiSLang Setup

To analyze an optiSLang setup within Electronics Desktop:

- From the **Project Manager**, right-click a setup and select **Analyze**.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Considerations for Using the Analyze All Command

The **Maxwell** > **Analyze All** command solves all projects in a design, including Optimetrics setups. See: Running More Than One Simulation.

> **Warning:** Before using the **Analyze All** command, remove any unnecessary analysis setups from the project.

Some reasons for doing so:

- During regular DSO run mode, optiSLang creates Optimetrics DSO design tables. By leaving just one setup for analysis, there is no ambiguity about which analysis setup is associated with any new table.
- Numerical convergence settings steer the performance trade-off between speed and accuracy. If a small number of designs take exceptionally long to converge, it can make a substantial performance difference whether designs are solved independently or in groups (DSO jobs). In group jobs, one slowly converging simulation can delay the entire group. This increases the importance of upper bounds on iterations for mesh refining, interpolation point addition, and so on.
- The unintended presence of Optimetrics setups (including optiSLang setups) can lead to long execution times. When used intentionally, however, Optimetrics setups can be used to define parametric sweeps. These sweeps are easy to set up and represent the sweep per design as a nested system in optiSLang. Note that reports must be adjusted accordingly, so that data is drawn from the entire sweep and not a single point, in order to reach the optiSLang database.
- Analysis setups contain settings that drastically impact the resulting file and folder sizes (for example, the **Save Fields** check box). If saved fields consume disk space in the range of gigabytes for a single project, then the creation of hundreds of design folders with solution data by optiSLang can quickly fill a hard drive.

Considerations for Distributed Analysis

There are additional considerations for using Distributed Analysis and Large-Scale DSO to analyze optiSLang setups.

- Using the AEDT node's run mode "regular DSO" is not the only way to create DSO jobs. Also in run mode "single simulation" DSO jobs can be spawned. For example, due to the **Analyze All** command, a user-created pre-existing Optimetrics setup could be solved for every one of optiSLang's design evaluations.
- The integration node's **Designs Per Execution** setting can work with HPC and DSO settings for AEDT solvers in many different ways. If set to 4, optiSLang can send 4 designs per AEDT call when AEDT is geared at solving 4 design variations

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

simultaneously. If **Designs Per Execution** is set to 9999, however, each optiSLang algorithm will always send all designs waiting in the pipeline at once (a robustness sampling of 1000 designs can be cast into one single DSO job). Using HPC settings, AEDT can still be made to break it down with ntasks=ncores=4. The drawback is that the success/failure info becomes visible in optiSLang only at the very end of the job. A large value for **Designs Per Execution** can make sense for optimization algorithms demanding design sets of fluctuating size.

- With population-based optimizers, it can be useful to match the population size with compute resources to avoid job load fluctuations.
- Several optiSLang nodes offer the feature to list files to be copied from the reference location into each design folder. However, this may not work for directories. A simple solution is to use a Python node and the shutil.copytree command.
- Avoid redundant solve actions. It may be helpful to copy not only the bare *.aedt file but also the *.aedtresults directory into each design folder. When the AEDT node issues the **Analyze All** command on the project level, pre-existing solution data can reduce the scope of analysis setups that are triggered to be solved. Compare the time for copying the data of solved projects against the time gain due to less computation. Avoid disc space concerns by activating design directory purging rules to be applied to every design folder after finished evaluation.
- AEDT offers a special type of postprocessing variables. When they are varied while other parameters stay constant, a new design variation can be evaluated with very little computational burden as compared to solving from scratch. The AEDT integration node provides copy functionalities for *.aedtresults folders allowing to benefit from AEDT features around postprocessing variables in certain use cases.

optiSLang Menu Options in AEDT

Once an optiSLang setup has been created in AEDT, additional menu options become available.

Right-click options include:

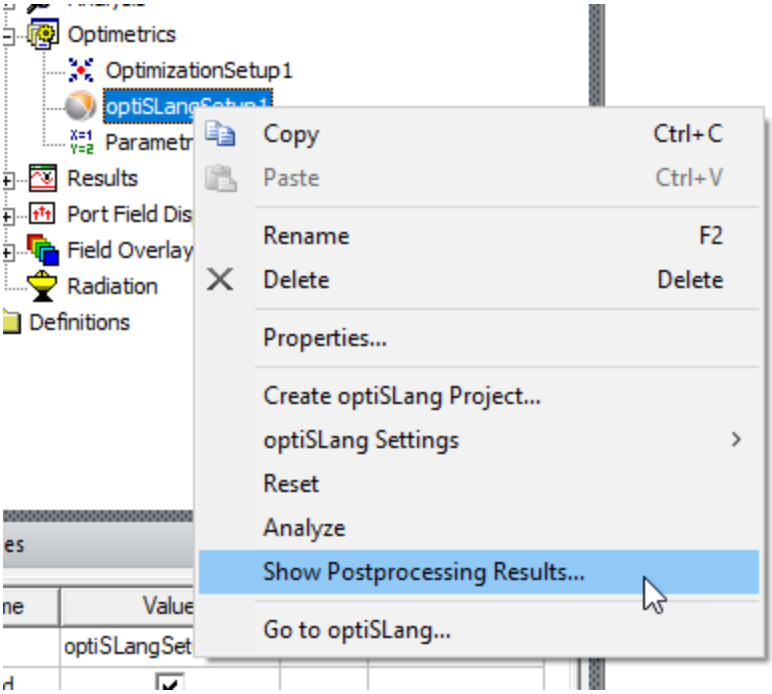
- **Create optiSLang Project** – creates an optiSLang project from the current design.
- **Edit AEDT Node Settings** – launches optiSLang with the AEDT project connected, on the Parametrization tab. For more information on node settings, consult the optiSLang help.
- **Edit Algorithm settings** – launches optiSLang and opens the Analysis Wizard, with options to analyze for Sensitivity, Optimization, or Robustness/Reliability. For more information on the Analysis Wizard, consult the optiSLang help.
- **Reset** – resets the optiSLang project.
- **Analyze** – launches the analysis of the optiSLang Optimetrics setup. When it has finished, post-processing results are available.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Show Postprocessing Results** – launches an optiSLang postprocessing results window. This menu option only available after analysis has been performed.
- **Go to optiSLang** – launches optiSLang.

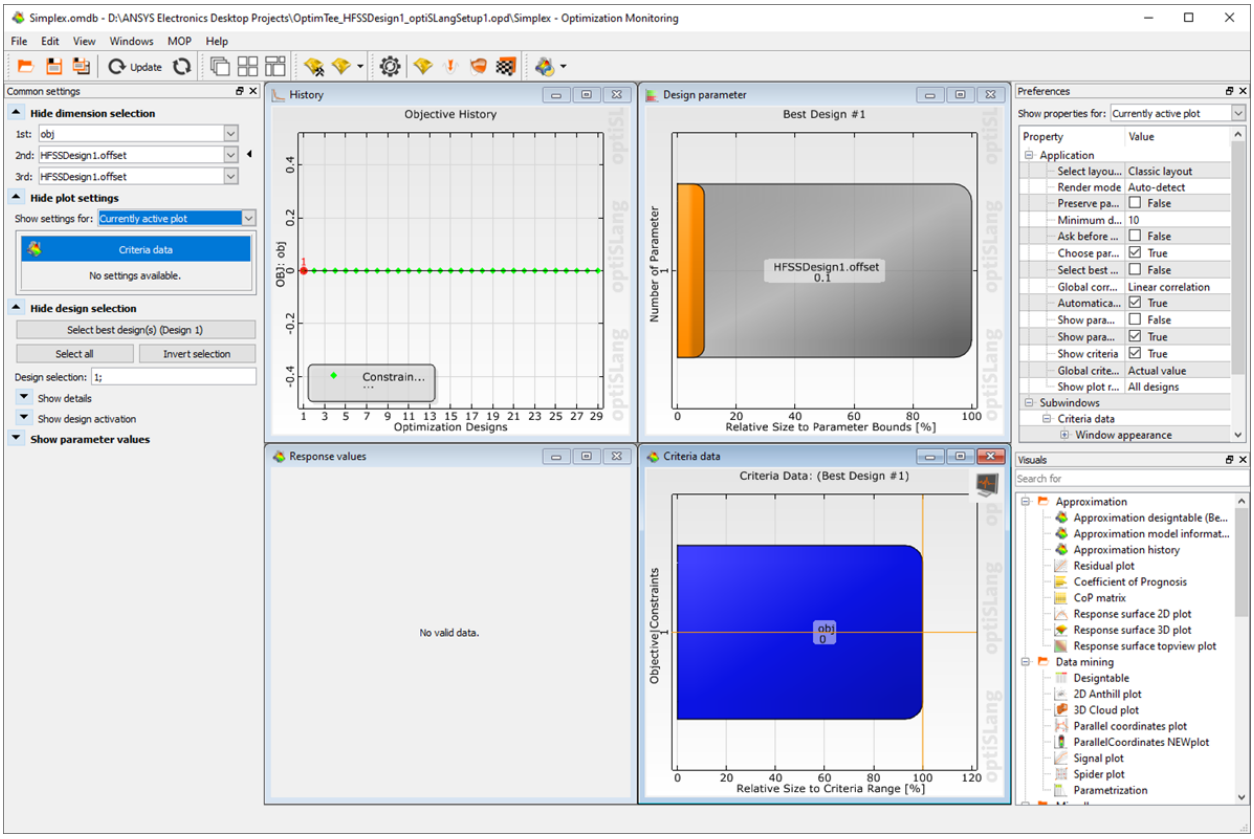
Viewing optiSLang Postprocessing Results

Once you have created and run an optiSLang setup, right-click it and select **Show Postprocessing Results**.



An optiSLang window opens, showing postprocessing features.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For more information, use optiSLang's **Help** menu to access the optiSLang online help.

Tuning Overview

Tuning a variable is useful when you want to manually modify its value and immediately perform an analysis of the design. For example, it is useful after performing an optimization analysis, in which Optimetrics has determined an optimal variable value, and you want to fine tune the value to see how the design results (for example, traces in a report) are affected.

A design can be updated after a tuning analysis to reflect a design variation solved during a tuning analysis and the results, including field solutions if you select **Save Fields and Mesh** on the **Options** tab of the associated setup dialog box.

Tuning a Variable

If you want to ensure that tuning does not resolve variations already solved by an optimization setup, you must check **Save Fields and Mesh** in the **Options** tab of that setup.

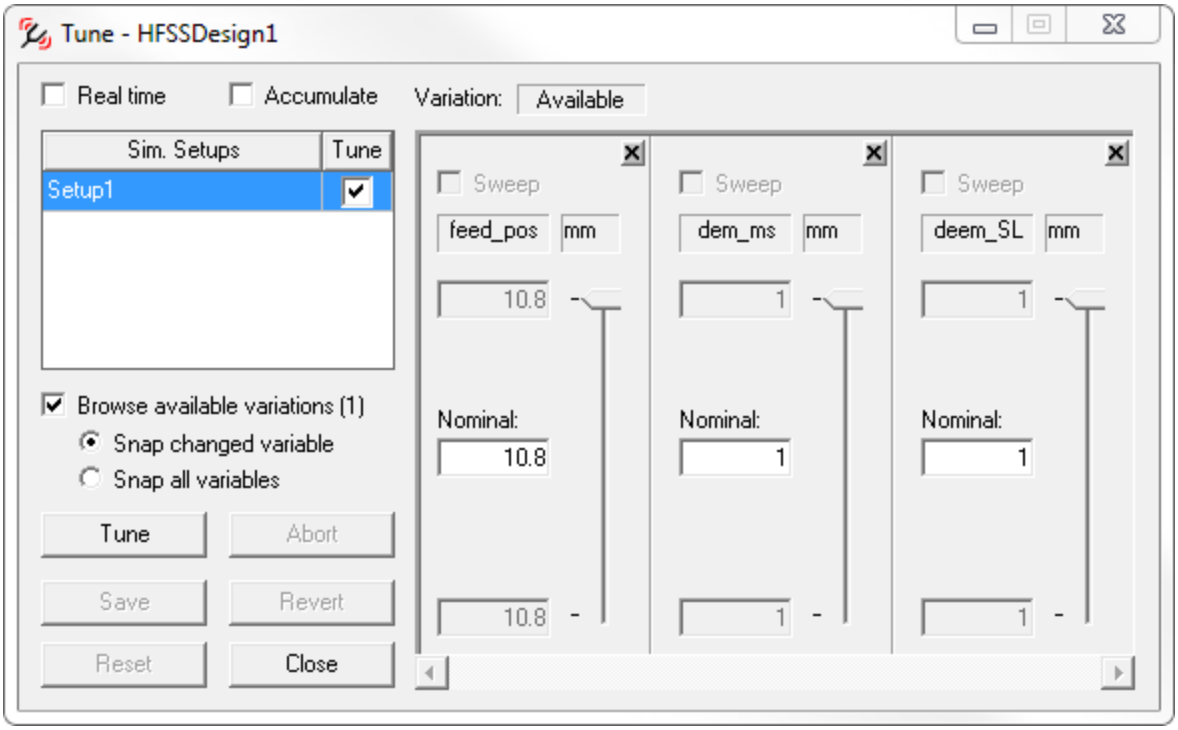
1. Before a variable can be tuned, you must specify that you intend for it to be used during a tuning analysis in a Project or Design **Properties** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. After running the simulation, click the product in the menu bar and then select **Tune**.

   Alternatively, right-click **Optimetrics** in the Project Manager and choose **Tuning** from the shortcut menu.

   The **Tune** dialog box appears, listing the variables which have been included for tuning.



3. Clear the **Real Time** option.

   Clearing the **Real Time** option enables the **Tune** button. If this option is selected, a simulation begins immediately after you move the slider. Otherwise, you use the **Tune** button to apply the current values to a simulation.

4. If you want to see updates to an open Report plot while tuning a post processing variable, you must select the **Browse available variations** check box. Selecting **Browse available variations** disables the sweep check box, and the fields for minimum and maximum variable values. This feature lets you see the effect of changes to the post processing variables on plotted results.

   Clearing **Browse available variations** enables the Sweep check box, the minimum and maximum fields, and changes the **Nominal** field to **Step**. (See step 6.)

5. In the **Sim. Setups** column, select the solution setup you want to use when it solves the specified design variation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The analysis is solved using the solution setup you select. If you select more than one, results are generated for all selected solution setups.

Checking the Tune box for a Sim Setup enables the Real Time check box, the Browse available variations check box, and the Snap radio buttons. Clearing the Tune box disables those selections.

6. In the **Nominal** text box for the variable you want to tune, type the value of the variable you want to solve, or drag the slider to increase or decrease its value.

   > **Warning:**
   >
   > Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any optimetric analysis.

   Alternatively, if you want to solve a range of values, specify a linear range of values with a constant step size:

   a. Select the **Sweep** check box. (You must have cleared the **Browse available variations** check box).
   b. In the text box below the **Step** value, type the starting value in the variable range.
   c. Type the step size, or difference between variable values in the sweep definition, in the **Step** text box. The step size determines the number of design variations between the start and stop values. The model is solved at each step in the specified range, including the start and stop values.
   d. In the text box just below the variable name, type a stopping value in the variable range.

7. If you have cleared the Real Time check box, click **Tune** to apply the changes you have made to the variable values.

   > **Note:**
   >
   > Sweeping or using a complex variable is not allowed in any optimetrics setup, including optimization, statistical, sensitivity, and tuning setups.

8. Changing a variable value with the sliders of by typing in the text field enables the **Save** and **Reset** buttons.

   Clicking **Save** opens a **Save As** dialog box with a name field and an **Apply tuned values to design** check box.

   Clicking **Reset** changes the variable values back to what they were originally.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. If you have changed one or more included variables, clicking **Close** on the **Tuning** dialog box opens the **Apply Tuned Variation** dialog box. This lists the included variables and the values for each tuning. If you have tried multiple values, they are listed, and the current value is highlighted. Select another value to change the highlight. Click **OK** to apply the highlighted values to the design, or **Don't Apply** to ignore the changes from the original variable values.

   If you have applied variant values, you should see the new values listed in the relevant Design or Project Properties lists of variables and values, and if the changes affect plots or physical features of a model, those changes should also appear.

   Click **Cancel** to close the dialog box and go back to the **Tune** dialog box.

Applying a Tuned State to a Design

You can apply the variable values solved during a tuning analysis to the nominal design in one of the following three ways:

- When closing the **Tune** dialog box
- When saving a tuned state
- When reverting to a tuned state

When closing the **Tune** dialog box:

1. Click **Close** to exit the **Tune** dialog box.

   The **Apply Tuned Variation** dialog box appears.

2. Click the design variation you want to apply, and then click **OK**.

   The variable values from the solved design variation become the current variable values for the nominal design. If you have applied variant values, you should see the new values listed in the relevant Design or Project Properties lists of variables and values, and if the changes affect plots or physical features of a model, those changes should also be apparent.

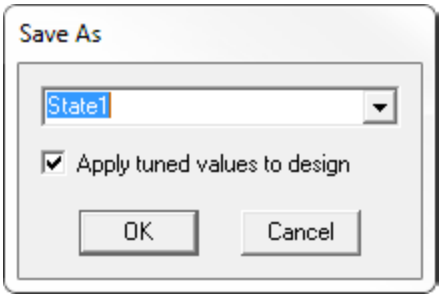
Saving a Tuned State

You can save the settings in the **Tune** dialog box, including the variable values you specified for a tuning analysis. Saved states are only available during the current session of the **Tune** dialog box; they are not stored for the next session.

1. After tuning a variable, click **Save** in the **Tune** dialog box.

   A **Save As** dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Type a name for the tuned state in the text box.
3. Select **Apply tuned values to design** if you want to update the model to the new variable values.
4. Click **OK** to return to the **Tune** dialog box.

Resetting Variable Values after Tuning

If you want to reset variable values to the values they were set to when you started the current session of the **Tune** dialog box:

- After tuning a variable, click **Reset** in the **Tune** dialog box.

  Solutions for the design variations solved during tuning analyses remain available for post processing.

Reverting to a Saved Tuned State

You can revert to a group of saved settings in the **Tune** dialog box, including the variable values you specified for a specific tuning analysis. Saved states are only available during the current session of the **Tune** dialog box; they are not stored for the next session.

1. In the **Tune** dialog box, click **Revert**.

   The **Revert** dialog box appears.

2. Type the name of the tuned state you want to apply or click a name in the drop-down menu.
3. Select **Apply tuned values to design** if you want to update the model to the selected tuned state's variable values.
4. Click **OK** to return to the **Tune** dialog box.

Saving Field Solutions for Optimetrics Analyses

In order to preserve disk space, by default Ansys Electronics Desktop does not save field solution data for every solved design variation in an optimization analysis. It only saves the field

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

solutions for the nominal design when an adaptive analysis is specified in the solution setup or when you request that fields be saved for each solved point in a sweep. If the nominal design is not included in the optimization analysis, all field solutions are deleted.

You can change the general options for a particular design type (HFSS, Q3D, and so on) so that any new Optimetrics setup you define will default to saving the fields. The instructions to do so are included on this page.

Alternatively, you can choose to save the fields on an individual setup basis, without enabling the option to save fields by default. See the subtopics listed in the Related Topics section at the bottom of this page for instructions to do so for each type of optimetrics setup.

To save the fields for all design variations, change the default setting for all projects:

1. Using the menu bar, select **Tools** > **Options** > **General Options**, or from the **Desktop** ribbon tab, click **General Options**.

   The **Options** dialog box appears.

2. In the tree on the left side of the dialog box, expand the branch for the product/design type of interest (HFSS, Q3D, Mechanical, and so on).
3. Select **General**.

   The appropriate group of options appears on the right side of the dialog box.

4. Select **Save Optimetrics field solutions**.
5. Repeat steps 2 through 4 for all products in which you want to save field results by default for all new Optimetrics setups.

   The **Save Fields** option will now be selected by default when you create a new Optimetrics setup in each product for which you selected the **Save Optimetrics field solutions** option.

Saving Field Solutions for a Parametric Setup

In order to preserve disk space, by default Ansys Electronics Desktop does not save field solution data for every solved design variation in a parametric setup. It only saves the field solutions for the nominal design. If the nominal design is not included in the parametric setup, by default field solutions will not be available.

To save the fields for all design variations solved during a parametric analysis:

1. Right-click **Optimetrics** in the Project Manager and choose **Add** > **Parametric** from the shortcut menu to create a new parametric setup.

   Alternatively, to change the settings of an existing setup, right-click the desired Parametric setup under **Optimetrics** in the Project Manager and choose **Properties**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Setup Sweep Analysis** dialog box appears.

2. Select the **Options** tab.
3. Select **Save Fields And Mesh**. Optionally, select **Copy geometrically equivalent meshes**, and choose whether to **Solve with copied meshes only**, or to **Solve with copied meshes and continue adaptive passes**.

   Ansys Electronics Desktop will save the field solution data for every solved design variation in the parametric setup.

> **Note:**
>
> Alternatively, you can modify the Ansys Electronics Desktop General Options to save fields by default for any new Optimetrics setup. See: Saving Field Solutions for Optimetrics Analyses for instructions.
>
> Changing the default setting in the General Options has no effect on previously defined Optimetrics setups.

Saving Field Solutions for an Optimization Setup

In order to preserve disk space, by default Ansys Electronics Desktop does not save field solution data for every solved design variation in an optimization analysis. It only saves the field solutions for the nominal design when an adaptive analysis is specified in the solution setup or when you request that fields be saved for each solved point in a frequency sweep. If the nominal design is not included in the optimization analysis, all field solutions are deleted.

To save the fields for all design variations solved during an optimization analysis:

1. Right-click **Optimetrics** in the Project Manager and choose **Add** > **Optimization** from the shortcut menu to create a new optimization setup.

   Alternatively, to change the settings of an existing setup, right-click the desired Optimization setup under **Optimetrics** in the Project Manager and choose **Properties**.

   The **Setup Optimization** dialog box appears.

2. Select the **Options** tab.
3. Select **Save Fields And Mesh**. Optionally, select **Copy geometrically equivalent meshes**, and choose whether to **Solve with copied meshes only**, or to **Solve with copied meshes and continue adaptive passes**.

   Ansys Electronics Desktop will save the field solution data for every solved design variation in the optimization setup.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Alternatively, you can modify the Ansys Electronics Desktop General Options to save fields by default for any new Optimetrics setup. See: Saving Field Solutions for Optimetrics Analyses for instructions.

Changing the default setting in the General Options has no effect on previously defined Optimetrics setups.

Saving Field Solutions for a Sensitivity Setup

In order to preserve disk space, by default Ansys Electronics Desktop does not save field solution data for every solved design variation in a sensitivity analysis. It only saves the field solutions for the nominal design when an adaptive analysis is specified in the solution setup or when you request that fields be saved for each solved point in a frequency sweep. If the nominal design is not included in the sensitivity analysis, all field solutions are deleted.

To save the fields for all design variations solved during a sensitivity analysis:

1. Right-click **Optimetrics** in the Project Manager and choose **Add** > **Sensitivity** from the shortcut menu to create a new sensitivity analysis setup.

   Alternatively, to change the settings of an existing setup, right-click the desired Sensitivity analysis setup under **Optimetrics** in the Project Manager and choose **Properties**.

   The **Setup Sensitivity Analysis** dialog box appears.

2. Select the **Options** tab.
3. Select **Save Fields And Mesh**. Optionally, select **Copy geometrically equivalent meshes**, and choose whether to **Solve with copied meshes only**, or to **Solve with copied meshes and continue adaptive passes**.

   Ansys Electronics Desktop will save the field solution data for every solved design variation in the sensitivity analysis.

Note:

Alternatively, you can modify the Ansys Electronics Desktop General Options to save fields by default for any new Optimetrics setup. See: Saving Field Solutions for Optimetrics Analyses for instructions.

Changing the default setting in the General Options has no effect on previously defined Optimetrics setups.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Saving Field Solutions for a Tuning Analysis

In order to preserve disk space, by default Maxwell does not save field solution data for every design variation solved in a tuning analysis. It only saves the field solutions for the nominal design when an adaptive analysis is specified in the solution setup or when you request that fields be saved for each solved point in a frequency sweep. If the nominal design is not included in the tuning analysis, all field solutions are deleted.

To save the fields for all design variations solved during a tuning analysis:

In the **Tuning** dialog box, select **Save Fields**.

Maxwell will save the field solution data for every solved design variation in a tuning analysis.

Saving Field Solutions for a Statistical Setup

In order to preserve disk space, by default Ansys Electronics Desktop does not save field solution data for every design variation solved in a statistical analysis. It only saves the field solutions for the nominal design when an adaptive analysis is specified in the solution setup or when you request that fields be saved for each solved point in a frequency sweep. If the nominal design is not included in the statistical analysis, all field solutions are deleted.

To save the fields for all design variations solved during a statistical analysis:

1. Right-click **Optimetrics** in the Project Manager and choose **Add** > **Statistical** from the shortcut menu to create a new statistical analysis setup.

   Alternatively, to change the settings of an existing setup, right-click the desired Statistical analysis setup under **Optimetrics** in the Project Manager and choose **Properties**.

   The **Setup Statistical Analysis** dialog box appears.

2. Select the **Options** tab.
3. Select **Save Fields And Mesh**. Optionally, select **Copy geometrically equivalent meshes** and choose whether to **Solve with copied meshes only**, or to **Solve with copied meshes and continue adaptive passes**.

   Ansys Electronics Desktop will save the field solution data for every solved design variation in the statistical analysis.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Alternatively, you can modify the Ansys Electronics Desktop General Options to save fields by default for any new Optimetrics setup. See: Saving Field Solutions for Optimetrics Analyses for instructions.

Changing the default setting in the General Options has no effect on previously defined Optimetrics setups.

Saving Field Solutions for a Design of Experiments Setup

In order to preserve disk space, by default Ansys Electronics Desktop does not save field solution data for every solved design variation in a Design of Experiments setup. It only saves the field solutions for the nominal design. If the nominal design is not included in the Design of Experiments setup, by default field solutions will not be available.

To save the fields for all design variations solved during a Design of Experiments analysis:

1. Right-click **Optimetrics** in the Project Manager and choose **Add** > **Design of Experiments** from the shortcut menu to create a new Design of Experiments setup.

   Alternatively, to change the settings of an existing setup, right-click the desired Design of Experiments setup under **Optimetrics** in the Project Manager and choose **Properties**.

   The **Design of Experiments Setup** dialog box appears.

2. Select the **Options** tab.
3. Select **Save Fields And Mesh**. Optionally, select **Copy geometrically equivalent meshes** and choose whether to **Solve with copied meshes only**, or to **Solve with copied meshes and continue adaptive passes**.

   Ansys Electronics Desktop will save the field solution data for every solved design variation in the design of experiments setup.

Note:

Alternatively, you can modify the Ansys Electronics Desktop General Options to save fields by default for any new Optimetrics setup. See: Saving Field Solutions for Optimetrics Analyses for instructions.

Changing the default setting in the General Options has no effect on previously defined Optimetrics setups.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copying Meshes in Optimetrics Sweeps

An option in the Optimetrics Analysis setups allows you to request Ansys Electronics Desktop to copy a mesh that was calculated for one sweep variation for reuse on a geometrically equivalent sweep variation. For example, with this option selected a sweep on a scan angle would not need to generate meshes for each solution. The option is available on the setups for sweeps on parametrics, optimization, sensitivity, and statistics.

To copy and reuse meshes on geometrically equivalent parametric variations:

1. Define a variable for the kind of Optimetrics sweep you intend to set up.
2. Select Ansys Electronics Desktop and then select the appropriate **Optimetrics** > **Add** command to display the associated **Setup** dialog box.
3. Click the **Options** tab in the setup dialog box.
4. Select **Copy geometrically equivalent meshes**.
    - When this option is enabled, you can additionally select **Solve with copied meshed only** or **Solve with copied meshes and continue adaptive passes**. The **Solve with copied meshes only** option is not available for Maxwell 3D/2D magnetic and electric transient designs. It is available for HFSS Transient but does not apply for a Transient solve setup with a mesh link.

    Ansys Electronics Desktop copies the mesh for a particular parametric sweep for reuse on each geometrically equivalent sweep variation.

> **Note:**
>
> This option is available with all Optimetrics setups, and is applied when these analyses generate geometrically equivalent values. However, it is most relevant to parametric sweep, where such equivalences are more likely to occur.

The **Copy geometrically equivalent mesh** option is not recommended for use when the frequency is varying, since meshing is frequency-dependent. You may wish to turn this option off when the first geometrically equivalent variation requires numerous passes after the initial mesh, but the other geometrically equivalent variations require fewer additional passes, so that it is cheaper to start with the initial mesh each time.

Adding an Expression in the Output Variables Window

When you are in the **Output Variables** window (after clicking **Edit Calculation** from one of the setup analysis windows), do the following to specify an expression:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Type a name for the expression in the **Name** text box.
2. Do the following in the **Calculation** section of the window to insert a quantity into the expression:
    a. Select the **Report Type** and **Solution** from the drop-down menus.
    b. Select a **Category**, **Quantity**, and **Function** from the lists, and click **Insert Quantity Into Expression**.
    c. If you want to insert a specific predefined function, select one from the **Function** drop-down menu, and click **Insert Function**.
3. You can also type numbers or expression by hand directly into the **Expression** area.

Excluding a Variable from an Optimetrics Analysis

To exclude a variable from being optimized or included in a sensitivity or statistical analysis:

1. Do one of the following:
    - In the **Setup Optimization** dialog box, click the **Variables** tab.
    - In the **Setup Sensitivity Analysis** dialog box, click the **Variables** tab.
    - In the **Setup Statistical Analysis** dialog box, click the **Variables** tab.

    All of the independent variables that were selected for the optimization analysis are listed.
2. Clear the **Include** option for the variable you want to exclude from the analysis.

    The **Override** option is now selected. This indicates that, for this optimization analysis, the variable is not included.

    > **Note:**
    >
    > Alternatively, you can select the **Override** option first, and then clear the **Include** option for the variable you want to exclude.

3. Click **OK**.

Modifying the Value of a Fixed Variable

If you are not including a variable in an optimization, sensitivity, or statistical analysis, Optimetrics uses that variable's current value during the analysis.

To override the current value of a fixed variable for an Optimetrics setup:

1. Do one of the following:
    - In the **Setup Optimization** dialog box, click the **Variables** tab.
    - In the **Setup Sensitivity Analysis** dialog box, click the **Variables** tab.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- In the **Setup Statistical Analysis** dialog box, click the **Variables** tab.

2. Click **Set Fixed Variables**.

   The **Setup Fixed Variables** dialog box appears. Under **Fixed Variables**, all of the current independent variable values are listed.

3. Click the **Value** text box of the variable with the value you want to override.
4. Type a new value in the **Value** text box, and then press **Enter**.

   The **Override** option is now selected. This indicates that the value you entered is used for this Optimetrics setup; the current variable value set for the nominal design is ignored.

   > **Note:**
   >
   > Alternatively, you can select the **Override** option first, and then type a new value in the **Value** text box.

5. Optionally, click a new unit system in the **Units** text box.
6. Click **OK**.

To revert to a default variable value, clear the **Override** option.

Linear Constraints

Once the optimization variables are specified, the optimizer handles each of them as an *n*-dimensional vector *x*. Any point in the design space corresponds to a particular *x*-vector and to a design instance. Each design instance may be evaluated via Finite Element Analysis and assigned a cost value; therefore, the cost function is defined over the design space

$$\mathrm{cost}(x): R^n \rightarrow R$$

where *n* is the number of optimization variables.

In practice, a solution of the minimization problem is sought only on a bounded subset of the R^n space. This subset is called the feasible domain and is defined via linear constraints.

You may constrain the feasible domain of a design variable by defining linear constraints for the optimization process. The feasible domain is defined as the domain of all design variables that satisfy all upper and lower bounds and constraints. Linear constraints are defined by the following inequalities:

$$\sum_i \alpha_{ij} x_i < c_j \forall j$$

where

- α_{ij} are coefficients.
- c_j is a comparison value for the j^{th} linear constraint.
- x_i is the i^{th} parameter.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting a Linear Constraint

A linear constraint defines the linear relationship between variables. Setting linear constraints in Optimetrics is useful for establishing limitations involving linear combinations of variable values.

> **Important:**
>
> Linear constraints may only be applied to the following optimizers:
>
> - MATLAB
> - Sequential Nonlinear Programming (Gradient)
> - Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete)
> - Quasi Newton (Gradient)
> - Pattern Search (Search-based)
> - Genetic Algorithm (Random Search)

To set a linear constraint:

1. Access the **Variables** tab one of two ways:
   - If you are setting up an optimization analysis: In the **Setup Optimization** dialog box, click the **Variables** tab.
   - If you are setting up a sensitivity analysis: In the **Setup Sensitivity Analysis** dialog box, click the **Variables** tab.
2. Click **Linear Constraint**.

   The **Linear Constraint** dialog box appears.
3. Click **Add**.

   The **Edit Linear Constraint** dialog box appears.
4. Click a **Coeff** text box and type a positive or negative coefficient value.
5. Click a condition, **<** (less than) or **>** (greater than), from the drop-down menu.
6. Type the inequality value, which should be a constant value, in the text box to the right of the condition.
7. Click **OK**.

   You return to the **Linear Constraint** dialog box. The left-hand side of the constraint appears in the **LHS** (left-hand side) column. The condition is listed in the **Condition** column, and the inequality value is listed in the **RHS** (right-hand side) column.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modifying a Linear Constraint

1. Do one of the following:
   - If you are setting up an optimization analysis: In the **Setup Optimization** dialog box, click the **Variables** tab.
   - If you are setting up a sensitivity analysis: In the **Setup Sensitivity Analysis** dialog box, click the **Variables** tab.
2. Click **Linear Constraint**.

   The **Linear Constraint** dialog box appears.
3. Click the row listing the constraint you want to modify, and then click **Edit**.

   The **Edit Linear Constraint** dialog box appears.
4. Optionally, click a **Coeff** text box and type a new coefficient value.
5. Optionally, click a different condition, **<** (less than) or **>** (greater than), in the pull-down list.
6. Optionally, type a different inequality value in the text box to the right of the condition, and then click **OK**.

   You return to the **Linear Constraint** dialog box. The new coefficient value, the condition, and the inequality value appear in the **LHS** (left-hand side), **Condition**, and **RHS** (right-hand side) columns, respectively.

Deleting a Linear Constraint

1. Do one of the following:
   - If you are setting up an optimization analysis: In the **Setup Optimization** dialog box, click the **Variables** tab.
   - If you are setting up a sensitivity analysis: In the **Setup Sensitivity Analysis** dialog box, click the **Variables** tab.
2. Click **Linear Constraint**.

   The **Linear Constraint** dialog box appears.
3. Click the row listing the constraint you want to delete, and then click **Delete**.

   The constraint is deleted.

Running an Optimetrics Analysis

Once you have created all necessary Optimetrics based analyses, you have several options for running the simulations.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To use the **Analyze All** command at the Project or design level to simulate the nominal problem and subsequently run all Optimetrics setups, do the following:
  1. In the Project Manager window, right-click on the **project** or **design** name.
  2. Click **Analyze All** from the shortcut menu.
- To use the **Analyze All** command from the Optimetrics menu to simulate only the Optimetrics based setups, do the following:
  1. In the Project Manager window, right-click on **Optimetrics**.
  2. Click **Analyze** > **All** from the shortcut menu.
- You can choose to analyze only the setups related to a specific Optimetrics type of analysis. In order to simulate setups of a specific type, do the following:
  1. In the Project Manager window, right-click on **Optimetrics**.
  2. Click **Analyze** > **All{*TYPE*}** from the shortcut menu, where *TYPE* is the specific analysis type of interest, Parametric, Optimization, Sensitivity, or Statistical.

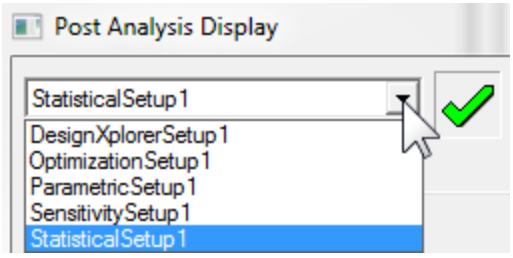
Viewing Analysis Results for Optimetrics Solutions

To view data specific to an Optimetrics solution:

1. In the project tree, right-click the Optimetrics setup for which you want to view the results, and select **View Analysis Result**.

   The **Post Analysis Display** dialog box appears.

2. Use the drop-down menu to select from available setups.



3. Click the **Results** tab to view results in plot or table form.

   When you view results in Table format, you can sort the results based on each column. Click a column's header to sort. Click again to invert the current sort.

4. Click **Options...** to open a dialog box that allows you to specify the Maximum number of significant digits to display when showing the analysis result. The default is 4.
5. Select the **Profile** tab to view start, stop, and elapsed times for each variable, and the analysis machine for each variation. Click a column heading to sort the table by variation number, variable value, start, stop, elapsed time, or machine.

See the help topics in this section for more details about viewing Optimetrics analysis results.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Viewing Solution Data for an Optimetrics Design Variation

To view the convergence information, or computing resources used, or matrices computed for any design variation solved during an optimization analysis, you must first select the design variation in the **Set Design Variation** dialog box. This dialog box is accessible from the **Solutions** window and via the **Results** > **Apply Solved Variation** command.

1. Click the **Maxwell** menu and then select **Results** > **Solution Data**.

   The **Solutions** dialog box appears.

2. Click the browsing dots beside the **Design Variation** box (...).

   The **Set Design Variation** dialog box appears.

3. Clear the **Use nominal design** option.
4. Click the design variation for which you want to view the solution data, and then click **OK**.

   The solution data is displayed in the table.

Viewing an Optimetrics Solution's Profile Data

At any time during or after the Optimetrics solution process, you can see an overview of the computing resources or profile data that was used by Maxwell as it solved each design variation. Optimetrics writes the variation information to the profile table before the solve. It then updates the entry with end data (end time, elapsed time, etc.) once the solve variation is completed.

1. In the project tree, right-click the Optimetrics solution setup of interest, and select **View Analysis Result**.

   The **Post Analysis Display** dialog box appears.

2. Click the **Profile** tab.
3. Select the Optimetrics setup with the results you want to view from the drop-down menu at the top of the dialog box.
4. Optionally, to examine more detailed profile data for a specific design variation, do the following:
   a. Click a design variation in the table.
   b. Click **Solver Profile**.

   The **Solutions** dialog box appears with the profile data for the selected design variation.

   The profile line for the matrix solver is in the following format:

   Solver *123*

   where:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- *1* is the precision type: M (mixed) or D (double)
- *2* is the matrix data type: R (real) or C (complex)
- *3* is the symmetry type: S (symmetric), A (asymmetric), H (hermitian)

Viewing Cost Results for an Optimization Analysis

To view cost values versus completed iterations in data table format:

1. In the project tree, right-click the optimization setup for which you want to view the cost results, and select **View Analysis Result**.

   The **Post Analysis Display** dialog box appears.

2. Under the **Result** tab, select **Table** as the view type, if it is not already selected.

   The cost value at each solved design variation is listed in table format.

3. Optionally, click a design variation in the table, and then click **Apply**.

   The software now points to the selected design variation as the nominal solution and as a result, the design displayed in the **Modeler** window is changed to represent the selected design variation.

   Click **Revert** to return the design in the view window to the original value.

Plotting Cost Results for an Optimization Analysis

To view cost values versus completed iterations in rectangular (x-y) plot format:

1. In the project tree, right-click the optimization setup for which you want to view the cost results, and then click **View Analysis Result** on the shortcut menu.

   The **Post Analysis Display** dialog box appears.

2. Under the **Result** tab, select **Plot** as the view type.

   A plot of the cost value at each iteration appears.

Link to DesignXplorer

You can export a .xml file containing information on a Maxwell setup, optimization variables, and output variables that enables Ansys Design Xplorer to manage the simulations (for example, for design of experiments and optimization). Design Xplorer will launch Ansys Electronics Desktop simulations of design variations and evaluate the outputs.

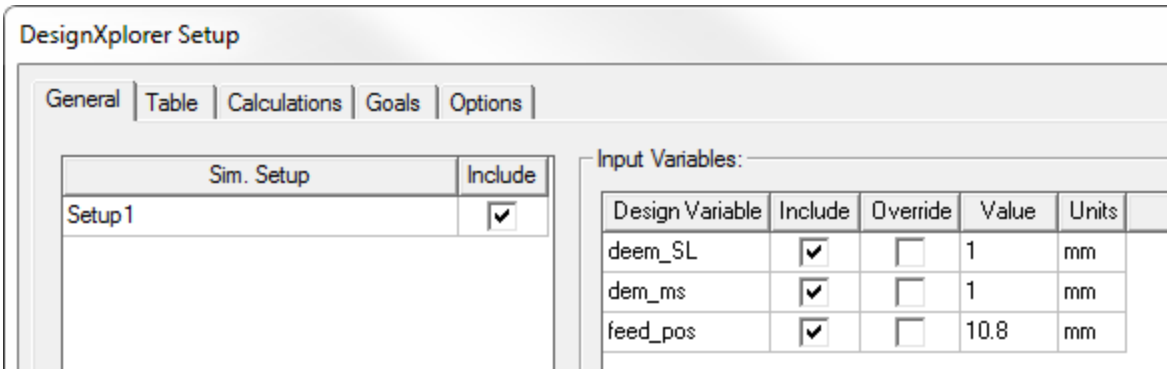
To do so:

1. Click your product on the menu bar and then **Optimetrics Analysis** > **Add Design Xplorer Setup** or right-click on **Optimetrics** in the **Project Manager** window, and select

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Add Design Xplorer Setup from the short-cut menu.

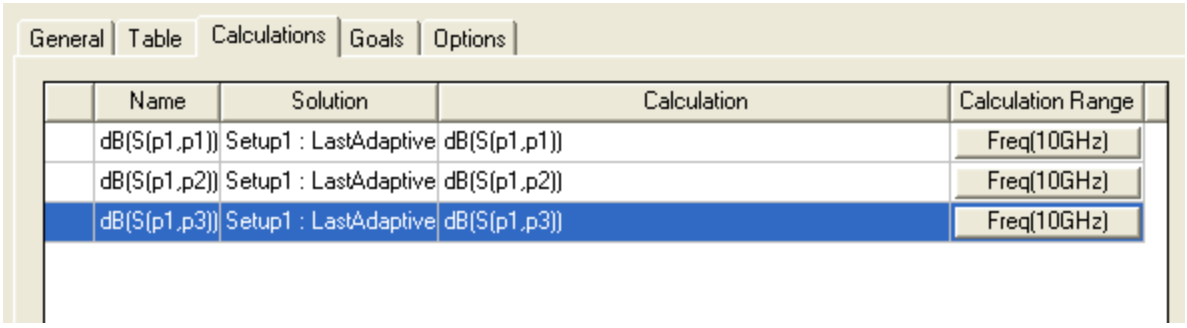
This opens the **Design Xplorer** dialog box with the **General** tab selected. It lists the setups available in the current project, and the input variables it contains.



2. Select **Include** for the simulation setups you want to use.
3. Check the Design variables to use. You can also chose to Override the value of a design variable. You can edit the Value and Units fields. Unchecking **Override** returns the values to their original state.
4. To set up any output calculations, click the **Calculation** tab and click **Setup Calculations**.

   This opens the **Add/Edit Calculation** dialog box. Here you can define the simulation results of interest. The dialog box contains distinct panes and tabs to set the **Context**, the **Calculation Expression**, and the **Calculation Range**. See: Setup Calculations for Optimetrics for details.

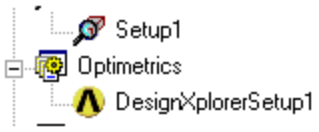
   Use the **Add Calculation** button to add expressions to the Calculations table.



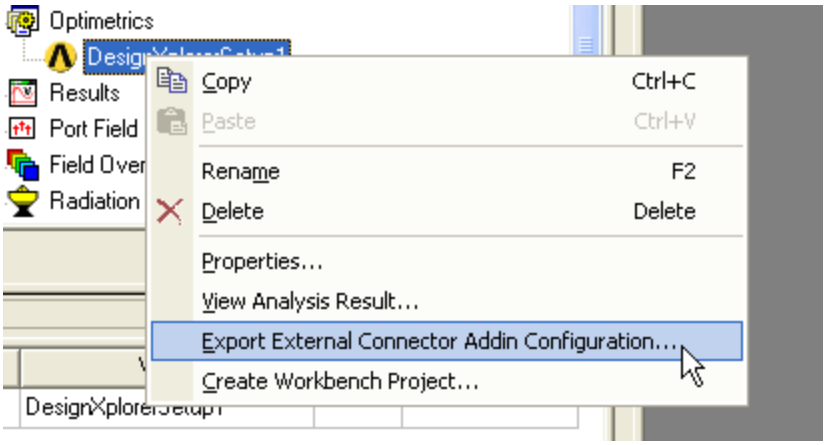
5. When you have added the calculations of interest, click **OK** to save the setup.

   An icon for the Design Xplorer setup appears under Optimization in the Project tree.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. To create a .xml file with the setup information for Design Xplorer, first Save your project.
7. Then right-click the setup and select **Export External Connector Addin Configuration**.



This displays a browser dialog that you can use to navigate your file system and name and saves the .xml file. This file contains information regarding the path along with the setup, variables, and simulation results that you specified.

8. If you have an Ansys Workbench installation, you can perform additional steps. You should have provided a path to the Workbench installation in the **Tools** > **General Options** dialog box Miscellaneous tab, to provide a path.
9. Then click **Create Workbench Project**.

   This lets you name a Workbench project containing the information in the setup. The Ansys Workbench will be launched with the connection to the project established. To this connection, you can add a Design Xplorer Setup. See the documentation of Ansys Workbench for details on Design Xplorer.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26 - Getting Started with RMxprt

RMxprt™ (Rotating Machines expert) is a template-based electrical machine design tool that provides fast, analytical calculations of machine performance and 2-D and 3-D geometry creation for detailed finite element calculations in Ansys Maxwell®.

Using RMxprt, you can simulate and analyze the following types of machines:

- Three-phase and single-phase induction motors
- Three-phase synchronous machines
- Brushless permanent-magnet DC motors
- Adjust-speed synchronous motors and generators
- Permanent-magnet DC motors
- Switched reluctance motors
- Line-start permanent-magnet synchronous motors
- Universal motors
- General DC machines
- Claw-pole alternators
- Three-Phase Non-Salient Synchronous Machine
- Generic Rotating Machine

When you start to design a new model in RMxprt, you first select one of the above motor or generator types. You then enter the parameters associated with that machine type in each RMxprt Properties window. The properties windows are accessed by clicking each of the machine elements (for example, stator, rotor, shaft) under **Machine** in the Project Manager tree. General options are available directly at the **Machine** level of the Project Manager tree. Solution and output options (such as the rated output power) are set when you add a solution setup (by right-clicking **Analysis** in the Project Manager tree).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Getting Started with RMxprt

RMxprt™ (Rotating Machines expert) is a template-based electrical machine design tool that provides fast, analytical calculations of machine performance and 2-D and 3-D geometry creation for detailed finite element calculations in Ansys Maxwell®.

Using RMxprt, you can simulate and analyze the following types of machines:

- Three-phase and single-phase induction motors
- Three-phase synchronous machines
- Brushless permanent-magnet DC motors
- Adjust-speed synchronous motors and generators
- Permanent-magnet DC motors
- Switched reluctance motors
- Line-start permanent-magnet synchronous motors
- Universal motors
- General DC machines
- Claw-pole alternators
- Three-Phase Non-Salient Synchronous Machine
- Generic Rotating Machine

When you start to design a new model in RMxprt, you first select one of the above motor or generator types. You then enter the parameters associated with that machine type in each RMxprt Properties window. The properties windows are accessed by clicking each of the machine elements (for example, stator, rotor, shaft) under **Machine** in the Project Manager tree. General options are available directly at the **Machine** level of the Project Manager tree. Solution and output options (such as the rated output power) are set when you add a solution setup (by right-clicking **Analysis** in the Project Manager tree).

Creating a Project and Inserting a New RMxprt Design

To create a new project:

1. Click **File** > **New**.

   A new project is listed in the Project Manager tree. It is named Project*n* by default, where *n* is the order in which the project was added to the current session.

   Project definitions, such as material assignments, are stored under the project name in the project tree.

2. Click **Project** > **Insert RMxprt Design** or select **RMxprt** from the **Maxwell** drop-down menu on the **Desktop** ribbon.

   The **Select Machine Type** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select the machine type you want, and click **OK**.
4. Specify the name of the project when you save it using the **File** > **Save** or **File** > **Save As** commands.

> **Note:** Because RMxprt is part of the Ansys Electronic Desktop software suite, projects are managed (opened, saved, *etc.*) as they would be for any other Desktop solver. For more information on managing projects, see "Managing Projects and Designs" on page 3-98.

RMxprt Files

When you create any design in the Ansys Electronics Desktop, including an RMxprt design, it is given an **.aedt** file extension and stored in the directory you specify. Any files related to that project are also stored in that directory.

Some common file and folder types are listed below:

.aedt	Maxwell or RMxprt project.
*project_name.***aedtresults**	Folder containing results data for a project.
*design_name.***results**	Folder containing results data for a design. This folder is stored in the *project_name.***aedtresults folder.**
*design_name.***asol**	Results data for a design. This file's contents may be empty if a solution is unavailable. This file is stored in the *project_name.***aedtresults** folder.

Saving Project Notes in RMxprt

You can save notes about a project, such as its creation date and a description of the device being modeled. This feature is useful for keeping a running log on the project.

To add notes to a project:

1. Click **RMxprt** > **Edit Notes**.

   The **Design Notes** dialog box appears.

2. Click in the window and type your notes.
3. Click **OK** to save the notes with the current project.

To edit existing project notes:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Double-click the **Notes** icon in the project tree.
   The **Design Notes** window appears, where you can edit the project's notes.
2. Click **OK** to save any changes, or click **Cancel** to exit without saving edits.

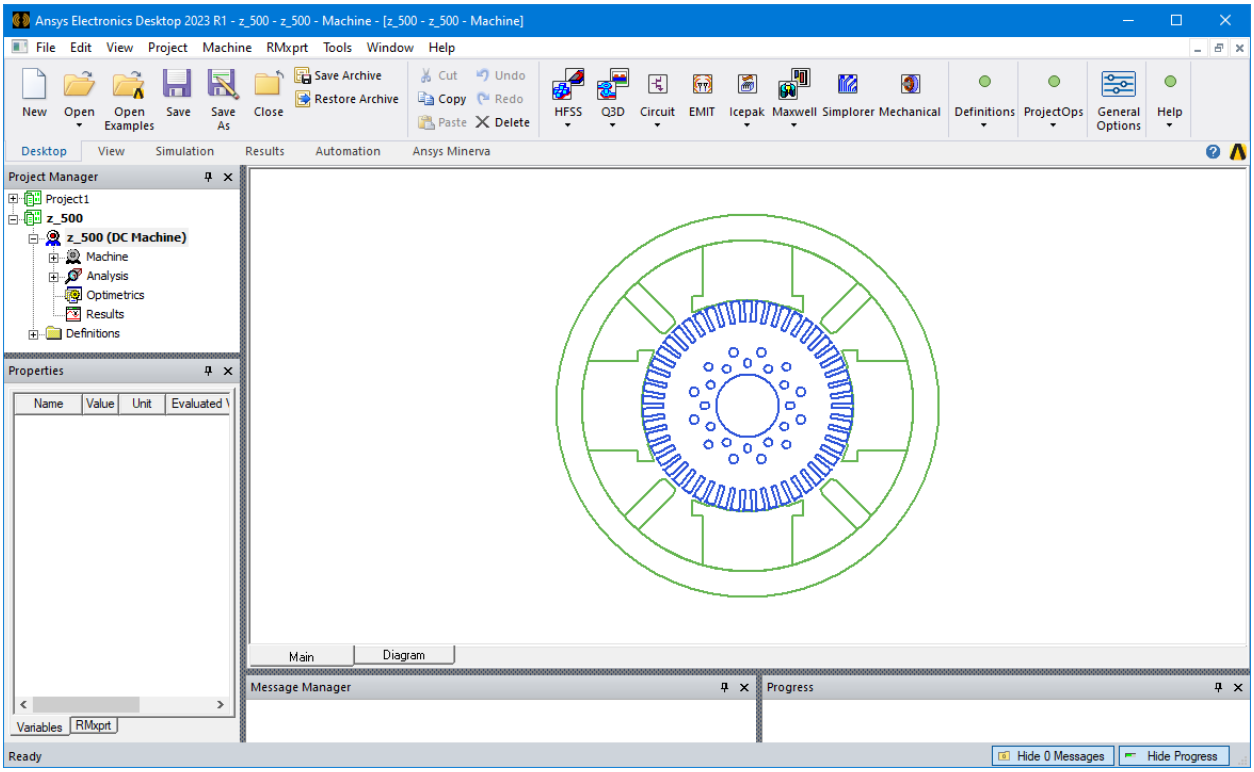
The RMxprt Desktop

RMxprt is integrated within the Maxwell desktop. Consistent with the Maxwell desktop, the RMxprt interface consists of several desktop components: a title bar, a menu bar, ribbons, a status bar, a project manager window, a properties window, a message manager window, and a progress window. The Project Manager window, the properties window, the message manager window and the progress window are dockable and resizeable.

The machine editor window is unique to RMxprt, and it will be detailed in this section. If user-defined rotor or stator slots are used in the design, a slot editor window also displays when a rotor or stator slot is selected in the project tree.

You can open multiple machine editor windows to display different parts at the same time. One can remain fixed on the winding, one on the diagram, and one on the main desktop window. To open a new window, click **Window** > **New Window**.

To move back and forth between windows, select the window you want to view from the **Windows**.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RMxprt Title Bar

The title bar is located at the top of the application window. It displays the information of the active design. If a machine editor window is maximized, its title is appended in the title bar within square brackets. The information of the active design includes the desktop name, the project name, the design name and the design type. For an RMxprt design, the design type is **Machine**.

Working with the RMxprt Menu Bar

The menu bar enables you to perform all Maxwell, ePhysics, and/or RMxprt tasks, depending on the software you purchased. Such tasks include managing project files, customizing the desktop, drawing objects, and setting and modifying all project parameters.

RMxprt contains the following menus, which appear at the top of the desktop:

File	Use the **File** menu commands to manage RMxprt project files and printing options.
Edit	Use the **Edit** menu commands to modify properties in the active design, manage designs in one or more projects, delete projects, and undo and redo actions.
View	Use the **View** menu commands to display or hide desktop components, and change the machine editor window view.
Project	Use the **Project** menu commands to add a Maxwell 3D, Maxwell 2D, or RMxprt design to the active project, analyze all designs of the active project, and define project variables and datasets.
Machine	Use the **Machine** menu to work with the machine data, such as edit winding layout, edit wire size, and set dimension unit for the active editor window.
RMxprt menu	Use the **RMxprt** menu commands to validate design input data, analyze designs, set up parameters, add analysis setups, set up Optimetrics, postprocess solutions, and other design tasks.
Tools	Use the **Tools** menu to modify the active project's material library, arrange the material libraries, run and record scripts, update project definitions from libraries, display options, and modify many of the software's default settings.
Window	Use the **Window** menu commands to rearrange the application windows.
Help	Use the **Help** menu commands to access the help system and view the current software version information.

Working with the RMxprt Shortcut Menus

A variety of shortcut menus — menus that appear when you right-click a selection — are available in the ribbons area of the desktop, in the **Machine Editor** window, in the **Project Manager** window, in the **Properties** window, and in the **Message Manager** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In Machine Editor window	Use the shortcut menu in the **Machine Editor** window to edit winding layout, display or hidden coil connection, change the view, and copy to Clipboard.
In the Slot Editor window	Use the shortcut menu in the **Slot Editor** window to insert, append, modify, and remove slot segments.
In the Project Manager window	Use the shortcut menus in the **Project Manager** window (or the project tree) to manage project files and design properties; these commands duplicate menu commands at the top of the screen.
In Properties window	Use the shortcut menus in the **Properties** window to edit (cut, copy, paste or delete) property values.
In Message Manager window	Use the shortcut menus in the **Message Manager** window to clear, copy message, or see message details.

> **Note:** Most of the commands on the shortcut menus are also available on the menu bar.

Working with the RMxprt Machine Editor Windows

You can open multiple machine editor windows in RMxprt. One can remain fixed on the **Winding Editor**, one on the **Diagram** tab, and one on the **Main** tab.

To open a new window, click **Window** > **New Window**.

To move back and forth between windows, select the **Windows** menu, and select the window you want to view.

You can cascade all **Machine Editor** windows, tile them horizontally or vertically. You can maximize, minimize or close a **Machine Editor** window by clicking the relevant button on the right-top corner of the window.

If no **Machine Editor** window is displayed, you can use **RMxprt** > **Machine Editor** to bring one window up.

When only one **Machine Editor** window is maximized, the window title is displayed within square brackets in the **Title Bar** of the main application window.

As you enter appropriate property values, the **Machine Editor** window dynamically updates the rotor, stator, slots, and windings in the **Main**, **Diagram** and **Winding Editor** tabs. As you provide winding information, the **Winding Editor** tab displays a table of values.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the RMxrpt Window View

To fit the entire diagram in the window, select **View** > **Fit All**.

To zoom into the diagram in the window, select **View** > **Zoom In**.

To zoom out of the diagram in the window, select **View** > **Zoom Out**.

Printing in RMxprt

The printing commands enable you to print the display in the active window.

To print the project:

1. Click **File** > **Print**.

   The **Print** dialog box appears.

2. You can change the print quality (a higher dpi produces a higher quality print but takes more time and printer memory), or you can send the output to a **.prn** file.
3. Do one of the following:
   - Click **OK** to print the project.
   - Click **Cancel** to dismiss the window without printing.
   - Click **Properties** to define printer settings.

Viewing RMxprt Design Details

Once you insert an RMxprt design into a project, it is listed as the second-level node in the Project Manager tree. It is named RMxprtDesign*n* by default, where *n* is the order in which the design was added to the project. Expand the design icon in the project tree to view specific data about the model.

The RMxprtDesign*n* node contains the following project details:

Machine	Allows you to specify parameters for various aspects of the machine. A whole or part geometry will be drawn in the **Main** tab of the **Machine Editor** window (based on the values you enter).
Analysis	Displays the solution setups for an RMxprt design. A solution setup specifies how RMxprt computes the solution.
Optimetrics	Displays any Optimetrics setups added to an RMxprt design.
Results	Displays any post-processing reports that have been generated.

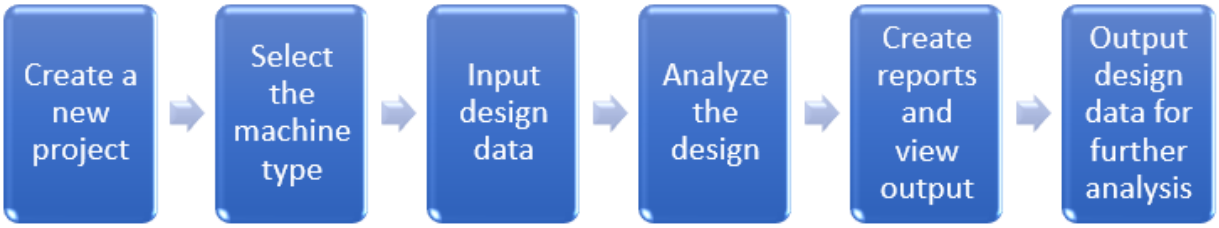
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** To edit a project's design details, in the Project Manager tree, double-click the design setup icon that you want to edit. A dialog box appears with that setup's parameters, which you can then edit.

Quick Start for RMxprt

This section briefly introduces how to enter the environment of **RMxprt** and quickly master its main functions by providing a simple example.

The basic process flow chart is shown below.



RMxprt Example Part 1: Create a New Project

To open Ansys Electronics Desktop and create a new project:

1. Double-click the **Ansys Electronics Desktop** icon on your desktop to launch Ansys Electronics Desktop.

   You can also start Ansys Electronics Desktop by clicking **Start** > **All Programs** > **Ansys EM Suite** [version] >**Ansys Electronics Desktop** [version] from Windows.

2. Click **File** > **New** from the menu bar.

   This creates a new project folder in the project window with the default name of Project*n.*

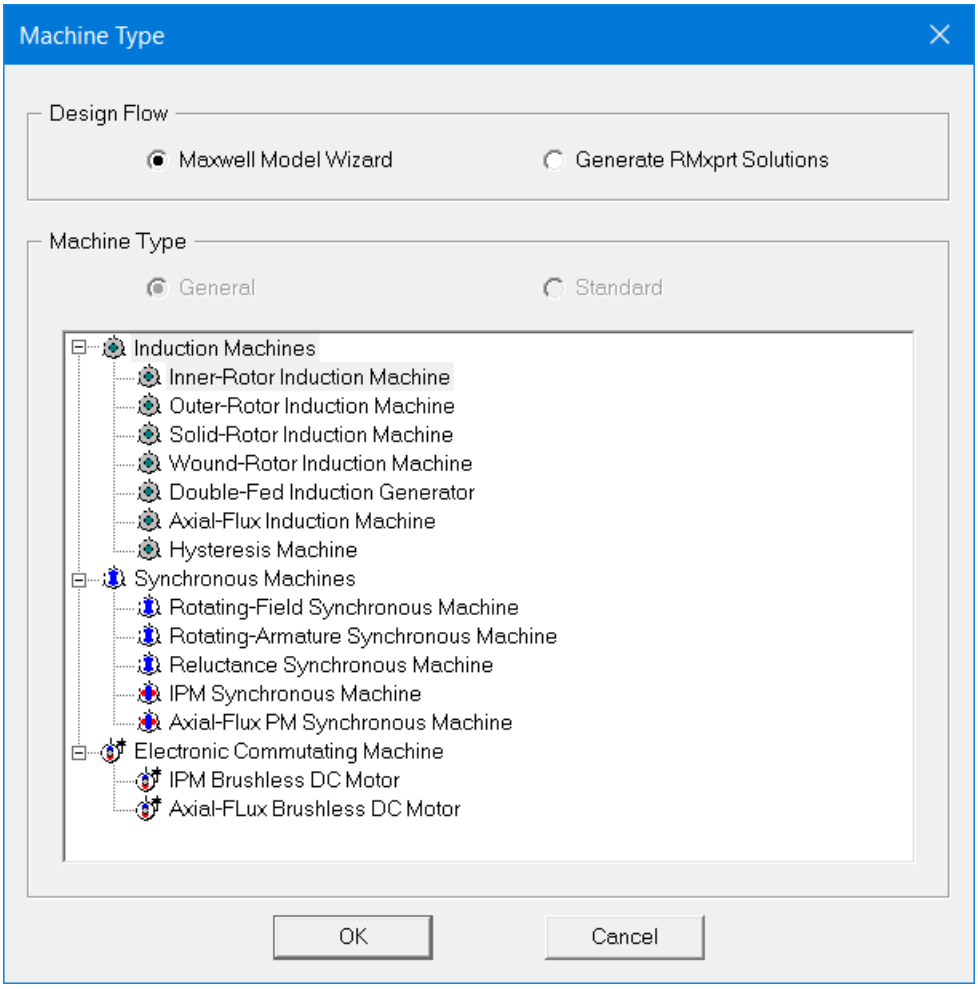
RMxprt Example Part 2: Select a Machine

To select a machine to insert into the new project:

1. Click **Project** > **Insert RMxprt Design** or select the **RMxprt** icon on the Desktop ribbon.

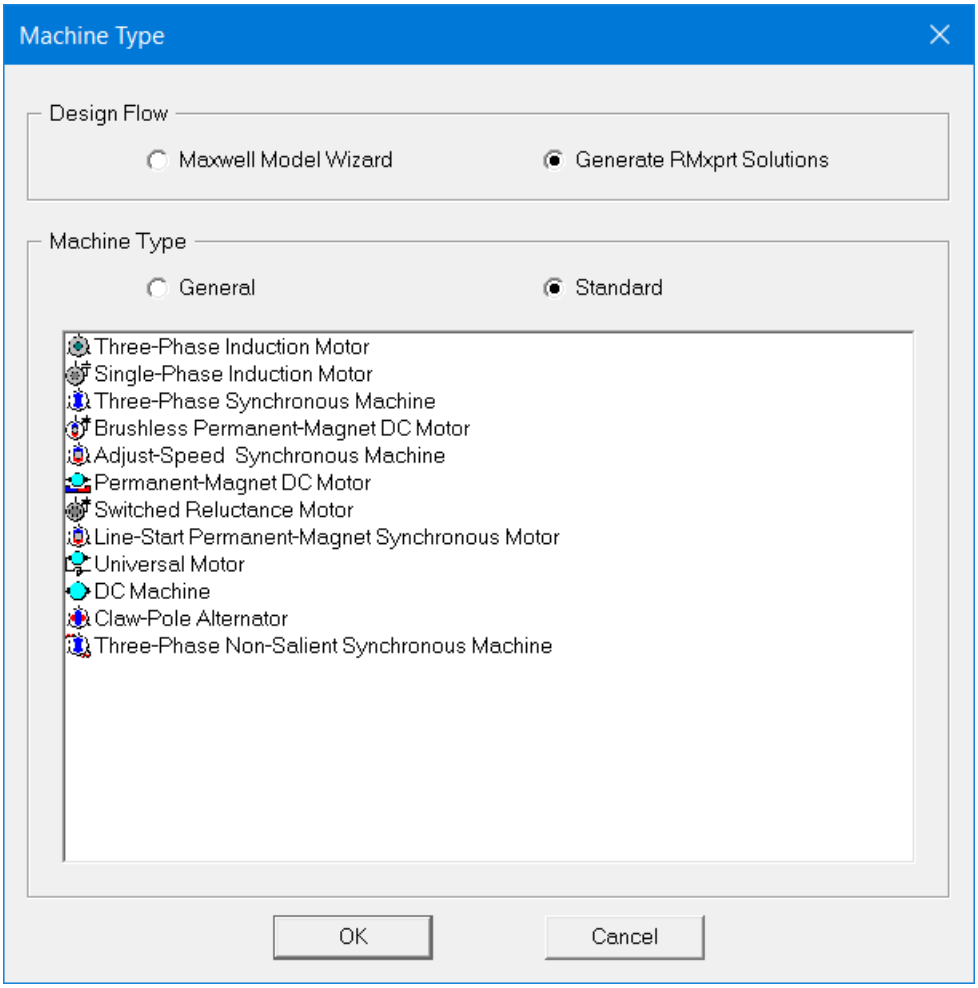
   This displays the **Machine Type** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. For the Design Flow, select the **Generate RMxprt Solutions** radio button; then select **Standard** for Machine Type.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. From the list of machine types, for this example, select **Brushless Permanent Magnet DC Motor** and click **OK**.

   This closes the window and inserts the Brushless Permanent Magnet DC Motor design in the project.

Continue to Part 3 of the example to Input Design Data.

RMxprt Example Part 3: Input Design Data

In this part of the example, you provide values for the design and for various parts.

1. Click the **+** button by the RMxprt:Design*n* icon in the project tree to view the design hierarchy.

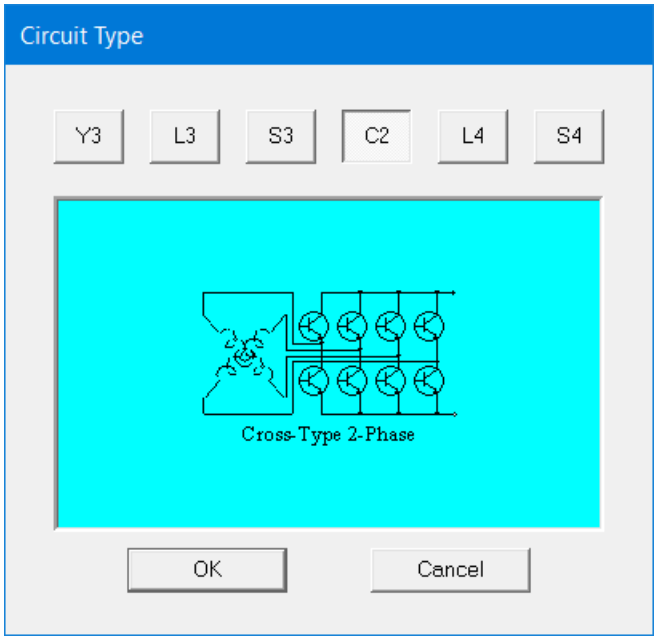
   This displays the Machine icon.

2. Double-click the icon to view the Machine Properties window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Set the values as indicated below.

Machine Type	Brushless Permanent Magnet DC Motor
Number of Poles	Set this to 4
Rotor Position	Set to Inner Rotor
Frictional Loss	Set this to 11 Watts (Frictional and windage loss is typically within the range of 1%~3% of the rated output power, in this example, 2% is estimated.) This value is referred to the given **Reference Speed**. The frictional loss at the computed rated speed will be modified if the computed rated speed is different from the given rated speed.
Windage Loss	0
Reference Speed	Set this to 1500 rpm
Control Type	DC
Circuit Type	Click the **C2** button to select the Cross-Type 2-Phase circuit.



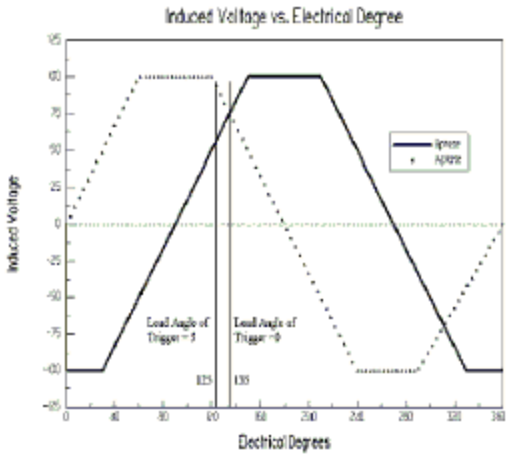
3. Click **OK** to close the Circuit Type dialog box.
4. Click **OK** to close the Machine properties window.
5. Click the **+** button by the Machine icon to view the design hierarchy of the motor.
6. Double-click the Circuit icon to view the Circuit properties window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Set the values as indicated below.

Lead Angle of Trigger

Set this to 0 degrees to obtain the maximum average emf for the following phase in the trigonal period.



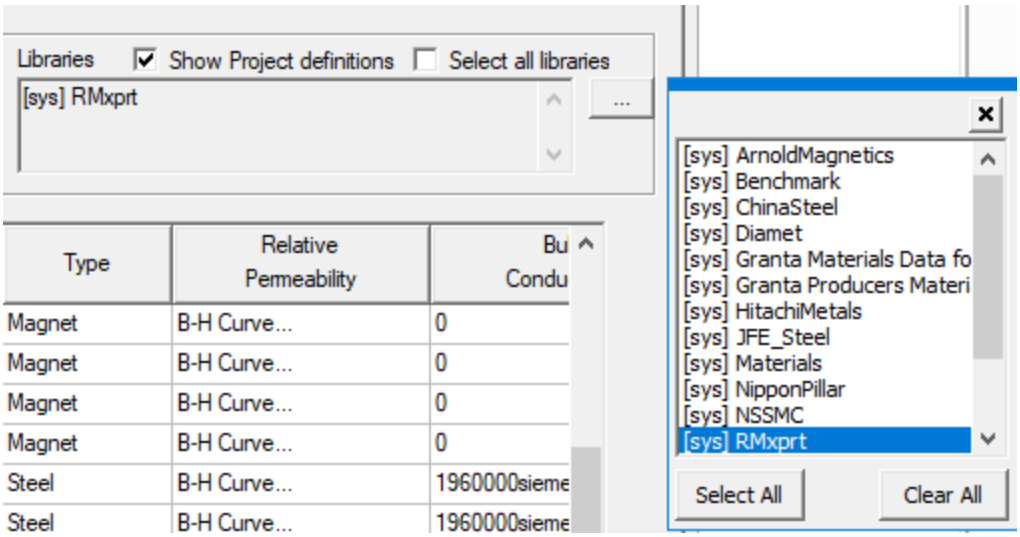
Trigger Pulse Width	Set this to 90 degrees
Transistor Drop	Set this to 2 Volts
Diode Drop	Set this to 2 Volts

7. Click **OK** to close the circuit properties window.
8. Double-click the Stator icon to view the Stator properties window.

   Set the values as shown below.

Outer Diameter	Set this to 120 mm.
Inner Diameter	Set this to 75 mm.
Length	Set this to 65 mm for the length of the Stator iron core.
Stacking Factor	0.95
Steel Type	Click on the button to display the **Materials** window. Select **RMxprt** library in the **Libraries** box in the upper right corner of the **Materials** window: then select **M19-24G** from the list.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



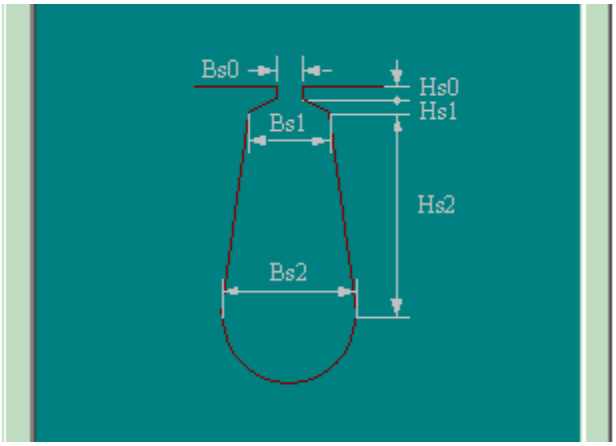
Note: If RMxprt is not listed in the libraries box in the upper right corner of the Materials window, quit the Materials window, click **Tools** > **Configure Libraries**, add **RMxprt** (under materials) and click the **Save as Default** check box. Then click OK.

Number of Slots

Set this to 24.

Slot Type

Select **2** as the Slot type. Click the button on the row cell to display the Select Slot Type window.



Click **OK** to close the window.

Skew Width

Set this to 1. (To skew one slot pitch.)

9. Click **OK** to close the **Stator Properties** window.

   Take a moment to look at the Maxwell Design window. If you click the **Main** tab, you will see two concentric rings that represent the inner and outer diameters you specified. If you

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

click the **Winding Editor** tab, you see a table of the coils, with columns for **Phase**, **Turns**, the **In Slots**, and the **Out Slots**. There is also a drawing showing the placement of the 24 slots of the type that you defined here.

10. Click the **+** button by the **Stator** icon to view the hierarchy under the stator.
11. Click the slot icon to view the **Slot Properties** window.

    Set the values as shown below. Some of the properties will not appear until you disable the **Auto Design** property in the first row.

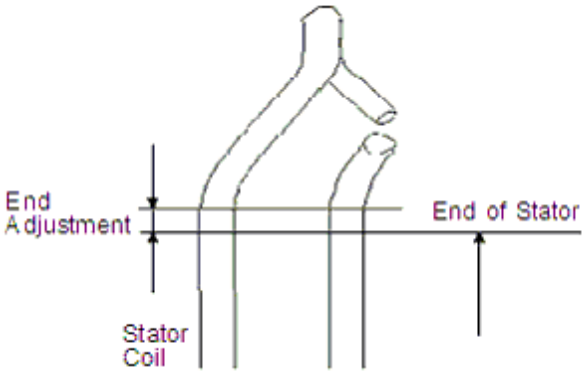
Auto Design	Uncheck the box to disable auto design. Close the properties window and open it again. Then set the given values for the slot shapes.
Parallel Tooth	Uncheck this box. The Tooth Width property becomes invisible.
Tooth Width	Invisible when Parallel Tooth is unchecked.
Hs0	Set to 0.5 mm
Hs1	Set to 1.0 mm
Hs2	Set to 8.2 mm
Bs0	Set to 2.5 mm
Bs1	Set to 5.6 mm
Bs2	Set to 7.6 mm

12. Click the stator **Winding** icon to view the winding **Properties** window.

    Set the values as shown below.

Winding tab	**Winding Layers**	Set this to 2.
	Winding Type	"Whole-Coiled."
	Parallel Branches	Enter 1 for the number of parallel-connected branches, i.e. the coils in all the slots per phase are in series-connected.
	Conductors per Slot	Set this to 60 for the number of conductors per slot, i.e. the number of turns per coil is equal to 30 for double-layer winding.
	Coil Pitch	Set this to 5. For this example, full pitch = 24 slots / 4 poles = 6. This example uses short coil pitch, *5*, i.e. a coil spans from slot 1 to slot 6.
	Number of Strands	Select *1* for the number of strands (or number of wires per conductor).
	Wire Wrap	Select *0*. This is the total thickness of double side wire insulation. The input value *0* means that **RMxprt** will automatically check into the wire gauge library for the wrap thickness relevant to the wire

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		gauge. Different manufacturers produce different **Wire Wrap Thickness** for electromagnetic wire. Typically, **Wire Wrap Thickness** for electromagnetic wire is 7~10% of **Wire Diameter**.
	Wire Size	Click on the Properties field to display the Wire Size window and select *AUTO* for automatic design of wire **Gauge**. Wire Size will be set to 0 in the Wire Size window. This example relies on **RMxprt** to automatically select the optimum diameter and the gauge code for electromagnetic wire. Close the window.
End/Insulation tab	**Input Half-turn Length**	Uncheck this box.
	Half Turn Length	This item is not shown if **Input Half Turn Length** is unchecked.
		Set this to 0 mm for the linear overhang of the end part of the coil out of the iron core as shown below. In this example, the coil turns immediately at the slot opening, therefore input 0**.**
	End Extension (End Adjustment)	
	Base Inner Radius	0
	Tip Inner Diameter	0
	End Clearance	0
	Slot Liner	Set this to 0.3 for the single side thickness of slot insulation.
	Wedge Thickness	0
	Layer Insulation	0
	Limited Fill Factor	0.75 (The slot fill factor is the ratio between the cross-

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

sectional area of all conductors in one slot and the entire slot area.)

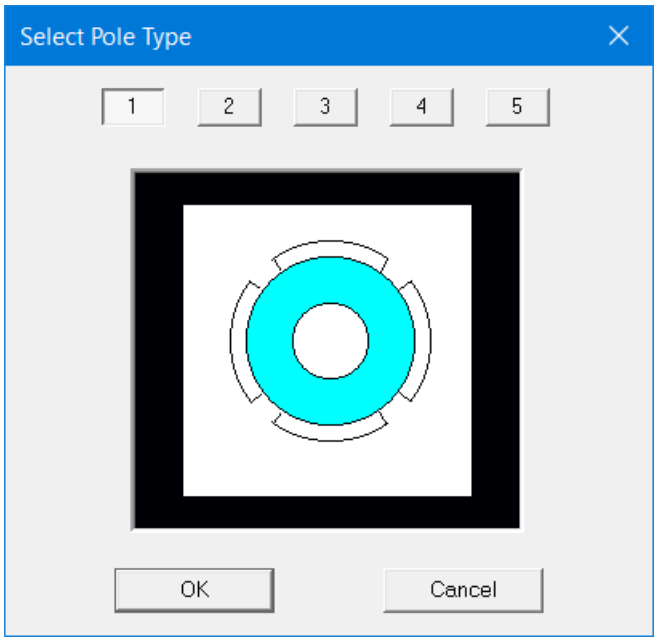
13. Click **OK** to close the stator Winding Properties window.
14. From the menu bar, select **Machine** > **Winding** > **Connect all coils**.

    The **Winding** tab in the main window shows all coils connected.

15. Double-click the Rotor icon to view the **Rotor Properties** window.

    Set the values as shown below.

Outer Diameter	Set this to 74.0 mm. This is the Stator inner diameter - 2* AirGap.
Inner Diameter	Input 26 mm for the inner diameter of the rotor core. This is also the diameter to match the shaft.
Length	Input 65 mm for the length of the rotor core. In this example, the lengths of the iron cores of the stator and the rotor are the same.
Steel Type	Select M19-24G for the material of the silicon-steel sheet for the rotor. In this example, the laminations are punched together on the same sheet; therefore, the material of the silicon-steel sheet and the stacking factors are the same for the stator and the rotor.
Stacking Factor	Input 0.95.
Pole Type	Select 1. Click on the button on the Pole Type field to display the Select Pole Type window.



Click **OK** to close the window.

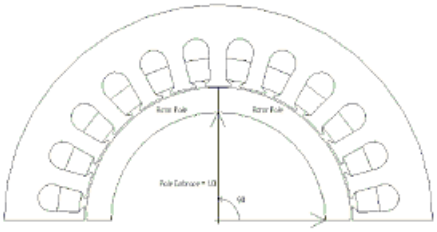
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

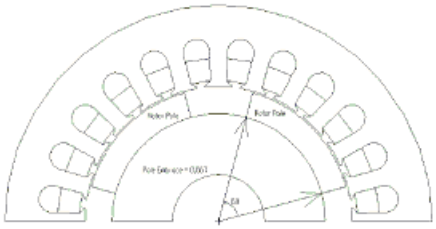
16. Click **OK** to close the Rotor Properties window.
17. Click the **+** button beside the Rotor icon to open the project hierarchy under the rotor.
18. Double-click the Pole icon to view the Pole Properties window.

    Set the values as shown below.

Embrace

Input 0.7. **Embrace** of the rotor represents the ratio of the rotor central angle corresponding to the arc length along the rotor surface of an arched permanent-magnetic piece to the rotor central angle corresponding to a rotor pole. In a four-pole machine with an **Embrace** value of 1, each arched permanent-magnetic piece covers 90 mechanical degrees along the rotor surface. Similarly, with an **Embrace** value of 0.667 each arched permanent-magnetic piece covers 60 mechanical degrees along the rotor surface, as shown in the figure.

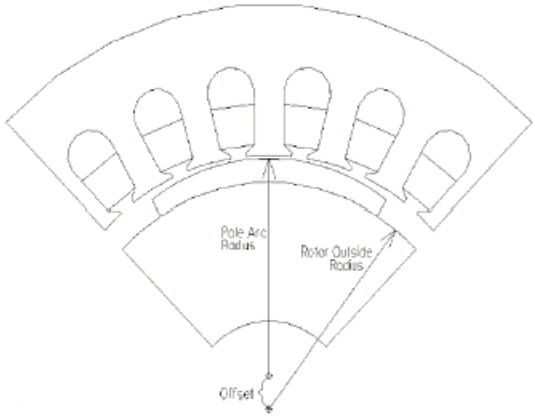




Offset

Input 0. The arched permanent-magnetic pieces to form the magnets of the rotor might not be concentric with the rotor as shown in the figure. In the electric machines with non-uniform air-gap, there exists an offset between the two centers. **RMxprt** terms it as **Pole Arc Offset**. This example uses uniform air-gap; therefore, the offset is set to *0*.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Magnet Type	Select XG196/96. This permanent-magnetic steel possesses residual flux density 0.96 Tesla, coercive force 690 kA/m, maximum magnetic energy product 183 kJ/m^3, and relative recoil magnetic permeability 1.0.
Magnet Thickness	Input 3.5 mm for the thickness of the permanent-magnetic steel.

19. Click **OK** to close the Pole Properties window.

To continue to Part 4 of the example, go to Analyze the Design.

RMxprt Example Part 4: Analyze the Design

Before analyzing a design project, a few options should be decided using the following procedures:

1. Click **Tools** > **Options** > **General Options** to open the **Options** window, displaying a hierarchical list of options. Click the **+** button next to **Machines** to show the following choice: **Wire Setting**.

   The System Libraries setting should be set to **American**.

2. Click **OK** to close the window.
3. Click **RMxprt** > **Analysis Setup** > **Add Solution Setup**.

   This displays the Solution Setup window. Add the following values.

Load Type	Const Power
Rated Output Power	550 W
Rated Voltage	220 V
Rated Speed	1500 rpm
Operating	75cel

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Temperature

4. Click **OK** to close the window and save the setup.
5. Click **RMxprt** > **Validation Check** to ensure that all values have been set.

   If any items do not pass validation, use the diagnostic information in the Message Window to resolve any issues.
6. When the design has been validated, click **RMxprt** > **Analyze All**.

The progress of the analysis is shown in the Progress window.

To continue to Part 5 of the example, go to Create Reports and View Output.

RMxprt Example Part 5: Create Reports and View Output

After you have run an analysis, you can view the solution data.

1. Click **RMxprt** > **Results** > **Solution Data**.

   This opens the **Solutions** window with the **Performance** tab selected, and the FEA Input Data displayed. The **Solutions** window contains tabs for the following:

   - **Performance** – the **Data** field in the **Performance** tab is a drop-down menu from which you can select the following:
     - FEA Input Data
     - Full Load Operation
     - Material Consumption
     - No Load Operation
     - Permanent Magnet
     - Rotor Data
     - Stator Slot
     - Stator Winding
     - Steady State Parameters
   - **Design Sheet** – Shows the various design parameter values
   - **Curves** – Selecting the **Curves** tab lets you view predefined graphs.
2. On the **Performance** tab, select **Stator Winding** as the **Data** selection.

   Except for some data corresponding to the wire gauge, this data should be the same as the data input in the Stator Winding Properties window. Since automatic design function for the wire gauge is selected in the input, **RMxprt** calculates the following data:

Wire Diameter	*0.8118* for the diameter of the electromagnetic wire.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(mm):	
Wire Wrap (mm):	*0* for the insulation thickness of the electromagnetic wire. Because input wire wrap is *0*, RMxprt picks it up from the selected wire library (American wire), but it still *0* based on the wire wrap data in the library.
Stator Slot Fill Factor (%):	*61.6599*

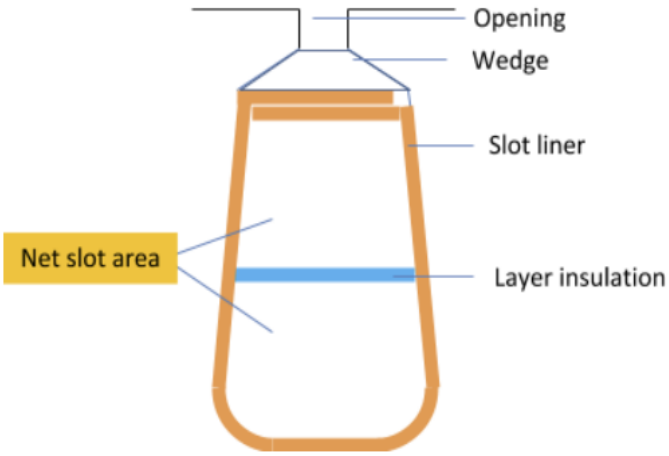
The electromagnetic wire with **Wire Diameter** of *0.8118* is equivalent to AWG 20. **Stator Slot Fill Factor** represents the percentage of occupation of the slot area, *i.e.* the ratio of the total square sectional area of wires (including Wire Wrap thickness) in a slot to the total slot area less the slot insulation.

The slot fill factor is defined as

$$F_{SF} = \frac{n \cdot d^2}{S_{net}}$$

where

- n is the total number of wires per slot (the number of conductors per slot times the number of wires per conductor),
- d^2 is the square area of a wire with d being the wire diameter including wire insulation, and
- S_{net} is the net slot area that is obtained from the gross slot area minus opening area, wedge area, liner area, and layer insulation area, as shown below:



3. Now that the **Wire Diameter** of the electromagnetic wire has been calculated by **RMxprt**, you can open the **Winding Properties** window and specify the value.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. For **Wire Size,** open the Wire Size selection window, select *0.8118* for the electromagnetic wire diameter, which corresponds to *20* for the wire gauge.
5. In the slot **Wire Wrap** field, input *0.08* for the insulation thickness of the electromagnetic wire.
6. Click **OK** to close the properties window.
7. Click **RMxprt** > **Analyze All**.

   After the second analysis is completed, click **RMxprt** > **Results** > **Solution Data** to view the effect of **Wire Wrap Thickness** of the electromagnetic wire on **Stator Slot Fill Factor**.

Wire Diameter (mm):	*0.8118*
Wire Wrap Thickness (mm):	*0.08*
Stator Slot Fill Factor (%):	*75.6182*

8. In the **Solutions** window, change the Data selection to **Rotor Data**.

   The Rotor data is displayed.

   Here most of the data is the same as input in the **Rotor Pole** properties window. The only difference is that the **Pole Arc Radius** replaces **Pole Arc Offset** and, in addition to **MechanicalPole Embrace** which is input based on the physical geometry, **Electrical Pole Embrace** is also given. **Electrical Pole Embrace** is calculated by the ratio of the average magnetic flux density to the maximum magnetic flux density according to the magnetic flux density distribution along the air-gap.

9. In the **Solutions** window, change the Data selection to **Permanent Magnet**.

   This part displays the characteristic data of the permanent magnets as well as the **Demagnetized Flux Density**, the **Recoil Residual Flux Density**, and the **Recoil Coercive Force** of the recoil line based on the demagnetization flux density, which are used for finite element analysis when a linear PM characteristics must be specified.

10. In the **Solutions** window, change the Data selection to **Steady State Parameters**.

    This part displays the **Stator Winding Factor**, direct- and the quadratic-axis inductances, the **Armature Leakage Inductance**, the **Armature Phase Resistance** of the phase winding, the direct- and the quadratic-axis time constants, the ideal **Start Torque Constant** K_T and the **Ideal Back EMF Constant** K_E.

11. In the **Solutions** window, change the Data selection to **No-Load Operation**.

    This part displays the magnetic flux densities in the teeth and the yoke of the stator, and the yoke of the rotor. The maximum value among the three magnetic flux densities is *1.59* Tesla, which locates at the knee part of the B-H curve, below the saturation situation.

    The mmfs of the teeth and the yoke of the stator, the air-gap, the yoke and the permanent magnet of the rotor are given respectively for half magnetic reluctance path.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The armature reaction mmf due to the armature current is referred to the demagnetization mmf. The magnetic flux leakage coefficient takes into account the part of the magnetic flux in the rotor not linking with the stator. The correction factors for the yoke lengths of the stator and the rotor to calculate the yoke mmfs of the stator and the rotor are also given here.

The no-load revolution speed of this machine is approximately *1946* rpm.

12. In the Solutions window, change the Data selection to **Full Load Operation**.

    At **Rated Output Power (kW):***0.550*, the following characteristic parameters of the machine are calculated as:

Parameters	**Calculated Values**	**Units**
Average Input Current (of input current waveform in one voltage period)	*2.93*	A
RMS Armature Current (of phase current waveform in one voltage period)	*2.45*	A
Armature Thermal Load (product of **Specific Electric Loading** and **Armature Current Density**	*70.88*	A^2/mm^3
Specific Electric Load (stator current distribution per circumferential length along air-gap)	*14.71*	A/mm
Armature Current Density (through cross-sectional area of stator wire)	*4.65*	A/mm^2
Frictional and Wind Loss (at computed **Rated Speed**)	*11.42*	W
Iron-Core Loss (due to loss curves of stator and rotor iron-core materials)	*20.02*	W
Armature Copper Loss (stator winding ohmic loss)	*47.92*	W
Transistor Loss (transistor switching loss)	*9.14*	W
Diode Loss (diode power consumption)	*0.60*	W
Total Loss (sum of above losses)	*89.1*	W
Output Power (the rated operating point is derived based on **Output Power**)	*550*	W

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Input Power (product of **Rated Voltage** and **Average Input Current**)	*639.2*	W
Efficiency (ratio of **Output Power** to **Input Power**)	*86.1*	%
Rated Speed (at **Rated Output Power**)	*1557*	rpm
Rated Torque (at **Rated Output Power**)	*3.37*	Nm
Locked-Rotor Torque (starting torque at zero revolution speed)	*32.3*	Nm
Locked-Rotor Current (starting current at zero revolution speed)	*52.1*	A

13. In the **Solutions** window, select the **Design Sheet** tab, and scroll down to **Winding Arrangement**.

    This is the layout and the arrangement of the whole two-phase winding of phases A and B, and the short coil pitch factor *5* is taken into account. The 2-phase, 2-layer winding can be arranged in 6 slots as below:

    AAABBB

Angle per slot (elec. degrees):	30
Phase-A axis (elec. degrees):	105
First slot center (elec. degrees):	0

14. In the **Solutions** window with the Design Sheet tab selected, scroll down to Transient FEA Input Data. (This is at the very bottom.)

    The following data of the armature winding corresponds to one phase armature winding.

Number of Turns (total number of turns viewed into output terminals)	*360*	
Parallel Branches	*1*	
Terminal Resistance (stator winding dc resistance under given operating temperature, 75^{o}C)	*4.13*	Ohm
End Leakage Inductance (of stator winding)	*1.3*	mH

    The following data is the equivalent values used to 2D electromagnetic field analyses.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equivalent Model Depth	*65*	mm
Equivalent Stator Stacking Factor	*0.95*	
Equivalent Rotor Stacking Factor	*0.95*	
Equivalent B_r (residual flux density)	*0.87*	Tesla
Equivalent H_c (coercive force)	*690*	kA/m

15. In the **Solutions** window, click the **Curves** tab.

    This displays the Input DC Current Versus Speed graph. If the text is too small to read, you can resize the window. You can view other predefined graphs by selecting from the drop down menu in the Name field.

    Selecting the Curves tab lets you view predefined graphs for the following relations:

    - Input DC Current vs Speed
    - Efficiency vs Speed
    - Ratio of air-gap torque to DC current vs Speed
    - Output Power vs Speed
    - Output Torque vs Speed
    - Cogging Torque in Two Teeth
    - Induced Coil Voltages at Rated Speed
    - Air-Gap Flux Density
    - Induced Winding Phase Voltage at Rated Speed
    - Winding Currents Under Load
    - Phase Voltage Under Load

16. You can also create additional plots with multiple curves. For example, click **RMxprt** > **Results** > **Create RMxprt Report** > **Rectangular Plot**.

    This displays the **Report** dialog box.

17. In the **Traces** tab, for the **Current** category, select **InputDCCurrent**, and click the **New Report** button to generate the report.
18. Then for the **Torque** category select **Output Torque**, and click **Add Trace** to add the trace to the report. Click **Close** to close the **Report** window and display the combined graph.

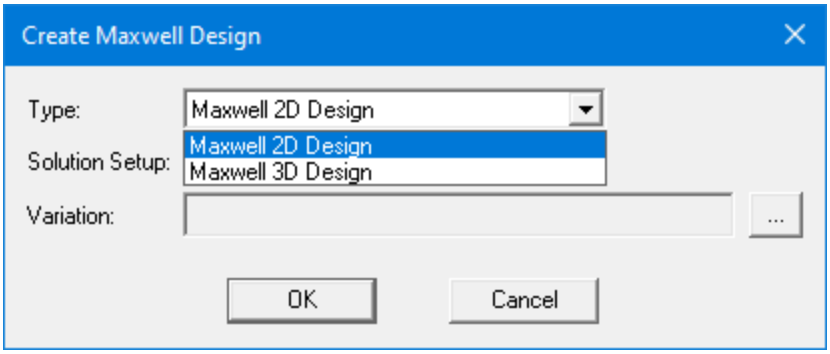
Continue to Part 6: Output Design Data.

RMxprt Example Part 6: Output Design Data for Further Analysis

To analyze the model further:

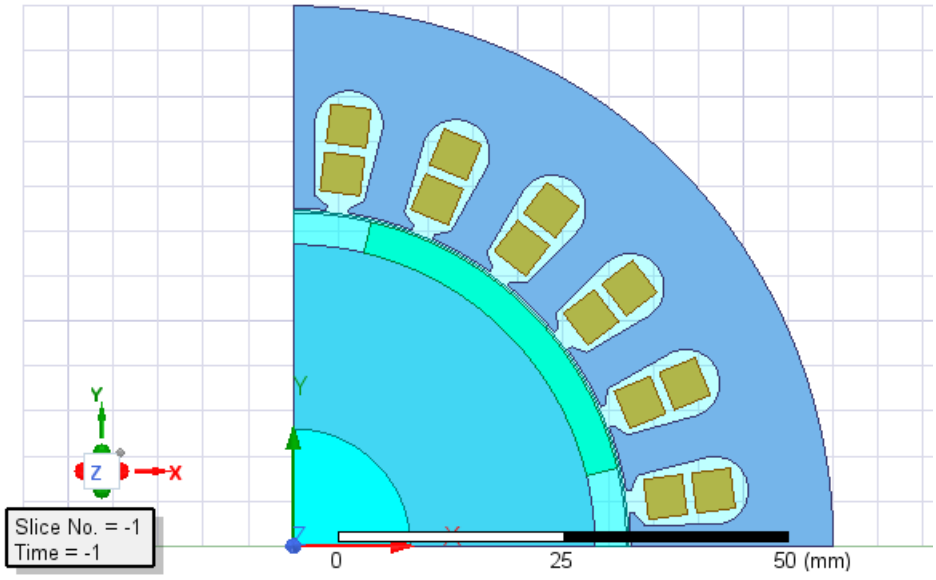
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select **RMxprt** > **Analysis Setup** > **Create Maxwell Design**.

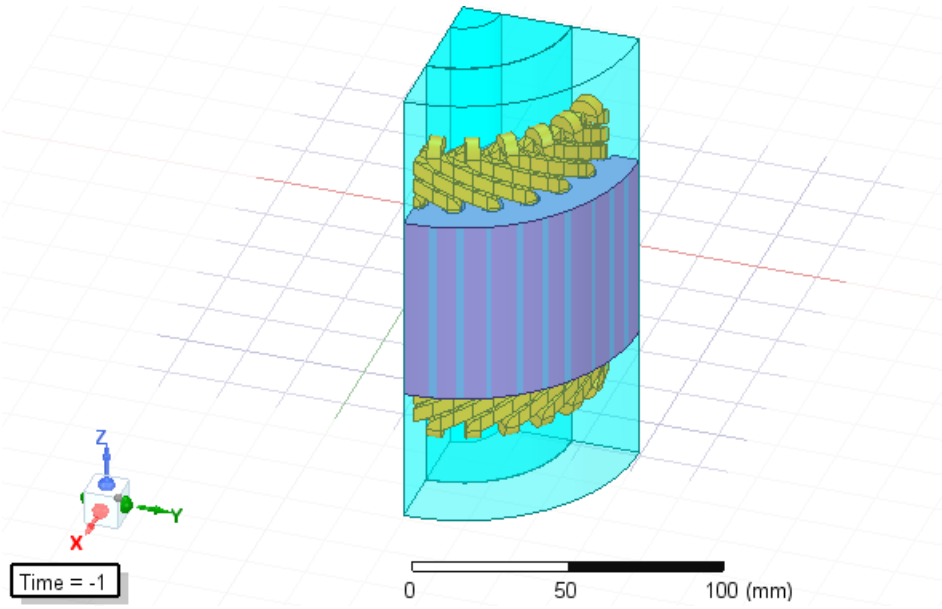


   In the example design, there is only one **Solution Setup** and no design **Variation**.

2. For **Type** choose to create either a **Maxwell 2D Design**, or a **Maxwell 3D Design**.
3. Click **OK** to create the Maxwell design. Examples of the resulting 2D and 3D designs are shown below.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note: You can also export the model to a Simplorer model or Customized Design Sheet.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Up RMxprt Projects

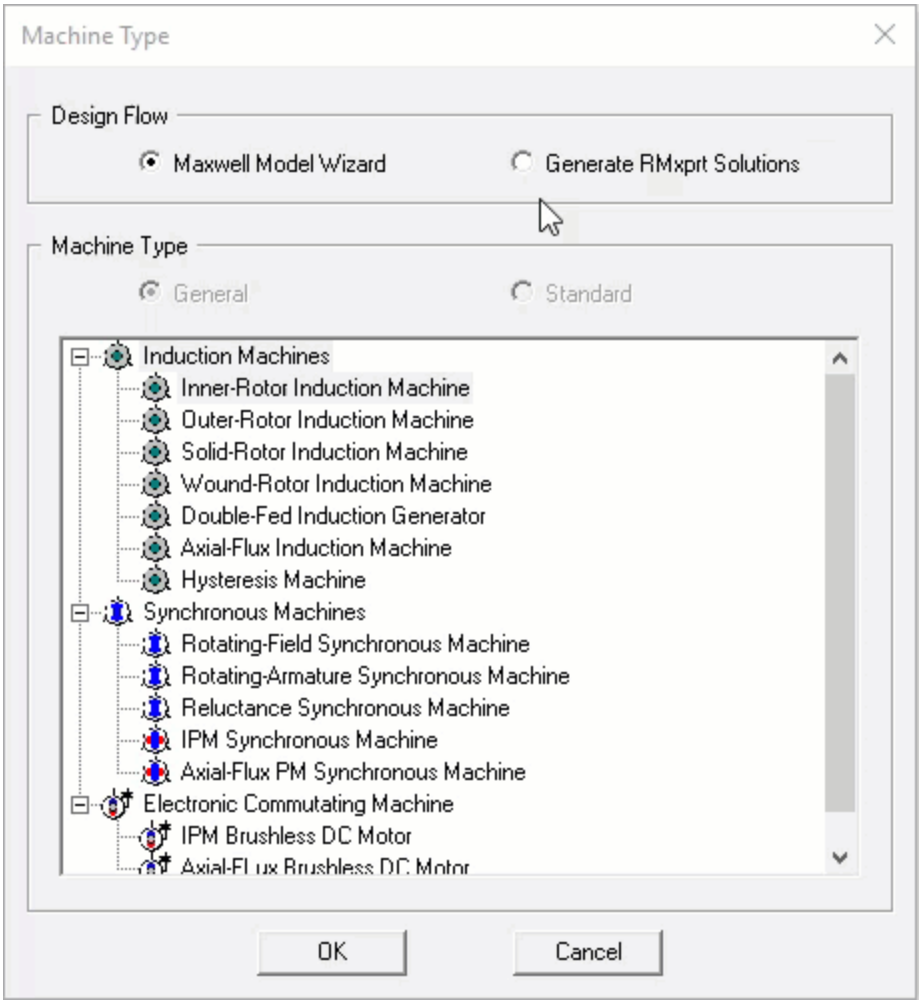
An RMxprt project is a folder that includes one or more models, or *designs*. Each design ultimately includes a geometric model, material assignments, and field solution and post-processing information.

A new project called *Projectn* is automatically created when the software is launched, where *n* is a number. You can also open a new project by clicking **File** > **New**. In general, use the **File** menu commands to manage projects. If you move or change the names of files without using these commands, the software may not be able to find information necessary to solve the model.

To insert an RMxprt design:

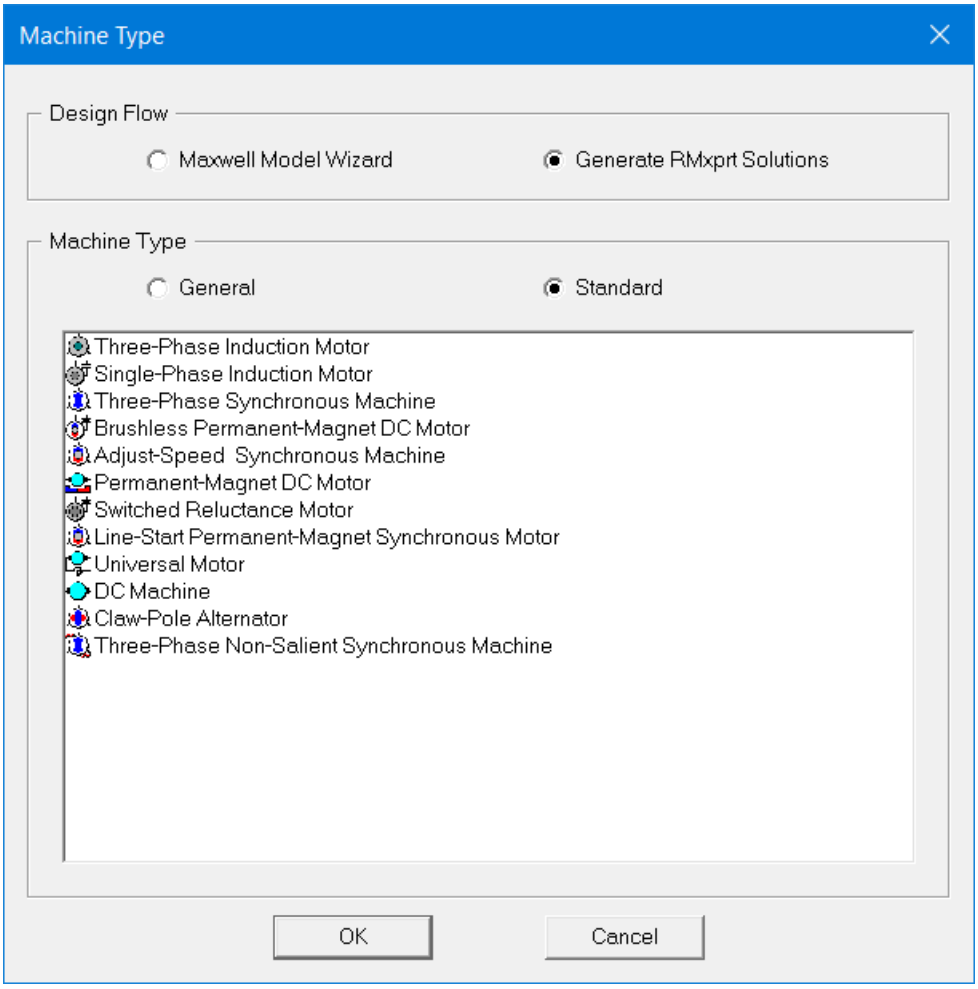
1. Insert an RMxprt design. (Click **Project** > **Insert RMxprt Design**, and specify the machine type from the **Machine Type** window.)
2. Choose the desired **Design Flow** option. You can select either **Maxwell Model Wizard** or **Generate RMxprt Solutions**. If **Maxwell Model Wizard** is selected, you can create an RMxprt design aimed for Maxwell design creation only. You can import your own stator and rotor geometry, with or without material assignment.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If **Generate RMxprt Solutions** is selected, you can select either **General** for various predefined generic rotating machine types, or **Standard** for other predefined machine types.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Right-click on the design tree machine item, or from the RMxprt main menu, click on **Machine Type** to open the **Machine Type*<project_name>* - *<design_name>*** dialog box.

> **Note:** The design flow cannot be changed once it has been set; however, the machine type can be changed for **Generate RMxprt solutions**.

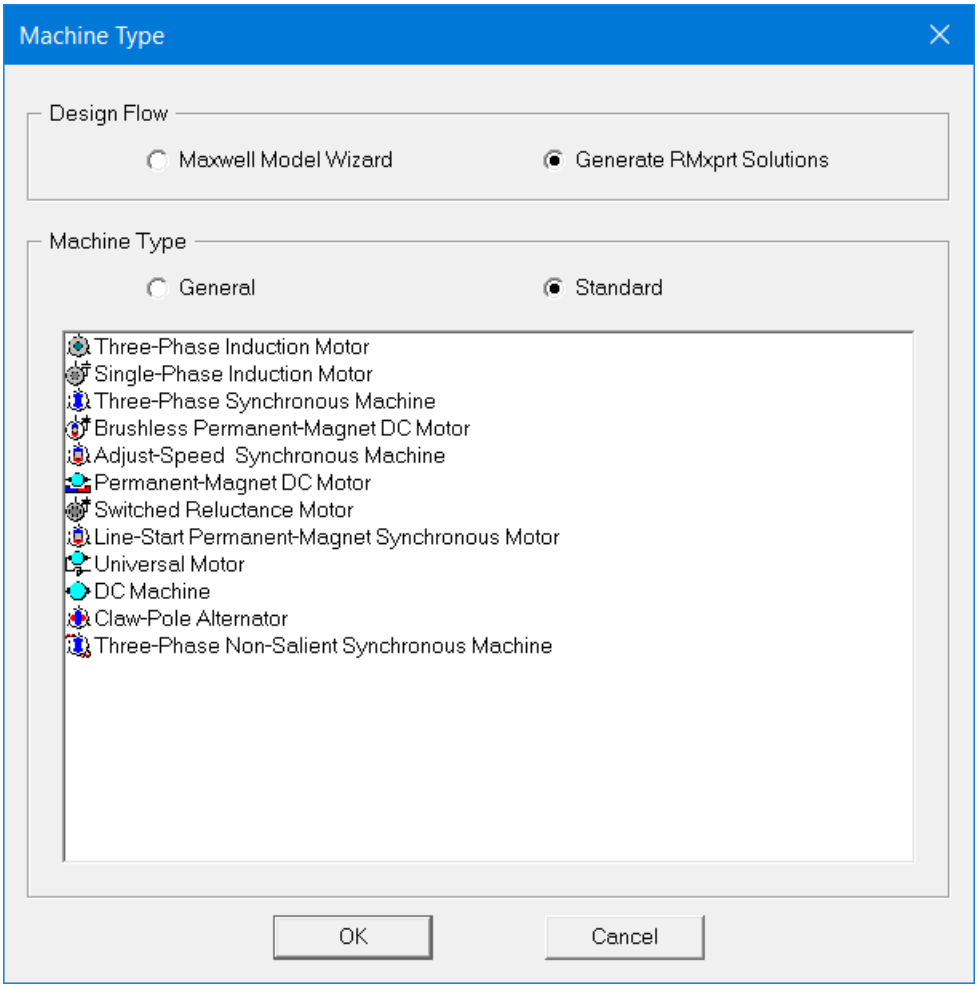
Setting Up A Machine Model

To set up an RMxprt model, follow this general procedure:

1. Insert an RMxprt design. (Click **Project** > **Insert RMxprt Design**, and specify the machine type from the **Machine Type** window.) For the **Design Flow**, choose either **Maxwell Model Wizard** and select a **Machine Type**; or **Generate RMxprt Solutions**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. **Generate RMxprt Solutions** enables the **General** and **Standard** radio buttons. **General** allows you to pick the desired **Generic Rotating Machine** (**GRM**) machine type from the expandable list. **Standard** changes the listing to the basic RMxprt machine templates.



3. Use the **Tools** menu to specify general options (such as post-processing and auto-save settings), solver options (such as the default process priority), and specific RMxprt options. Also specify the Machine options (such as the units and the wire setting such as the wire shape and gauge).
4. Double-click the Machine items in the Project Manager tree, to specify the settings for the various parts of the selected machine parameters.
5. Under Definitions in the Project Manager tree, assign any Materials to the machine parts, setting values such as:
    - Permanent magnet definition, including the coercivity, energy density, and relative recovery permeability.
    - B-H curve parameters.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Use the **Setup** functions (either on the RMxprt menu or on the Analysis or Optimetrics submenus via the project tree) to specify variable, parametric, and optimization settings.
7. Use the Validate function to validate the design.
8. Use the Analyze functions to generate a solution, run a parametric analysis, or run an optimization.
9. Use the Results postprocessing functions to display the lamination and plot the solutions.

Maxwell Model Wizard

For Maxwell Model Wizard design flow, specify the imported geometry (if any) information through the **Machine** property window to let RMxprt know what parts and attributes to generate in the Maxwell design. Then, click **Create Maxwell Design** in the analysis setup menu to create a new Maxwell design.

You can also change machine type, and design flow in **Maxwell Model Wizard**.

Use the **Machine** property window to describe the general properties and geometry import information of the machine model to be created. It consists of the following:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Description
Source Type	AC or DC source to deliver electrical power
Inner Rotor	Inner rotor, outer rotor, or axial-flux rotor
Use Minimum Setting	Hide input settings not needed for basic Maxwell model creation. Use this option to simplify the RMxprt design process to create the Maxwell model (design).
Import Option	Import geometry or Maxwell design (for geometry with material assignment). **None**: Will not import external geometry file for stator/rotor. **Geometry only**: An external geometry file needs to be imported to the design. Supports third party geometry file as well as Maxwell design(.aedt). RMxprt will add material assignments as well as other Maxwell design setups. **Maxwell design**: Only supports import of Maxwell design. The material of geometry will be retained. RMxprt will not add or change material assignments of the imported geometry.
Core Type	The stator/rotor core type
Core Alignment	Use this option to determine the alignment position of core center. Used for RMxprt to calculate the positions of stator/rotor parts, such as magnets, windings, etc. **Pole or tooth center aligned with x-axis**: When the pole center or tooth center of user's geometry is aligned with x-axis, user should choose this option. **Inter-pole or slot center aligned with x-axis**: When the inter-pole or slot center of user's geometry is aligned with x-axis, user should choose this option. Otherwise, adjust the geometry and then choose either option to prevent incorrect Maxwell design geometry generation.
Import Geometry	The geometry file to be imported. **Note:** The imported geometry must be the same dimension as the specified Geometry Mode.
Geometry Mode	Specify the geometry dimension (**2D** or **3D**) so that RMxprt can create other geometry parts of same dimension.
Import Alignment	This option will only be available when fractions value is greater than 1.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Geometry center on x-axis: When the fraction center is aligned with x-axis, choose this option to let RMxprt know the import alignment. **Independent boundary on x-axis**: When edge (or independent boundary) is aligned with x-axis, choose this option to let RMxprt know the import alignment. Otherwise, adjust the geometry and then choose either option to prevent incorrect Maxwell geometry generation.
Import Fractions	Number of fractions of the import geometry. When the value of **Import Fractions** is 1, the created geometry will have the same fractions as the traditional design without importing cores; when the value of **Import Fractions** is greater than 1, the created geometry will have the same fractions as the Import Fractions.
Z Axis Fractions	Only used when geometry mode is 3D. **Whole**: The center of the geometry model is located at the center of the coordinate and symmetric along x-y plane. **Half**: The geometry only includes half of the motor machine. All parts of the geometry model are located at the positive z axis.
View/Edit Geometry Model	Click this button to load the geometry to be imported and view/edit the content of the geometry.

The following table shows the design settings under different import combinations:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Import geometry stator core only	Import geometry stator core with material assignments (Maxwell design)	No stator core import
Import geometry rotor core only	Rotor: material assignments and other rotor parts. Stator: material assignments and other stator parts.	Rotor: material assignments and other rotor parts. Stator: other stator parts.	Rotor: material assignments and other rotor parts. Stator: stator core with material assignments and other stator parts.
Import geometry rotor core with material assignments (Maxwell design)	Rotor: other rotor parts. Stator: material assignments and other stator parts.	Rotor: other rotor parts. Stator: material assignments and other stator parts.	Rotor: other rotor parts. Stator: stator core with material assignments and other stator parts.
No rotor core import	Rotor: rotor core with material assignments and other rotor parts. Stator: material assignments and other stator parts.	Rotor: rotor core with material assignments and other rotor parts. Stator: other stator parts.	Rotor: rotor core with material assignments and other rotor parts. Stator: stator core with material assignments and other stator parts.

Creating a Maxwell Model

To generate a Maxwell model:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Create or select a solution setup in the Project Manager tree.
2. Right-click, and select **Create Maxwell Design** if there is any imported geometry. The geometry mode (2D or 3D) of a generated Maxwell design will depend on the **Geometry Mode** entered in the Machine properties. A Maxwell geometry model is created.
3. If there is no imported geometry, you can select either **Create Maxwell 2D Design** or **Create Maxwell 3D Design** to specify the desired geometry mode to be created.

Changing the Machine Type

RMxprt allows you to change the machine type for an existing design. The original machine data which applies to the new machine type is not retained. Instead, the initial default data for the new machine type is used.

To change the machine type for an existing model:

1. Right-click on the design tree machine item, or from the RMxprt main menu, click on **Machine Type** to open the **Machine Type*<project_name> - <design_name>*** dialog box.
2. Choose the desired new machine type and click the **OK** button.

- The design's machine type label is changed to that of the new machine type. For example, if the original design name and type was **3hp (Single Phase Induction Motor)**, and you change the machine type to **Switched Reluctance Motor**, the design name would be **3hp (Switched Reluctance Motor)**. You can **Undo/Redo** the machine type change if you wish to revert to the original machine type and vice-versa.
- The design tree items (such as the Machine Rotor, Stator, and Shaft data) under the design type will be updated with the default machine data applicable to the new machine type.

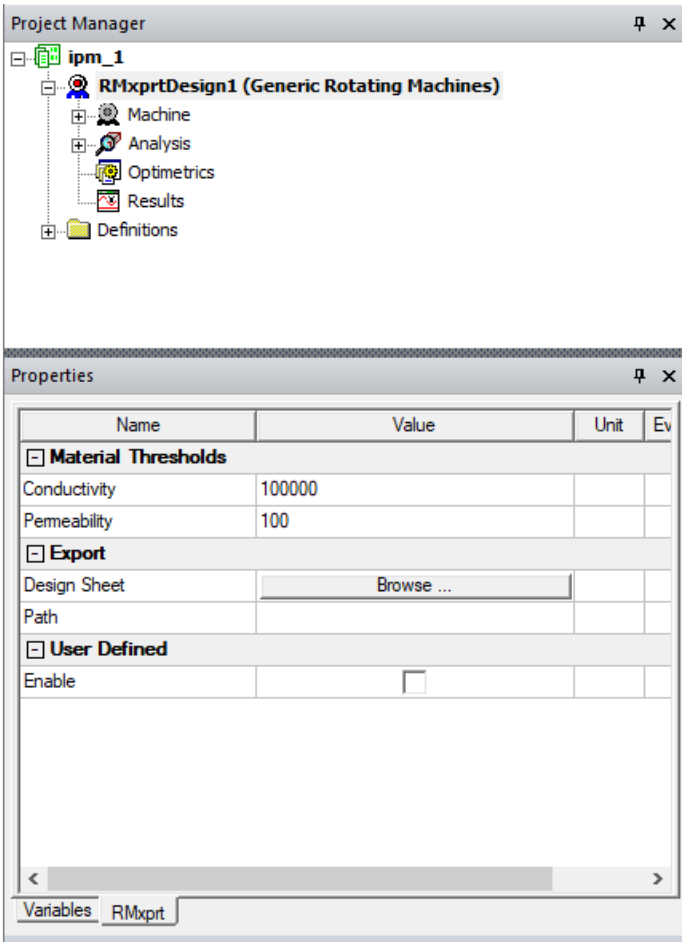
Design Settings in RMxprt

The Design Settings dialog allows you to specify how the simulator will deal with some aspects of the design.

- Set the Material Threshold for treating materials as conductors/insulators.
- Set Export Options.
- Specify User Defined Data.

You can also view and set design settings by selecting the current design, and then in the **Properties** window, select the tab for the solver.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Setting the Material Threshold in RMxprt

1. Click **RMxprt** > **Design Settings.**
   The **Design Settings** window appears with the **Material Threshold** tab selected. You can also view and set design settings by selecting the current design, and then in the **Properties** window, select the tab for the solver.
2. Type a value in the **Conductivity Threshold** text box (Default=10,000).
3. Type a value in the **Permeability** text box (Default=100).

   > **Note:** RMxprt will treat materials with conductivity greater than 10,000 as conductors, and materials with Permeability greater than 100 as steels.

4. If you want these values to be the default, change the values by clicking the **Tools** > **Options** > **General Options** menu and setting the material thresholds in the **RMxprt** > **Threshold** panel.
5. Click **OK**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RMxprt Export Options

To set export options for the project:

1. Click **RMxprt** > **Design Settings**.

   The **Design Settings** dialog box appears. You can also view and set design settings by selecting the current design, and then in the **Properties** window, select the tab for the solver.

2. For the **Design Sheet**, type a file name in the **Excel Template** text box.

   You can also click the **...** button to find and select a file.

3. Click **OK**.

Setting User-Defined Data for a Design

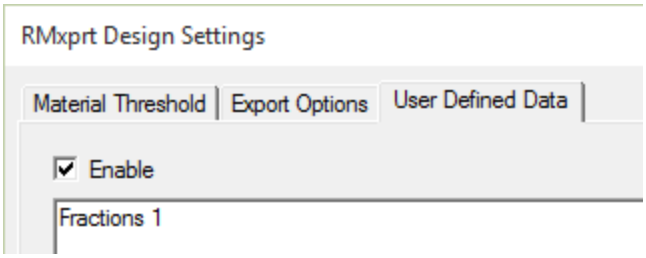
RMxprt allows a user to define some design data in a text file that can be created by a text editor, instead of by RMxprt UI, for the following special circumstances:

- Some special requests from a user that are not necessary to be added to RMxprt UI
- Some common requests from users that have been implemented in RMxprt solver, but have not been added in RMxprt UI
- Specifying various calibration factors
- Specifying Data Based on Design Template Files

When a user's requests have been implemented in an RMxprt solver but have not been added in the RMxprt UI, the updated solver and the required file format for user defined data will be sent to the user. To use the user defined data feature, the user must first edit the data file using a text editor according to the format provided. Then, select **RMxprt** > **Design Settings** to display the Design Settings dialog box. You can also view and set design settings by selecting the current design, and then in the **Properties** window, select the tab for the solver.

1. Select the **User Defined Data** tab.
2. Click the **Enable** check box to enable the use of **User Defined Data**.
3. User defined data may be entered directly into the text box. Click in the box and enter the data entries desired.
   - For example: If you want to create a full 360 deg model of the solved RMxprt design, enter **Fractions 1** in the text box and click **OK**. After solving the RMxprt design, if you choose to create a Maxwell 2D or 3D design, a full 360 model is generated.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Similarly, if you want to create a 180 deg half-model of the solved design, enter **Fractions 2** in the text box.

> **Note:** There are template files of file type **.temp** installed with many of the RMxprt examples located in the *<installdir>***Examples\RMxprt** folder. These text files list keywords (such as **Fractions**), default values, and descriptions of supported user defined data for their related machine types. Refer to *Specifying Data Based on Design Template Files* for details of these template files.

4. Alternatively, click **Import File** to import user defined data from an external file.
5. Browse to the directory containing the file.
6. Select the user defined data file which will be displayed in **File** name box.
7. Click **Open** to confirm the selection.
8. The file contents will be imported into the text box. Click **OK** to complete the setup.

User Defined Data is saved in the design file. Changes to **User Defined Data** will cause existing solutions to become invalid.

Specifying Calibration Factors via User-Defined Data

Calibration factors for various machine parameters are listed below. These can be used for setting user-defined data for a design.

Keyword	Default Value	Description	Applicable to
KCoreloss	1.0	Calibration factor for core loss	All machines
KE0	1.0	Calibration factor for induced voltages produced by PM or field current excitation	All synchronous machines, BLDC motors
KLad	1.0	Calibration factor for d-axis armature reactive inductance	All machines with AC armature windings
KLaq	1.0	Calibration factor for q-axis armature	All machines with AC

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Keyword	Default Value	Description	Applicable to
		reactive inductance	armature windings
KLs_Armature	1.0	Calibration factor for armature winding slot leakage inductance	All machines with AC armature windings
KLs_Damper	1.0	Calibration factor for damper, or squirrel-cage, slot leakage inductance	All machines with damper
KLs_Field	1.0	Calibration factor for field winding slot leakage inductance	All machines with field winding
KLe_Armature	1.0	Calibration factor for armature winding end leakage inductance	All machines with AC armature windings
KLe_Damper	1.0	Calibration factor for damper, or squirrel-cage, end leakage inductance	All machines with damper
KLe_Field	1.0	Calibration factor for field winding end leakage inductance	All machines with field winding
KLend_Armature	1.0	Calibration factor for armature winding end-coil length	All machines with AC armature windings
KLend_Field	1.0	Calibration factor for field winding end-coil length	All machines with field winding

Note:

- For field winding, the spaces between poles are referred to as slots.
- Rotor winding is referred to as "Field Winding" for DFIG machines.

Specifying Data Based on Design Template Files

There are template files of file type **.temp** installed with some of the RMxprt examples located in the <*installdir*>**Examples\RMxprt** folder. These text files list keywords, default values, and descriptions of supported user defined data for their related machine types. These can be used to set user-defined data for a design. Below are the contents of the **.temp** files located in the <*installdir*>**Examples\RMxprt** folder.

...\afpm\afbldc.temp (Axial flux brushless DC motor)

LeadAngle 0 // the angle that applied AC voltage leading the back EMF

Fractions 0 // 0(MinimumModel); 1(FullModel); 2(HalfModel); 3(ThirdModel); ...

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

...\assm\assm.temp (Adjustable-speed synchronous motor)
WireResistivity 0 // when WireResistivity = 0, use default value of 0.0217 ohm.mm^2/m
WireDensity 0 // when WireDensity = 0, use default value of 8900 kg/m^3
LimitedTorque 0 // when LimitedTorque < ComputedRatedTorque, use ComputedRatedTorque for flux weakening control
ControllingIq 0 // controlling q-axis current for dq-current control, 0 without dq-current control
ControllingId 0 // controlling d-axis current for dq-current control
ArcBottom 0 // 0(LineBottom); 1(ArcBottom) for slot types 3 & 4
Fractions 0 // 0(MinimumModel); 1(FullModel); 2(HalfModel); 3(ThirdModel); ...
...\bldc\bldc.temp (Brushless DC motor)
ArcBottom 0 // 0(LineBottom); 1(ArcBottom) for slot types 3 & 4
Fractions 0 // 0(MinimumModel); 1(FullModel); 2(HalfModel); 3(ThirdModel); ...
...\cpsg\cpsg.temp (Claw-pole synchronous machine)
Connection 0 // 0(Wye); 1(Delta) for stator winding connection
ArcBottom 0 // 0(LineBottom); 1(ArcBottom) for slot types 3 & 4
Fractions 0 // 0(MinimumModel); 1(FullModel); 2(HalfModel); 3(ThirdModel); ...
...\dcm\dcm.temp (DC machines)
ArcBottom 0 // 0(LineBottom); 1(ArcBottom) for slot types 3 & 4
Fractions 0 // 0(MinimumModel); 1(FullModel); 2(HalfModel); 3(ThirdModel); ...
LineRes 0 // additional lead wire resistance in the armature branch
...\dfig\dfig.temp (Double-fed induction generator)
StatorCktType 0 // 0(Y3); 1(L3); 2(S3); 3(C2); 4(L4); 5(S4)
RotorCktType 0 // 0(Y3); 1(L3); 2(S3); 3(C2); 4(L4); 5(S4)
MPPTUntilRatedSpeed 0 // 0(Normal MPPT); 1(MPPT until rated speed, then const wind power)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

...\indm1\indm1.temp (Single phase induction motor)
SpeedAdjustMode 0 // 0(None); 1(L-Mode); 2(T-Main); 3(T-Aux)
AdjustTurnRatio 0 // turn ratio of the adjusting winding to the original main/aux winding
AdjustDiaRatio 0 // wire diameter ratio of the adjusting winding to the original main/aux winding
WireResistivity 0 // when WireResistivity = 0, use default value of 0.0217 ohm.mm^2/m
WireDensity 0 // when WireDensity = 0, use default value of 8900 kg/m^3
AuxResistivity 0 // when AuxResistivity = 0, use default value of 0.0217 ohm.mm^2/m
AuxDensity 0 // when AuxDensity = 0, use default value of 8900 kg/m^3
BarFitGap 0 // the fitting gap between bar and slot for squirrel-cage rotor
StartConstrain 0 // the constraint of the ratio of the start current or torque to the rated one
AuxCoilOnTop 0 // 0(MainOnTop); 1(AuxOnTop): for sin-wave windings only
ArcBottom 0 // 0(LineBottom); 1(ArcBottom) for slot types 3 & 4
Fractions 0 // 0(MinimumModel); 1(FullModel); 2(HalfModel); 3(ThirdModel); ...
...indm1\3hp.temp (Single-phase induction motor - 3 hp)
SpeedAdjustMode 1 // 0(None); 1(L-Mode); 2(T-Main); 3(T-Aux)
AdjustTurnRatio 0.5
AuxCoilOnTop 0 // 0(MainOnTop); 1(AuxOnTop): for sin-wave windings only

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

...\indm3\indm3.temp (Three-phase induction motor)
WireResistivity 0 // when WireResistivity = 0, use default value of 0.0217 ohm.mm^2/m
WireDensity 0 // when WireDensity = 0, use default value of 8900 kg/m^3
Xe1CorrectionFactor 1 // correction factor for stator-winding end leakage reactance
TopSpareSpace 0 // the spare space above the working coil in a slot, between [0, 1]
BottomSpareSpace 0 // the spare space below the working coil in a slot, between [0, 1]
LimitedTorque 0 // when LimitedTorque < ComputedRatedTorque, use ComputedRatedTorque for flux weakening control
ArcBottom 0 // 0(LineBottom); 1(ArcBottom) for slot types 3 & 4
Fractions 0 // 0(MinimumModel); 1(FullModel); 2(HalfModel); 3(ThirdModel); ...
CtrlType 3 // 0(DC); 1(CCC); 2(PWM); 3(AC)
BarFitGap 0 // the fitting gap (mm) between bar and slot for squirrel-cage rotor
...\indm3\yz200-6.temp (Three-phase induction motor - 6 pole)
TopSpareSpace 0.57 // between [0, 1]
BottomSpareSpace 0 // between [0, 1]
...\indm3\yz200-24.temp (Three-phase induction motor - 24 pole)
TopSpareSpace 0 // between [0, 1]
BottomSpareSpace 0.43 // between [0, 1]
...\ipm\ipm.temp (Interior permanent-magnet machine)
ArcBottom 0 // 0(LineBottom); 1(ArcBottom) for slot types 3 & 4
Fractions 0 // 0(MinimumModel); 1(FullModel); 2(HalfModel); 3(ThirdModel); ...
SolidCore 1 // 0 for original core; 1 for solid inner rotor core
SleeveThick 0.5 // the sleeve thickness of a solid rotor core when SolidCore=1
HysThick 7 // thickness of the hysteresis layer when SolidCore=1
Hci 50000 // intrinsic Hc of the hysteresis loop when SolidCore=1
Br 1.6 // Br of the hysteresis loop when SolidCore=1
PartialSol 0 // 1 for not output performance data in case error

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

...\lssm\lssm.temp (Line start permanent-magnet synchronous machine)
ArcBottom 0 // 0(LineBottom); 1(ArcBottom) for slot types 3 & 4
Fractions 0 // 0(MinimumModel); 1(FullModel); 2(HalfModel); 3(ThirdModel); ...
CtrlType 3 // 0(DC); 1(CCC); 2(PWM); 3(AC)
BarFitGap 0 // the fitting gap (mm) between bar and slot for squirrel-cage rotor
...\nssm\nssm.temp (Non-salient synchronous machine - 3 phase)
ArcBottom 0 // 0(LineBottom); 1(ArcBottom) for slot types 3 & 4
Fractions 0 // 0(MinimumModel); 1(FullModel); 2(HalfModel); 3(ThirdModel); ...
...\pmdc\pmdc.temp (Permanent-magnet DC motor)
ArcBottom 0 // 0(LineBottom); 1(ArcBottom) for slot types 3 & 4
Fractions 0 // 0(MinimumModel); 1(FullModel); 2(HalfModel); 3(ThirdModel); ...
...pmsg\pmsg.temp (Adjust-speed synchronous machine)
CapacitivePF 0 // 0(Inductive); 1(Capacitive) for power factor angle
ArcBottom 0 // 0(LineBottom); 1(ArcBottom) for slot types 3 & 4
Fractions 0 // 0(MinimumModel); 1(FullModel); 2(HalfModel); 3(ThirdModel); ...
...pmsg\pmsg1.temp (Adjust-speed synchronous machine - generator)
CapacitivePF 1 // 0(Inductive); 1(Capacitive)
Fractions 0 // for 2D/3D design of all machine types. 0: default; 1: whole model
...\srm\srm.temp (Switched reluctance motor)
ArcBottom 0 // 0(LineBottom); 1(ArcBottom) for slot types 3 & 4
Fractions 0 // 0(MinimumModel); 1(FullModel); 2(HalfModel); 3(ThirdModel); ...
...\syng3\syng3.temp (Three-phase synchronous machine)
CapacitivePF 0 // 0(Inductive); 1(Capacitive) for power factor angle
ArcBottom 0 // 0(LineBottom); 1(ArcBottom) for slot types 3 & 4
Fractions 0 // 0(MinimumModel); 1(FullModel); 2(HalfModel); 3(ThirdModel); ...
BarFitGap 0 // the fitting gap between bar and slot for squirrel-cage rotor
...\unim\unim.temp (Universal motor)
ArcBottom 0 // 0(LineBottom); 1(ArcBottom) for slot types 3 & 4
Fractions 0 // 0(MinimumModel); 1(FullModel); 2(HalfModel); 3(ThirdModel); ...

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Validating RMxprt Projects

Before you run an analysis on a model, it is very important that you first perform a validation check on the project. When you perform a validation check on a project, RMxprt runs a check on all setup details of the active project to verify that the necessary steps have been completed and their parameters are reasonable.

To perform a validation check on the active project:

1. Click **RMxprt** > **Validation Check**. You can also click the **Validate** button on the **Simulation** ribbon.

   RMxprt checks the project setup, and then the **Validation Check** window appears.

2. View the results of the validation check in the **Validation Check** window.

   The following icons can appear next to an item:

   Indicates the step is complete.

   Indicates the step is incomplete.

   Indicates the step may require your attention.

3. View any messages in the **Message Manager** window.
4. If the validation check indicates that a step in your project is incomplete or incorrect, carefully review the setup details for that particular step and revise them as necessary.
5. Click **RMxprt** > **Validation Check** to run a validation check after you have revised any setup details for an incomplete or incorrect project step**.**
6. Click **Close**.

Setting General Options in RMxprt

Default settings for many of the options in RMxprt may be set through the **Tools** > **Options** menu. To set general options for RMxprt:

1. Click **Tools** > **Options** > **General Options**.
   The **General Options** window appears, displaying six available tabs:
   - Project Options
   - Miscellaneous Options

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Default Units
- Remote Analysis Options
- Desktop Performance

2. Click each tab, and make the desired selections.
3. Click **OK**.

Setting RMxprt Options

To specify settings for RMxprt options:

1. Click **Tools** > **Options** > **General Options** to open the **Options** window, displaying a hierarchical list of options.
2. Click the **+** button next to **RMxprt** to show the following choices: **General**, **Machine Type**, **Threshold**.
3. In the **General** options panel, select or clear the following check boxes:
   - **Save before solving** – Note that when you enable this setting, the project is only saved if it has been modified since its last save.
   - **Apply variation deletions immediately**
4. On the **Machine Type** panel, to change the default machine type when you initially insert a design, select one of the following from the **Default machine type** drop-down menu:
   - Three-Phase Induction Motor
   - Single-Phase Induction Motor
   - Three-Phase Synchronous Machine
   - Brushless Permanent-Magnet DC Motor
   - Adjust-Speed Synchronous Machine
   - Permanent-Magnet DC Motor
   - Switched Reluctance Motor
   - Line-Start PM Synchronous Motor
   - Universal Motor
   - DC Machine
   - Claw-Pole Synchronous Machine
   - Three-Phase Non-Salient Synchronous Machine
   - Generic Rotating Machines
5. On the **Threshold** panel, enter the **Default conductivity** and **Default permeability** values in siemens/m.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** Setting the material thresholds under **Tools** > **Options** impacts the default setting for the current and all future projects/designs. To change the material threshold for the current design only, use the **RMxprt** > **Design Settings** window to change the material thresholds on the **Set Material Thresholds** tab.

6. Click **OK** to close the window.

Configuring a Wire Specification Library

In order to be able to select wire size for your design, your need to configure a wire specification library *before* you insert a new design in your project. RMxprt provides the following libraries of wire gauge specifications based on the current widely used standards for bare copper wire gauges (including both round and rectangular wires). These files are stored in the *syslib* directory.

American.wir	File American.wir provides dimensions for all ANSI bare wires.
ANSI_ SingleFilm.wir **ANSI_ HeavyFilm.wir** **ANSI_ TripleFilm.wir** **ANSI_ QuadFilm.wir**	Files ANSI*.wir provide dimensions for bare and film insulated copper wires with single, heavy, triple and quad builds of integer and half AWG numbers.
AWG_Int_ SingleFilm.wir **AWG_Int_ HeavyFilm.wir** **AWG_Int_ TripleFilm.wir** **AWG_Int_ QuadFilm.wir**	Files AWG_Int*.wir provide dimensions of integer AWG numbers.
Chinese.wir	
IEC_ R20Grade1.wir **IEC_ R20Grade2.wir** **IEC_ R20Grade3.wir**	Files IEC_R20*.wir provide dimensions of enameled copper wires with Grade-1, 2, and 3 insulations of R20 series. Files IEC_R40*.wir provide dimensions of R40 series. In files IEC*.wir, the gauge numbers are equivalent to the AWG numbers according to the nominal

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

IEC_ R40Grade1.wir	diameters.
IEC_ R40Grade2.wir	
IEC_ R40Grade3.wir	

1. Click **Tools** > **Options** > **General Options** to open the **Options** dialog box, displaying a hierarchical list of options. Click the **+** button next to **Machines** to show the following choice: **Wire Settings**.
2. Select the desired wire library from the drop-down menu. In addition to the System Libraries choices listed above, you can also choose wire libraries you may have in either the User or Personal libraries.
3. Edit wire data as needed.

Editing Wire Data

Users must generate their own data files for wire gauges using data for wire gauge and thickness of insulation provided by manufacturers. There are no national standards for insulation thickness, therefore different manufacturers produce electromagnetic wire with different thickness of insulation. The data file **American.wir** does not provide data for thickness of insulation; the data file **Chinese.wir** does provide the data for thickness of insulation, but only for the purpose of reference to users. These files are stored in the file folder *syslib* along with several other wire data reference libraries.

To define or edit wire data:

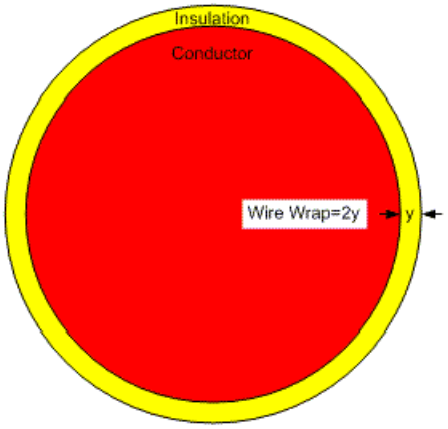
1. Click **Machine** > **Wire** to open the **Edit Wire Data** dialog box**.**
2. Select the units from the **Unit System** drop-down menu.

   English Unit System stands for British unit system, **Metric Unit System** stands for the metric unit system. When changing the unit system, the message box **Note** pops up to inform changing in unit system is only for specifying input data unit, but not for transferring data between two unit systems

3. Optionally, click **Import** to import wire data from a file. Provide the file name to import in the **File name:** edit box (or by browsing) and use the default file type *Wire Size File (*.wir)*. Several standard wire libraries are provided in the *syslib* directory.
4. Click the **Round** or **Rectangle** tab for the wire shape you want to edit.

   For **Round**:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



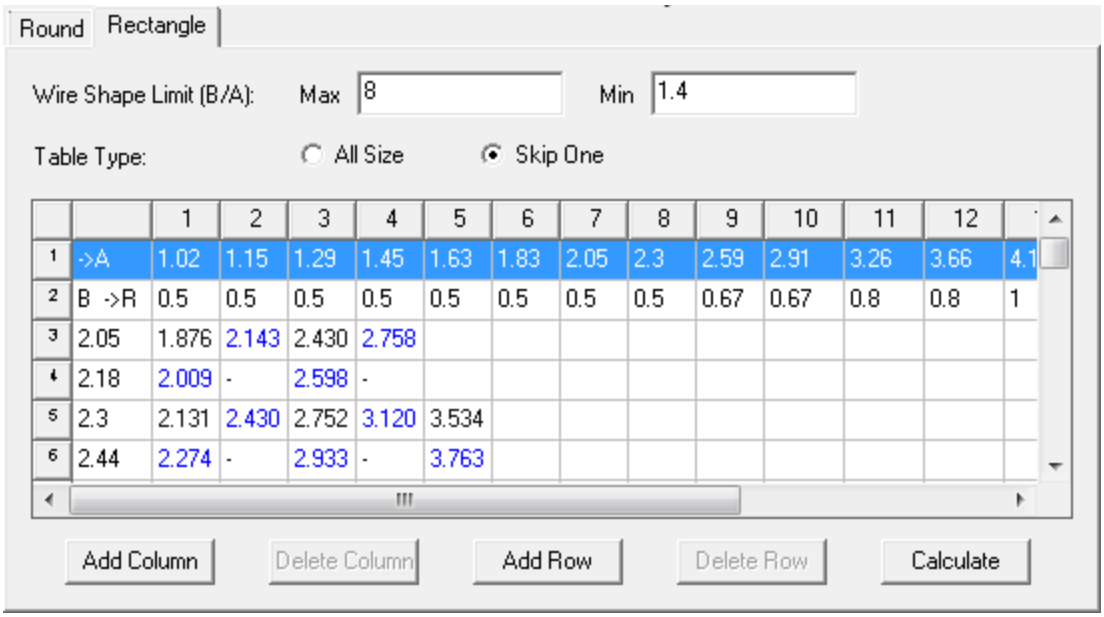
a. Specify the desired values for Gauge No., Diameter, and/or Wrap.

Gauge No.	wire gauge index number.
Diameter	diameter of bare copper wire, in mm or inch.
Wrap	thickness of insulation wrap, in mm or inch.

b. Modify the wire data as needed.

c. Optionally, Add or Delete Rows for wire values.

For **Rectangle**:



		1	2	3	4	5	6	7	8	9	10	11	12	
1	->A	1.02	1.15	1.29	1.45	1.63	1.83	2.05	2.3	2.59	2.91	3.26	3.66	4.1
2	B ->R	0.5	0.5	0.5	0.5	0.5	0.5	0.5	0.5	0.67	0.67	0.8	0.8	1
3	2.05	1.876	2.143	2.430	2.758									
4	2.18	2.009	-	2.598	-									
5	2.3	2.131	2.430	2.752	3.120	3.534								
6	2.44	2.274	-	2.933	-	3.763								

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Specify the desired values to limit ratios of the two sides.

Wire Shape Limit (B/A) max	the maximum ratio between the wide and the narrow sides.
Wire Shape Limit (B/A) min	the minimum ratio between the wide and the narrow sides.

b. Use the radio buttons to specify whether to consider priority factors.

All Size – for No Consideration of Priority Factors. Select the radio button **All Size**, and then click the **Calculate** button. All the sectional areas of wire gauge with the ratio B/A between the wide and the narrow sides satisfying the condition **(B/A) max** > B / A > **(B/A) min** appear in the table.

Skip One – for Consideration of Priority Factors. Select the radio button **Skip One**, and then click the **Calculate** button. All the sectional areas of wire gauge with the ratio B/A between the wide and the narrow sides satisfying the condition **(B/A) max** > B / A > **(B/A) min** appear in three different modes in the table **Rectangular Wire Data**.

- At the intersection of the odd columns and the odd rows, the sectional areas appear in black numbers (recommended to use).
- At the intersection of the odd columns and the even rows or the even columns and the odd rows, the sectional areas appear in blue numbers (rarely used).
- At the cross of the even columns and the even rows, the sectional areas do not show (generally not used).

c. Optionally, add or delete rows or columns for wire values.

5. Optionally, click **Export** to export the data you entered to a file. Provide the file name to import in the **File name:** edit box (or by browsing) and use the default file type *Wire Size File (*.wir)*. The default directory for an exported wire data file is *userlib*.

6. When you are finished, click **Save** to save the data, and click **Close** to close the window.

> **Note:** Saving wire data only updates the wire data in the active design.

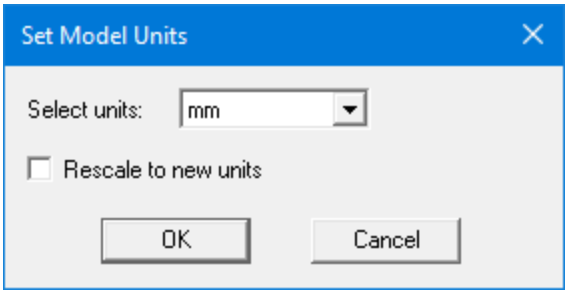
Setting RMxprt Machine Model Units

To set the machine model's units of measurement:

1. Click **Machine** > **Units**.

   The **Set Model Units** dialog box appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select the new units for the model from the **Select units** drop-down menu.
3. Specify how the change in units affects the model:
    - Select the **Rescale to new units** option to rescale the dimensions to the new units. For example, selecting centimeters (cm) as the new unit of measurement results in a dimension of 10 millimeters (mm) becoming 10 cm.
    - Clear the **Rescale to new units** option (the default) to convert the dimensions to the new units without changing their scale. For example, selecting cm as the new unit of measurement results in a dimension of 10 mm becoming 1 cm.
4. Click **OK** to apply the new units to the model.

Edit AC Windings

RMxprt can automatically arrange almost all commonly used single- or double-layer poly-phase ac windings provided all coils have the same number of turns. Users do not need to define coils one by one. For a double-layer winding, RMxprt can also handle the coils with half turns which are arranged in the order of even, odd, even, odd, …, as long as it is physically possible.

RMxprt also provides a very flexible tool, the **Winding Editor**, to allow users to design a variety of special winding types according to their own needs, such as compound single- and double-layer winding, big- and small-phase-spread variable-pole multiple-speed winding, sine-wave three-phase winding, and so forth. The **Winding Editor** is available to the following types of electric machines:

- Three-phase induction motors
- Single-phase induction motors
- Three-phase synchronous motors and generators
- Line-start permanent-magnet synchronous motors
- Claw-pole alternators

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

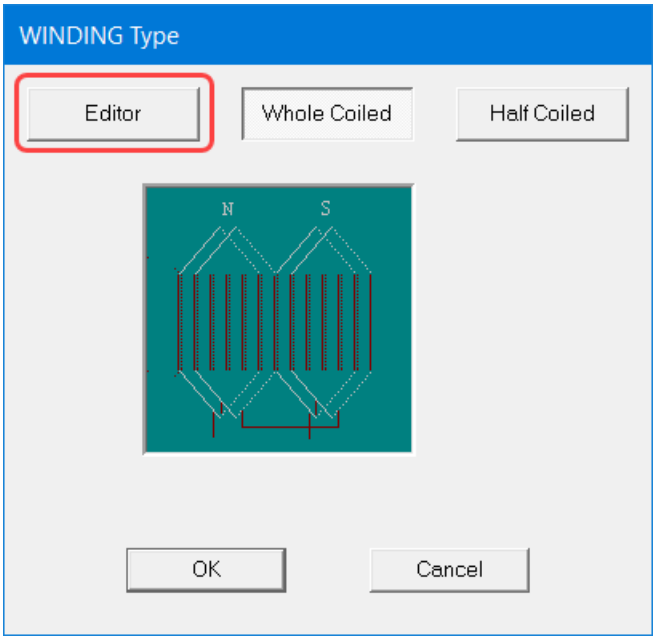
- Adjustable-speed permanent-magnet synchronous motors and generators
- Brushless permanent-magnet DC motors

When you edit the AC winding of a new design for the first time, RMxprt creates a default winding arrangement based on the basic winding specifications: **Number of Phases**, **Number of Poles**, **Number of Slots**, **Winding Layers**, **Conductors per Slot**, and **Coil Pitch**. Then you can edit the winding configuration based on the default arrangement.

Enable Winding Editor

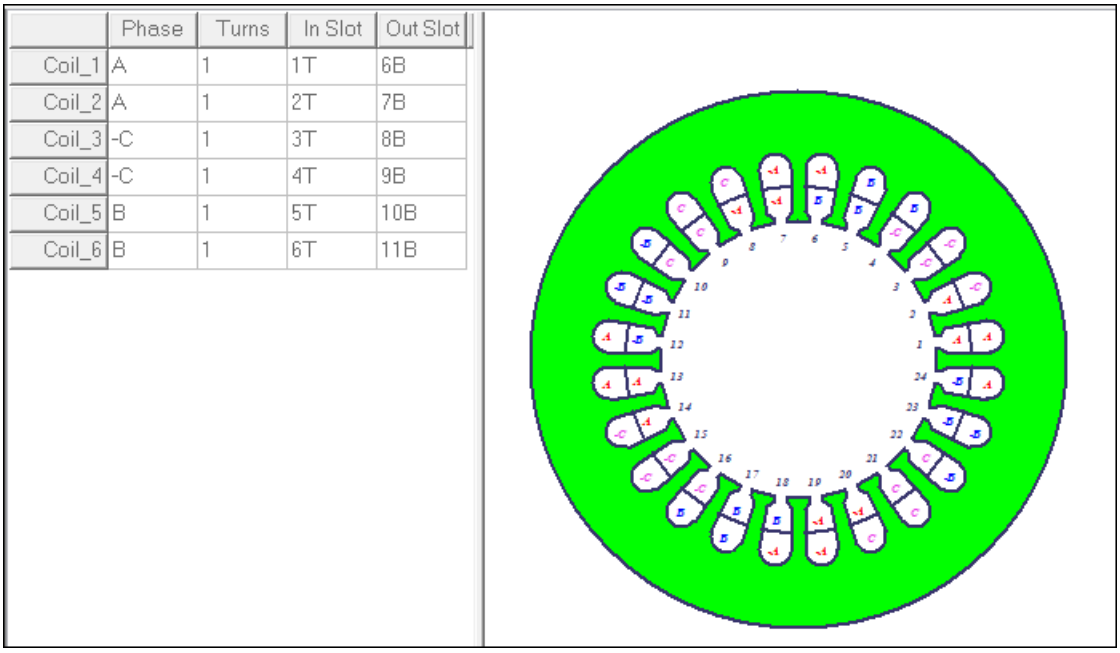
Setting the **Winding Type** property to **Editor** enables the command **Machine** > **Edit Layout** on the menu bar.
To display the dialog box **Winding Editor**:

1. Select **Winding** in the Project Manager tree. In the Properties window, set the Winding Type Value to **Editor**. To do this, click on the button **Winding Type Value** to display the WINDING Type selection window, as shown:



2. Select **Editor** as the Winding Type and click **OK**.
   This closes the WINDING Type selection window and sets the Winding Type Value to Editor. It also enables the command **Machine** > **Winding** > **Edit Layout** on the menu bar. Now the Machine Editor window displays the default winding arrangement, as shown:

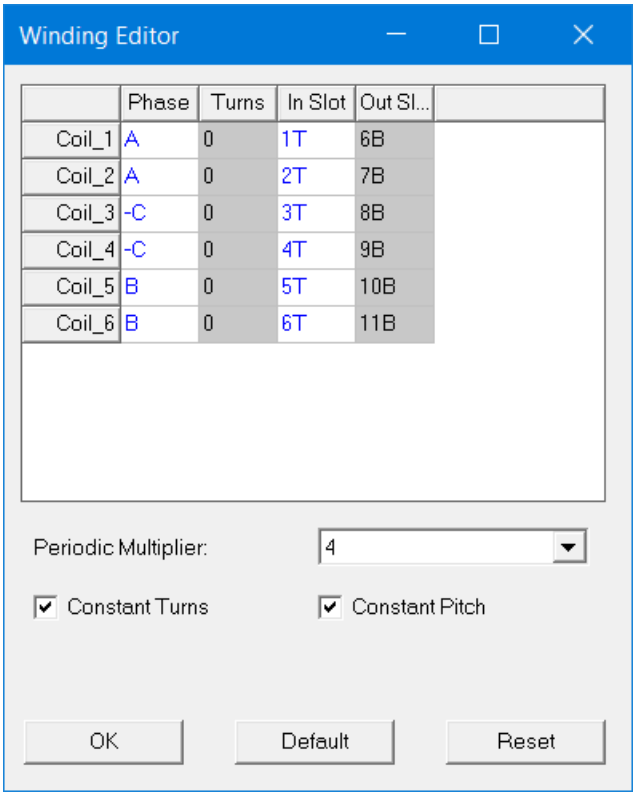
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



	Phase	Turns	In Slot	Out Slot
Coil_1	A	1	1T	6B
Coil_2	A	1	2T	7B
Coil_3	-C	1	3T	8B
Coil_4	-C	1	4T	9B
Coil_5	B	1	5T	10B
Coil_6	B	1	6T	11B

3. Click **Machine** > **Winding** > **Edit Layout**.
   This displays the **Winding Editor** dialog box as shown. The Winding Editor dialog box includes functions that do not appear in the Winding Editor tab sheet in the RMxprt Machine Editor window.
   In addition, right-clicking in the data table section of the Winding Editor tab in the Machine Editor displays a shortcut menu where you may also select **Edit Layout**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Edit Winding Configuration

Each row of the winding data table in the Winding Editor dialog box is identified with the coil index in the column Coil. This information is displayed in the tab sheet Winding Editor in the RMxprt Machine Editor window as well, but it is editable in the dialog box Winding Editor.

The winding data table contains four columns:

Phase	Defines the phase to which the coil belongs
Turns	Defines the number of turns of the coil
In Slots	Defines the slot number with the coil side current flowing in ('flow-in-side' for short). If two layers are specified in the Winding Properties window, the slot number ends with a "T" to show the top layer.
Out Slots	Defines the slot number with the coil side current flowing out ("flow-out-side" for short). If two layers are specified in the Winding Properties window, the slot number ends with a "B" to show the bottom layer.

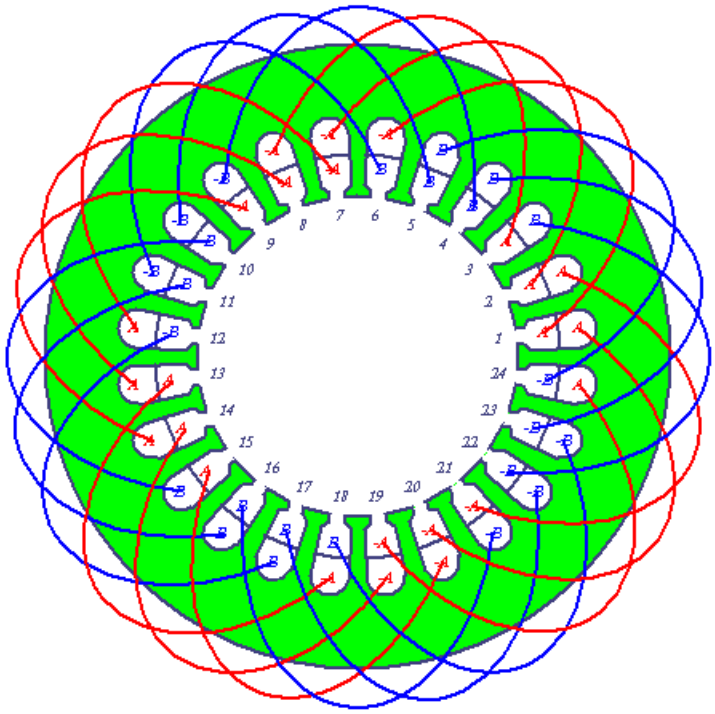
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

View Winding Connections

When you have specified the winding data, you can execute the following commands to display or hide the winding connections.

1. Click the menu command **Machine** > **Winding** > **Connect All Coils**.

   Upon executing, the graphical display in the Machine Editor window shows the connections as shown:



2. To remove the connections in the graphical display in the Machine Editor window, select **Machine** > **Winding** > **Disconnect All Coils**.
3. Winding connections may also be viewed by shortcut menu. **Right-click** on the **winding layout** section of the Machine Editor window, a shortcut menu pops up. Select **Connect All Coils** or **Disconnect All Coils** to toggle the coils display on or off.

Upon executing, the graphical display in the Machine Editor window shows the connections as shown:

If you right-click a slot layer, commands related to that slot layer will be enabled, and you will be able to view or hide only one coil or one phase connection related to the slot layer. You may copy the connection drawing to clipboard from the shortcut menu as well.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Working with the Slot Editor

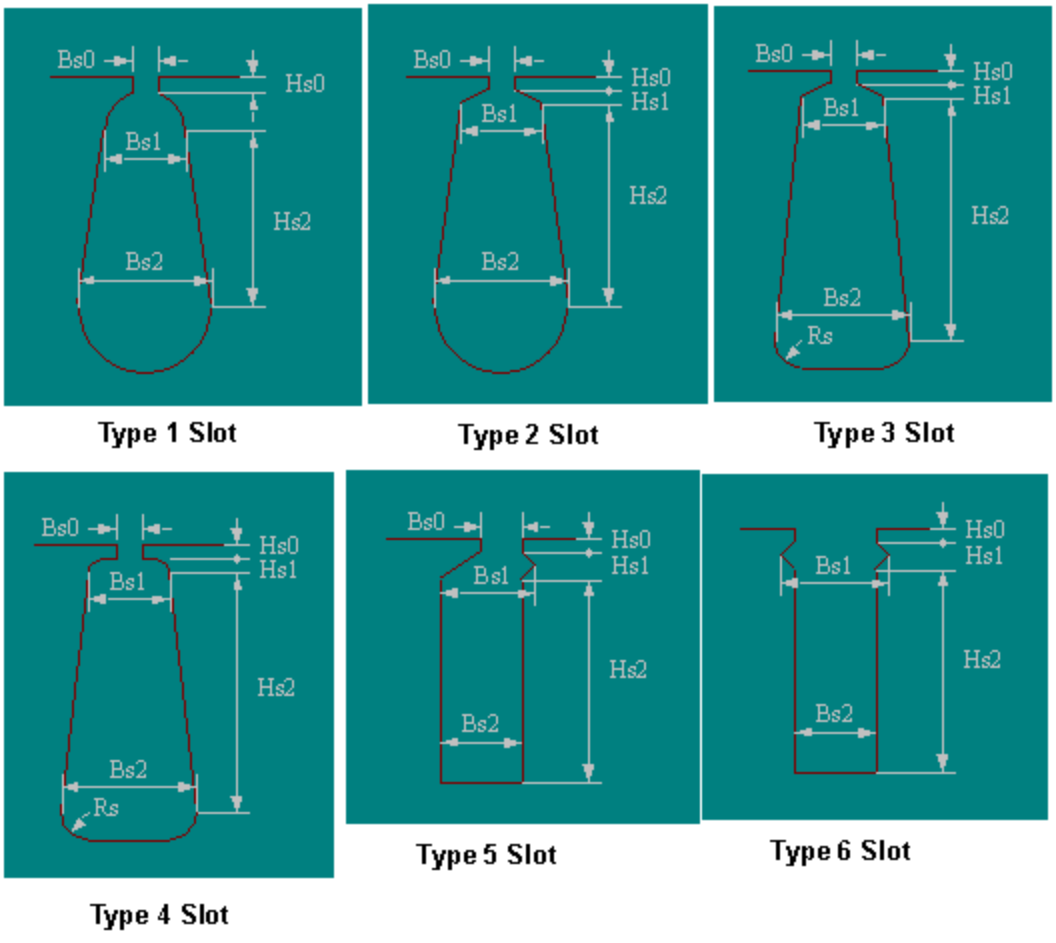
RMxprt provides a flexible tool, the **Slot Editor**, to enable users to edit user-defined slots easily.

Standard slot types in a project can be converted to equivalent user-defined versions that can be edited in the RMxprt Slot Editor tool as follows:

1. In the Project Manager tree, select the rotor or stator whose slot is to be converted.
2. In the Rotor or Stator Properties Window, click the **Slot Type Value** button to open the **Select Slot Type** dialog box.
3. Check **User Defined Slot**, and click **OK**.

The slot label in the Project Manager tree changes to one of the following:

UserDefSymmetricSlot (user-defined symmetric slot), **UserDefUnsymmetricSlot** (user-defined unsymmetric slot), or **UserDefHalfSlot** (user-defined half slot), depending on the standard Slot Type on which it is based.



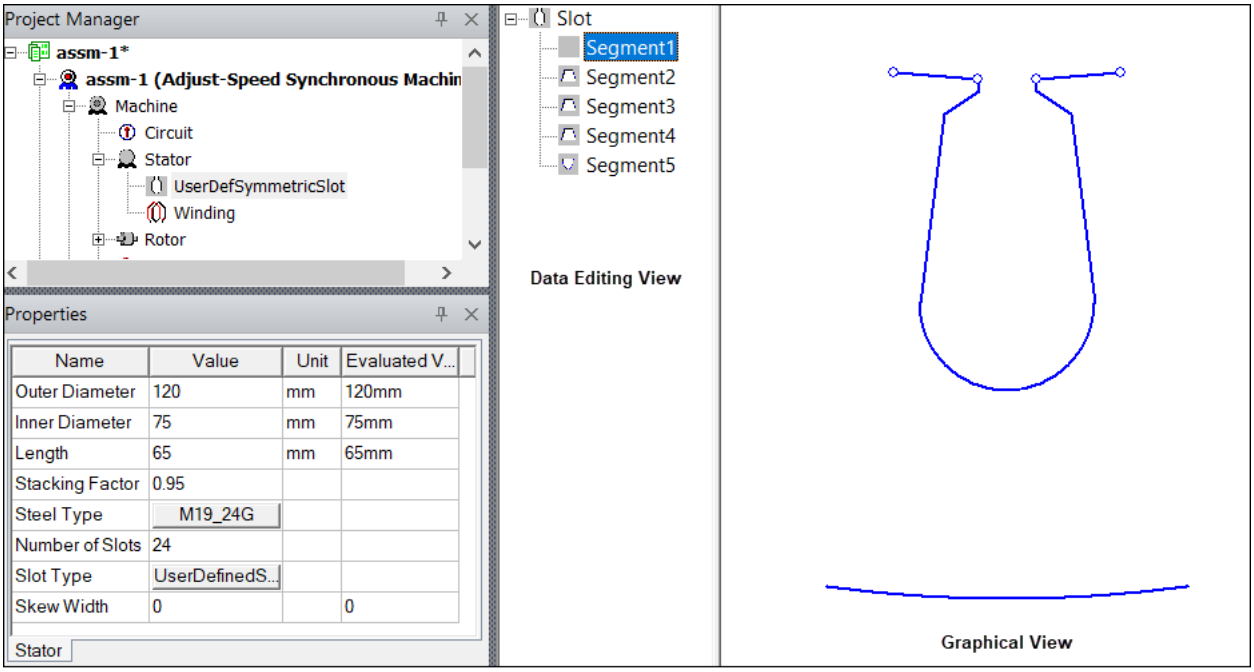
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Standard Slot Types 1, 2, 3, 4, and 6 are symmetric. Standard slot Type 5 is unsymmetric. Half slots are present if the machine's Rotor or Stator Properties Window includes a Half Slot property that has been enabled.

The Slot Editor Window

When you select a user-defined slot in the Project Manager tree, the Slot Editor window appears on the desktop. The slot editor window is split into two frames. The left frame is the data editing view, which contains an expandable tree view of the slot and its constituent segments. The right frame shows a graphical view of the slot geometry formed by its segments.

Slot geometry types that can be edited are: *symmetric*, *unsymmetric*, and *half-slot*. A symmetric slot is shown in the following figure.

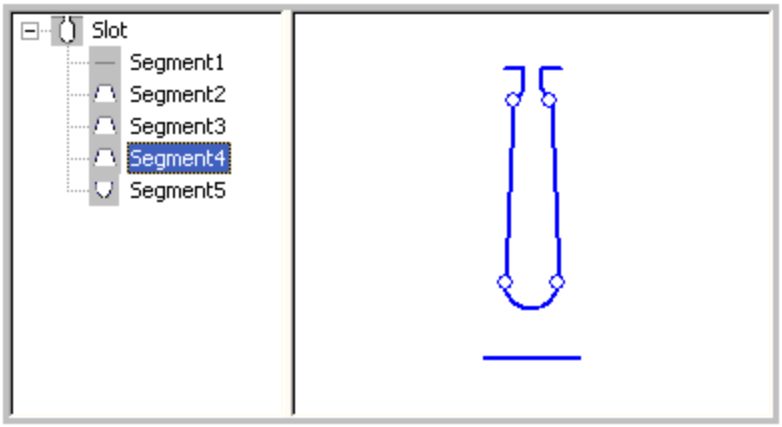


Slot Editor Data Editing View

In the tree view, the **Slot** root tree item is predefined and cannot be modified. The top segment in the tree, **Segment1**, cannot be deleted. In the slot data edit view, the slot segment tree items always follow in order from top to bottom of the slot. Segment names also follow this top-to-bottom order, **Segment1**, **Segment2**, **Segment3**, *etc.*, regardless of any operations that are made. Adjacent segments viewed in the tree are geometrically connected to each other as viewed on the slot graph. The data of neighboring segments are tightly coupled due to this geometry connection.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

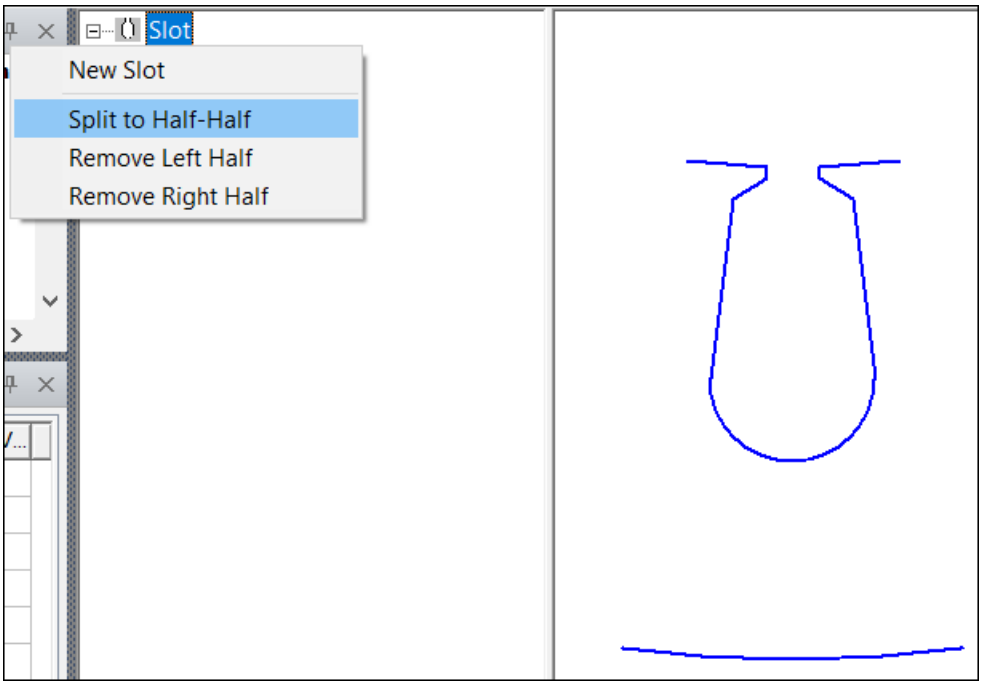
Selecting a segment in the tree highlights the corresponding segment (and its mirror image for symmetric slots) in the Slot Editor Graphic view – indicated by small open circles at the endpoints of the segments.



The Properties Window is also updated to show the selected segment's properties. Changing segment values in the slot editor updates the Properties Window, and vice-versa.

Editing Symmetric Slots

For a *symmetric* slot, right-clicking the **Slot** tree item displays the following choices:

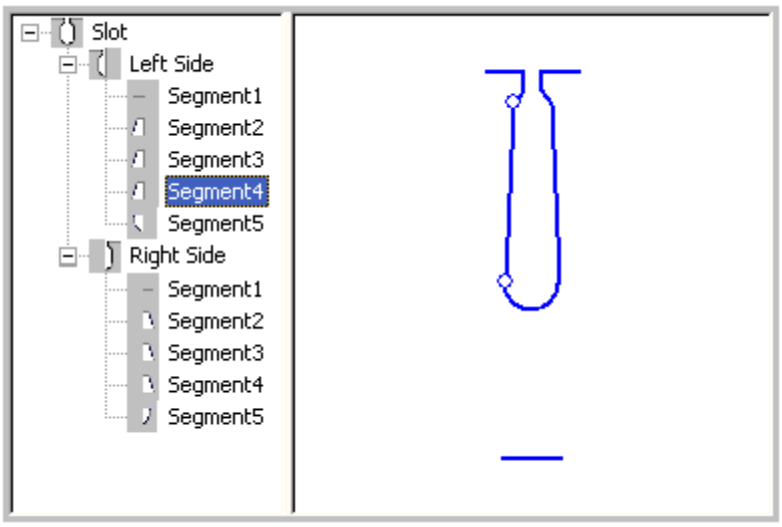


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **New Slot** - opens the **New Slot** dialog box in which the user can select a new standard slot.
- **Split to Half-Half** - splits the slot into a **Left Side** and a **Right Side** transforming it into an *antisymmetric* slot. Each side then can be edited independently.
- **Remove Left** (or **Right**) **Half**, **Remove Right Half** - removes the segments for the left (or right) half of the symmetric slot, transforming the slot into a *half-slot*.

Editing Antisymmetric Slots

For an *unsymmetric* slot type, two additional expandable sub-branches, **Left Side** and **Right Side**, are present in the slot tree. The segments in each side can be edited independently.



Right-clicking the **Slot** tree item displays the following choices:

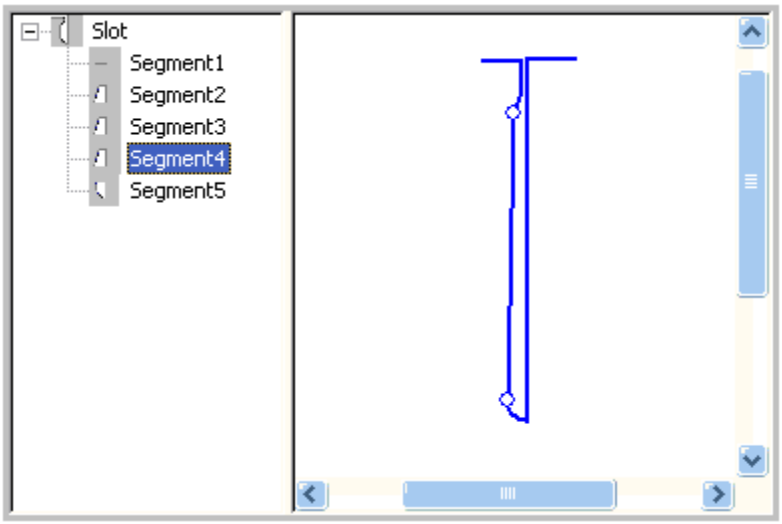
- **New Slot** - opens the **New Slot** dialog box in which the user can select a new standard slot.
- **Merge Left** (or **Right**) **to Symmetric** - the left (or right) side segments are mirrored and merged to form a *symmetric* slot.
- **Left Right Flip** - the left side and right side segments are flipped (reflected and interchanged). The slot remains antisymmetric.
- **Remove Left Half**, **Remove Right Half** - removes the segments for the left (or right) half of the symmetric slot, transforming the slot into a *half-slot*.

Right-clicking either the **Right Side** or **Left Side** sub-branch tree item opens a context menu on which you can choose either: **Merge to Symmetric**, which mirrors the selected side's segments and merges the segments into a *symmetric* slot; or **Remove**, which removes the selected side and all of its segments, resulting in a right or left half-slot.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Editing Half Slots

For a *half slot*, only the segments on one side of the slot can be edited.



Right-clicking the **Slot** tree item pops up a context menu displaying the following choices:

- **New Slot** - opens the **New Slot** dialog box in which the user can select a new standard slot.
- **Merge to Symmetric** - the left (or right) half-slot segments are mirrored and merged to form a symmetric slot.
- **Left Right Flip** - the left (or right) half-slot segments are flipped (reflected). The slot remains antisymmetric.
- **Mirror** - the left (or right) half-slot segments are mirrored (reflected and copied), transforming the slot into an *antisymmetric* slot.

The New Slot Dialog Box

Selecting **New Slot** in any of the **Slot** tree item context menus opens the **New Slot** dialog box.

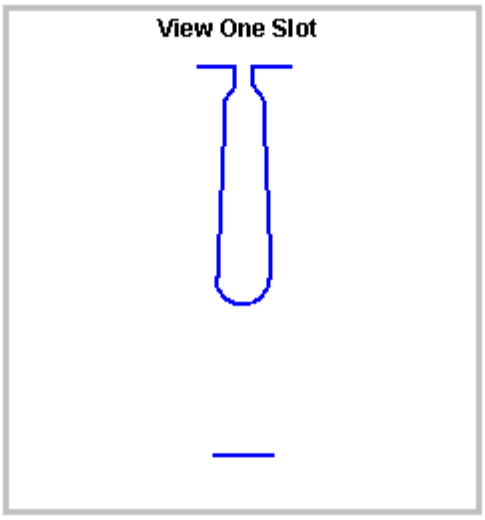
Radio buttons allow the user to select a **Symmetric Slot**, **Unsymmetrical Slot**, **Left Half Slot**, or **Right Half Slot** as the type to be added. Clicking **OK** confirms the choice and replaces the existing slot type in the editor with a slot of the selected type.

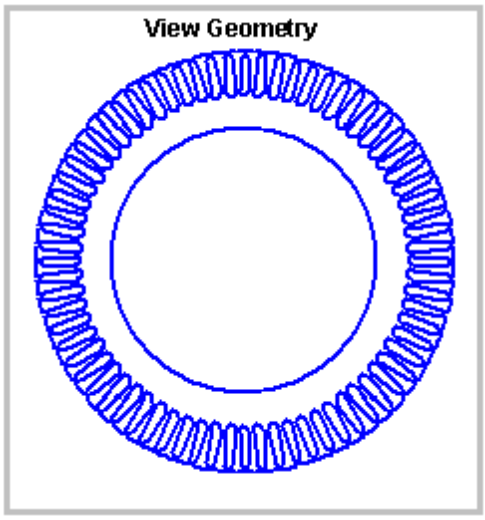
The new slot has only **Segment1** present. The user must then edit the slot segments to form the desired slot geometry.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Slot Editor Graphical View

The slot editor graphic view allows users either to **View One Slot** of the type currently being edited, or to **View** the **Geometry** of the machine stator or rotor with all of the slots in place.





Right-clicking anywhere in the graphic view pops up a context menu. In addition to changing the view, the menu provides commands to **Zoom In** and **Zoom Out**, to **Fit All** the view in the frame, as well as commands to **Insert**, **Append**, **Modify**, and **Remove** slot segments. These commands are discussed more fully in the section on Editing Slot Segments.

Editing Slot Segments

The RMxprt Slot Editor allows users to edit the segments that form the geometry of slots. In the slot data edit view, selecting a slot segment item on the tree, then right-clicking on it displays a context menu with commands that allow users to **Insert**, **Append**, **Modify**, and **Remove** segments.

Alternatively, users can select a slot segment directly in the graphic view, then right-click to bring up a context menu containing the same segment editing commands.

When a slot segment is selected, users can also modify the segment data directly in the desktop property window. Each edit triggers immediate validation. If edited values are improper, warning message windows are displayed describing the problem. Editing operations support Undo/Redo. Scripting functions are also available.

- **Insert Segment** – adds a segment *before* the currently selected segment. The added segment is defined by the user in the **Edit Slot Segment** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** You cannot insert a segment before **Segment1**.

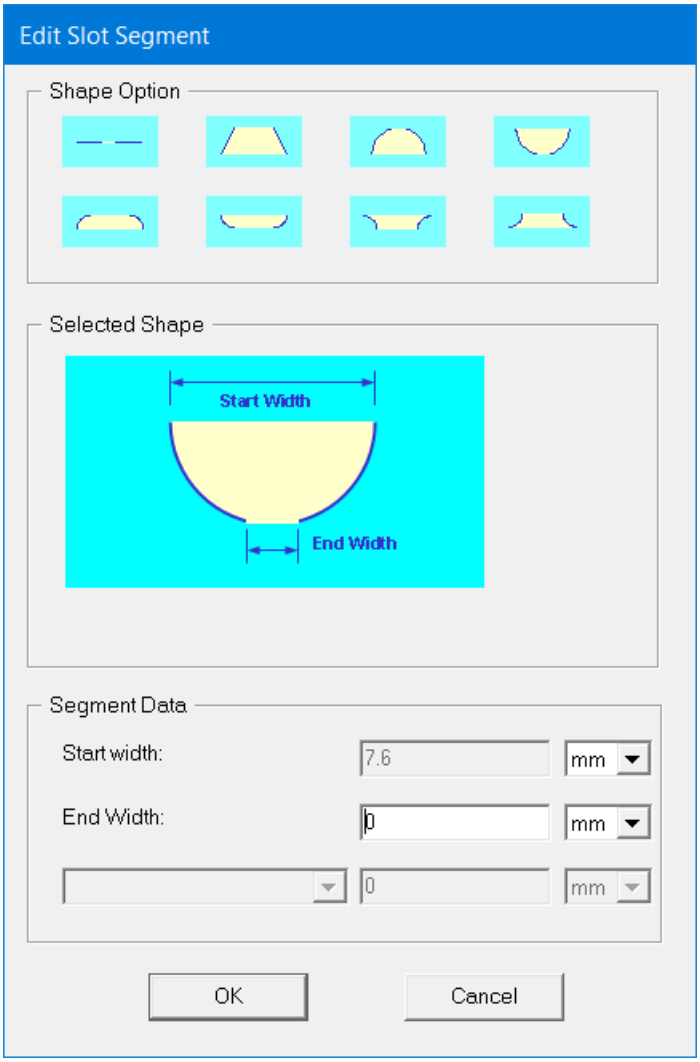
- **Append Segment** – adds a segment *after (i.e., below)* the currently selected segment. The appended segment is defined by the user in the **Edit Slot Segment** dialog box.
- **Modify Segment** – modifies the currently selected segment. The segment is modified by the user in the **Edit Slot Segment** dialog box.
- **Remove Segment** – removes the currently selected segment. The ends of the segments on either side of the removed segment are joined when the segment is removed.

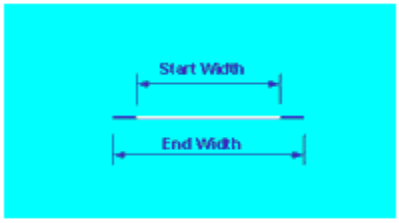
> **Note: Segment1** cannot be removed.

The Edit Slot Segment Dialog Box

In the **Edit Slot Segment** dialog box, user can define the shape and dimension(s) of the segment to be added, inserted, or appended relative to the currently selected segment in the slot data edit view. As shown below, eight basic geometric shapes are provided for defining the slot segment. The **Selected Shape** panel shows dimensional parameters that affect the segment shape such as height, starting and ending width, and radius. Parameters that determine the shape of the segment can be edited in the **Segment Data** panel.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





- **Start width** – sets the starting width and unit of measure. The value shown depends on the ending width of the previous segment and thus is not editable.
- **End width** – sets the ending width and unit of measure.

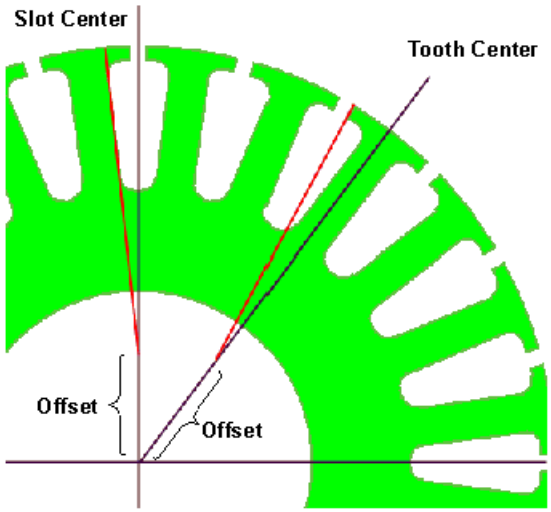
A drop-down box provides three additional options for controlling the segment shape:

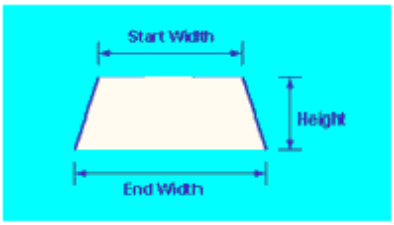
- **Line edge** (the default) – makes the segment a straight line.
- **Arc offset on slot center** – makes the segment an arc whose radius is determined by a user-

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

specified offset from the rotor/stator center with respect to the slot center.

- **Arc offset on tooth center** – makes the segment an arc whose radius is determined by a user-specified offset from the rotor/stator center with respect to the tooth center.





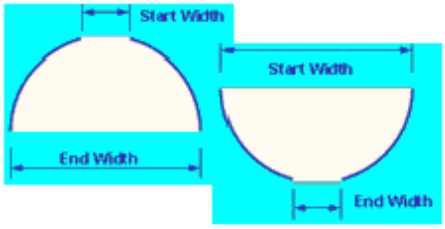
- **Start width** – sets the starting width and unit of measure. The value shown depends on the ending width of the previous segment and thus is not editable.
- **Height** – sets the height and unit of measure for the segment.

A drop-down box provides three options for controlling the segment shape:

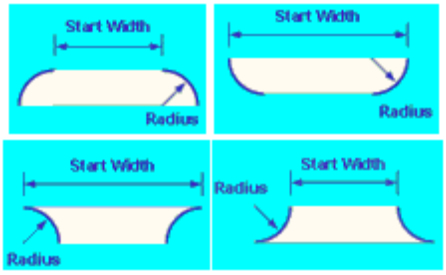
- **End width** (the default) – sets the ending width and unit of measure.
- **Parallel slot** – effectively sets the segment end width to the same value as the start width resulting in the slot sides being parallel to each other.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Parallel tooth** – sets the segment end width to a value such that the slot side is parallel to the adjacent slot side of the tooth formed between them.



- **Start width** – sets the starting width and unit of measure. The value shown depends on the ending width of the previous segment and thus is not editable.
- **End width** – sets the ending width and unit of measure.



- **Start width** – sets the starting width and unit of measure. The value shown depends on the ending width of the previous segment and thus is not editable.
- **Radius** – sets the radius and unit of measure for the segment.

Validation of the entered data is done when either the **OK** or **Preview** is clicked. If edited values are improper, warning message windows are displayed describing the problem. The **Preview** button allows users to preview the current changes in the slot graph window without confirming the changes.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clicking the **OK** button confirms the changes and closes the dialog box.

Clicking the **Cancel** button cancels the changes and closes the dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Working with Materials in RMxprt

RMxprt shares many common functions related to materials and material handling with **Maxwell 3D** and **Maxwell 2D**. For general topics such as, Assigning Materials, Searching for Materials, Adding New Materials, and Viewing and Modifying Material Attributes, see the topics in Maxwell help under **Assigning Materials**.

RMxprt includes a material library containing common materials used in motor design. However, this library needs to be configured so that it is automatically loaded for any new **RMxprt** design. Once the library is configured, you can add, remove, and edit RMxprt materials in two main ways:

- Using the **Tools** > **Edit Libraries** > **Materials** menu command.
- Right-clicking **Materials** in the project tree and selecting **Edit Library**.

Editing definitions from the project window does not modify the configured libraries for any particular design. To consider the current design, use the **Tools** > **Edit Libraries** option. Doing so ensures that new libraries are added to the configured list for the current design. If you edit materials from this command for the current and then export them, they will also be available to assign to objects in other designs.

Material Library Management for RMxprt

The three most crucial electromagnetic materials in the electric machine are soft-magnetic material (silicon steel sheet), hard-magnetic material (permanent magnet) and electromagnetic wire. It is convenient to create a data file library for them for quick selection while inputting design data.

Adding New Materials to an RMxprt Project

You can add a new material to a project or to the global user-defined material library. To make the new project material available to all projects, you must export the material to a global user-defined material library.

To assign a material to an object:

1. Click **Tools** > **Edit Libraries** > **Materials**.

   In the Project Manager tree, you can also right-click **Materials**, and select **Edit Library**.

   The **Edit Libraries** dialog box appears.

2. Click **Add Material**.

   The **View/Edit Material** dialog box appears.

3. Type a name for the new material in the **Material Name** text box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. In the **View/Edit Material for** section, select whether this material should be available for the **Active Design** only, for **This Product** only or for **All Products**. The selection makes a difference in the properties displayed.

   > **Note:** When you select certain **Type** or **Value** options, additional parameters appear beneath some properties in the same way that **Magnitude** appears beneath **Magnetic Coercivity**. As necessary, specify values for any additional parameters that appear.

5. In the **Properties of the Material** table, specify the **Type** and the **Value** for the following material properties, displayed when **Active Design** is selected:
   - Relative Permeability.
   - Bulk Conductivity
   - Magnetic Coercivity (including the Magnitude of the vector)
   - Core Loss Model - model selections for this property may enable access to coefficient calculation windows, and enable additional properties.
   - Mass Density

   If you select **This Product**, additional fields are displayed.

   - Relative Permittivity
   - Dielectric Loss Tangent
   - Magnetic Loss Tangent
   - Composition
6. Optionally, change the **Units** for any of the properties.
7. Click **Validate Material** to verify the settings you have specified are valid for the existing properties.
   If the material setup is valid, a green check mark appears below the **Validate Material** button.
8. Click **OK** to save the new material.
   The **Edit Libraries** dialog box reappears, with the new material added to the list of materials.
9. Click **OK** to close the **Edit Libraries** dialog box.

Soft-Magnetic Materials

The stator and the rotor iron cores in the electric machine are generally laminated with punched sheets of nonlinear soft-magnetic silicon steel. Some special types of electric machines, such as moment motor, turbo-generator etc., use integrated solid rotor iron core of soft-magnetic material. For magnetic field analysis and core loss analysis of the electric machine, the magnetization characteristics (**B-H Curve**) and the loss characteristics (**B-P Curve**) of the iron-core material must be defined. The dialog boxes to do so are accessed from the **View/Edit**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Material window, which, in turn is accessed from the **Edit Libraries** window. Access to the window for editing the B-H curve is enabled when you set the Magnetic Permeability value to nonlinear (rather than simple or Anisotropic). When you set the value to nonlinear, the value field changes to a B-H Curve button. Click the B-H curve button to open the window.

For the loss characteristics (B-P Curve), you first set the **Core Loss Model** of the material to **Electrical Steel** (rather than **None** or **Power Ferrite**) as a material property in the **View/Edit Material** window. When you do so, this enables the **Calculate Properties** selection for drop down at the bottom of the window. Select **Core Loss Coefficient** from the drop down menu to open the B-P Curve window.

Relative Permittivity for an RMxprt Material

Specify the following for Relative Permittivity and specify the units.

Type	Value
Simple	Type a value for the **Relative Permittivity**.
Anisotropic	The following two parameters appear: • T(1,1) • T(2,2) Type a simple value for each.
Nonlinear	The material's relative permittivity varies according to its associated D-E curve. **Note:** Not valid in RMxprt.

Note: This property is not used in RMxprt design analysis, but it will be transferred to a Maxwell 3D design automatically when the design is created by RMxprt.

Bulk Conductivity for an RMxprt Material

Specify the following for bulk conductivity and specify the units:

Type	Value
Simple	Type a value for the **Bulk Conductivity**.
Anisotropic	The following parameters appear: • T(1,1) • T(2,2) You can enter a simple value for each of these

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	parameters.
Nonlinear	The material's bulk conductivity varies according to its associated J-E curve.

Note: This property is not used in RMxprt design analysis, but it will be transferred to a Maxwell 3D design automatically when the design is created by RMxprt.

Dielectric Loss Tangent for RMxprt Material

Specify the following for dielectric loss tangent.

Type	Value
Simple	Type a value for the**Dielectric Loss tangent**.
Anisotropic	The following two parameters appear: • T(1,1) • T(2,2) Type a simple value for each.

Note: This property is not used in RMxprt design analysis, but it will be transferred to Maxwell 3D Design automatically when the design is created by RMxprt.

Magnetic Loss Tangent for RMxprt Material

Type	Value
Simple	Type a value for the **Bulk Conductivity**.
Anisotropic	The following two parameters appear: • T(1,1) • T(2,2) Type a simple value for each.

Note: This property is not used in RMxprt design analysis, but it will be transferred to a Maxwell 3D design automatically when the design is created by RMxprt.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Magnetic Coercivity for Maxwell and RMxprt Materials

Specify the following for magnetic coercivity and specify the units:

Type	Value
Vector	Appears by default. • If the Relative Permeability Type is either **Simple** or **Anisotropic**, enter a **Value** for the **Magnitude**. • If the Relative Permeability Type is **Nonlinear**, **Magnitude** becomes uneditable, and three additional fields of Type **Unit Vector**: **X Component**, **Y Component**, and **Z Component** appear in which you can enter values or specify functions.

Core Loss Model for an RMxprt Material

Specify the following for core loss type and specify the units:

Name	Value
None	No core loss is to be calculated for this material.
Electrical Steel	The following parameters appear: • **Kh** • **Kc** • **Ke** **Note:** Selecting **Electrical Steel** also enables the Calculate Properties for Core Loss Coefficient drop-down menu item at the bottom of the dialog box.
Power Ferrite	The following parameters appear: • **Cm** • **X** • **Y** **Note:** Selecting **Power Ferrite** also enables the Calculate Properties for Core Loss versus Frequency drop-down menu item at the bottom of the dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculating Properties for Core Loss in RMxprt (BP Curve)

To be able to extract parameters from the loss characteristics (B-P Curve), you first set the **Core Loss Model** of the material to Electrical Steel as a material property in the **View / EditMaterial** window.

To calculate core loss properties for an electrical steel material:

1. Click **Tools** > **Edit Libraries** > **Materials**.

   Or in the Project Manager tree, you can also right-click **Materials**, and select **Edit Library**.

   The **Edit Libraries** window appears.

2. Click **Add Material**.

   The **View/Edit Material** window appears.

3. In the **Core Loss Type** row, select **Electrical Steel** from the **Value** drop-down menu.

   This enables the **Calculate Properties for** drop-down menu at the bottom of the dialog box with the following two choices:

   - **Calculate Properties for: Core Loss at One Frequency**
   - **Calculate Properties for: Core Loss versus Frequency**

   It also displays parameters associated with Electrical Steel materials.

Electrical Steel Core Loss from a Single-Frequency Loss Curve

With the **Core Loss Type** set to Electrical Steel:

1. Select **Core Loss at One Frequency** from the **Calculate Properties for:** drop-down menu.

   The **BP Curve** window appears.

2. Do one of the following to specify a BP curve:
   - Import the curve from a saved file.
   - Enter the coordinates manually.

> **Note:** The accuracy in inputting the data for B-P Curve for the electrical steel material has significant effect on the correctness of the analyses to the electromagnetic characteristics of the electric machine. You should input the data for B-P Curve according to the accurate data provided by the manufacturers of materials.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select the units in which the B-P curve is measured from the **Core Loss Unit** drop-down menu.
4. Type values and select units for the following:
   - Mass Density
   - Frequency
   - Thickness
   - Conductivity

   The following parameters are dynamically updated with both the specified unit and the standard unit (w/m^3) as the input data changes.

   - **Kh** -Hysteresis
   - **Kc** - Classical Eddy
   - **Ke** - Excess
5. Click **OK**.

   The **View/Edit Material** window reappears. The property values fro **Kh**, **Kc**, and **Ke** are updated as new default values.

Computation of Core Loss from a Single-Frequency Loss Curve

The principles of the computation algorithm are summarized as follows.

The iron-core loss is expressed as

$$p_v = p_h + p_c + p_e = K_1 B_m^2 + K_2 B_m^{1.5}$$

where the eddy-current loss is

$$p_c = k_c (f B_m)^2$$

the hysteresis loss is

$$p_h = k_n f {B_m}^2$$

and the excessive loss is

$$p_e = k_e (f B_m)^{1.5}$$

Therefore,

$$K_1 = k_h f + K_c f^2$$

$$K_2 = k_e f^{1.5}$$

The classical eddy-current loss coefficient is calculated directly as

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$k_c = \pi^2 \sigma \frac{d^2}{\delta}$$

where σ is the conductivity and *d* is the thickness of one lamination sheets.

Minimize the quadratic form to obtain K_1 and K_2.

$$err\left(K_1, K_2\right) = \sum_i \left[p_{vi} - \left(K_1 B_{mi}^2 + K_2 B_{mi}^{1.5}\right)\right]^2 = min$$

where P_{vi}, B_{mi} – the i-th point of the data on the loss characteristics curve.

The other two loss coefficients are obtained as

$$k_h = \frac{K_1 - k_c f_0^2}{f_0}$$

$$k_e = \frac{K_2}{f_0^{1.5}}$$

where f_0 is the testing frequency for **B-H Curve**.

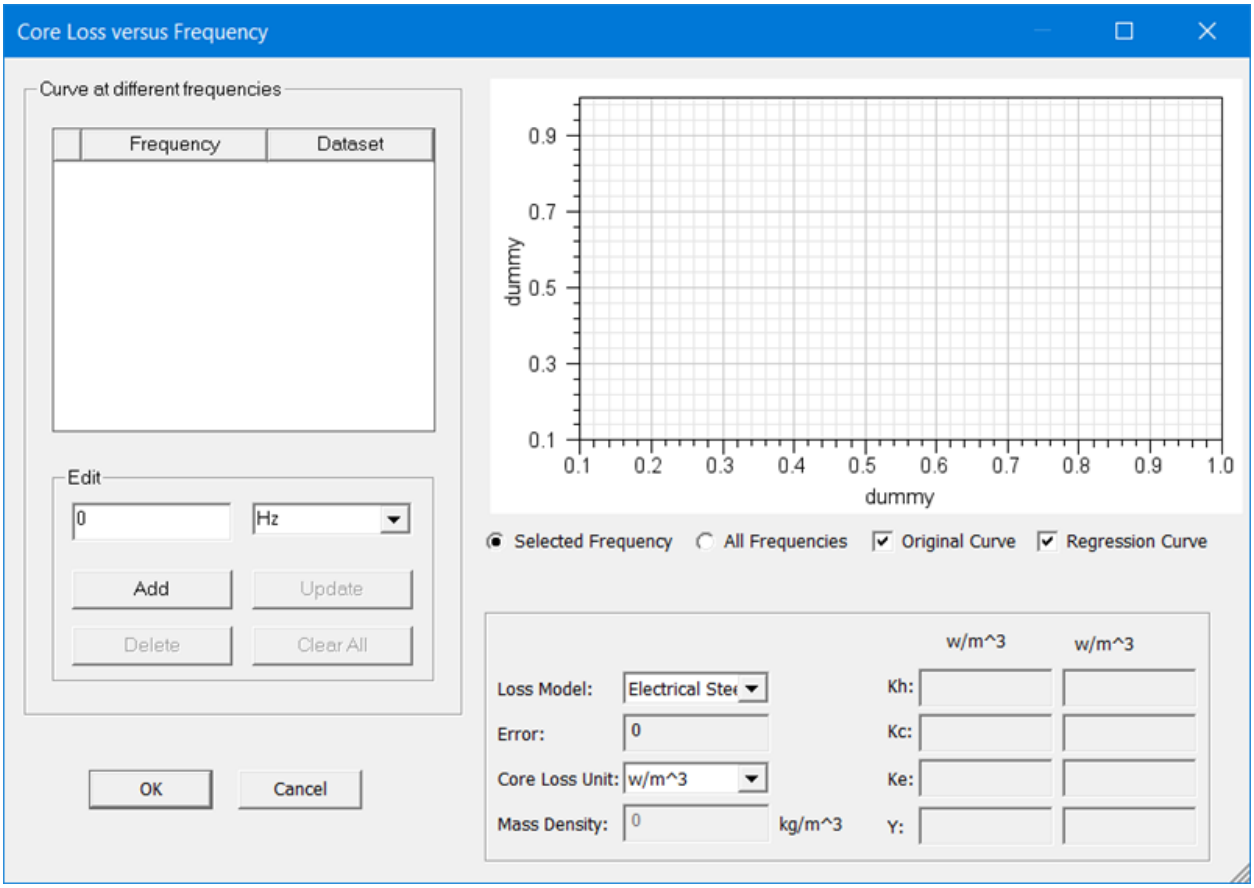
Electrical Steel Core Loss from Multi-Frequency Loss Curves

With the **Core Loss Type** set to Electrical Steel:

1. Select **Core Loss versus Frequency** from the **Calculate Properties for:** drop-down menu.

   The **Core Loss versus Frequency** window appears as shown:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Using the Edit area, **Add** frequency points at which a dataset is available for the Core Loss.
3. For each Dataset added, click the **Edit Dataset** button to launch the Edit Dataset dialog box.
4. Add Dataset information for the frequency by manually entering the data or importing the data from a table. Click **OK** to accept the dataset and return to the **Core Loss versus Frequency** window. Continue adding dataset information until all frequencies have datasets defined.
5. Click **OK** when all frequencies have valid data to complete the core loss calculation and return to the **View/Edit Material** window.

Computation of Core Loss from Multi-Frequency Loss Curves

The principles of the computation algorithm are summarized as follows.

The iron-core loss is expressed as

$$p_v = p_h + p_c + p_e = k_h f B_m^2 + k_c (f B_m)^2 + k_e (f B_m)^{1.5}$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Minimize the quadratic form to obtain k_h , k_c and k_e directly:

$$err\left(k_h, k_c, k_e\right) = \sum_{i=1}^{m} \sum_{j=1}^{n_i} \left[p_{vij} - \left(k_h f_i B_{mij}^2 + k_c f_i^2 B_{mij}^2 + k_e f_i^{1.5} B_{mij}^{1.5}\right)\right]^2 = min$$

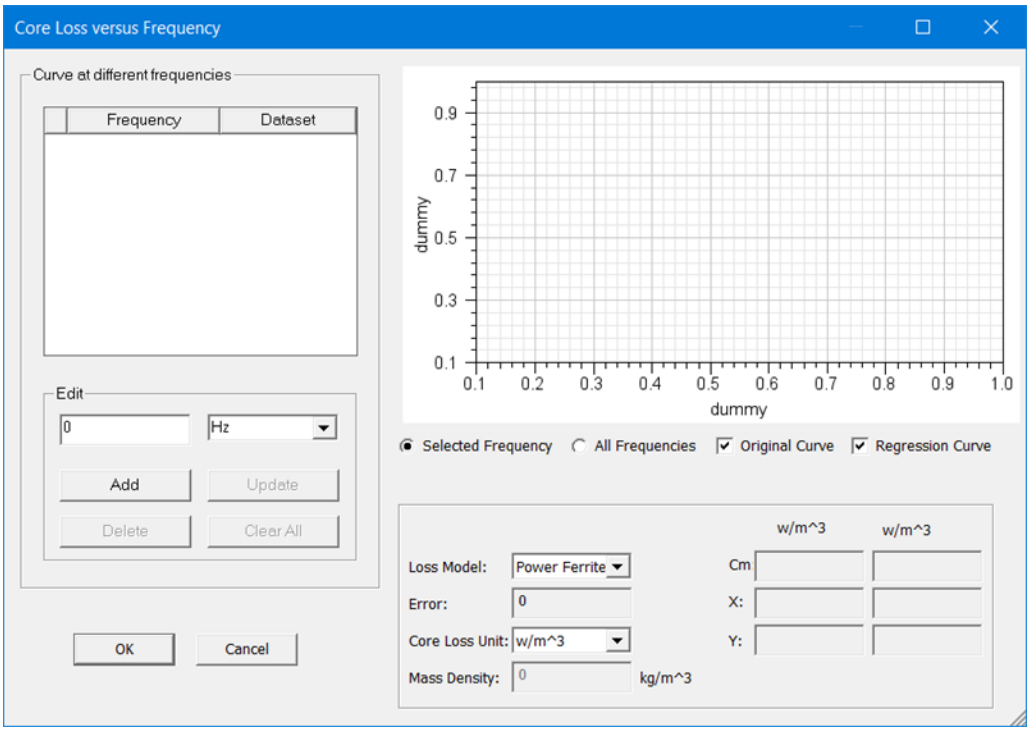
where m - the number of loss curves, n_i - the number of points of the i-th loss curve, and P_{vij} = f(f_i , B_{mij}) - two-dimensional lookup table for multifrequency loss curves.

Power Ferrite Core Loss from Multi-Frequency Loss Curves

With the **Core Loss Type** set to Power Ferrite:

1. Select **Core Loss versus Frequency** from the **Calculate Properties for:** drop-down menu.

   The **Core Loss versus Frequency** window appears as shown:



2. Using the Edit area, **Add** frequency points at which a dataset is available for the Core Loss.
3. For each Dataset added, click the **Edit Dataset** button to launch the Edit Dataset dialog box.
4. Add Dataset information for the frequency by manually entering the data or importing the data from a table. Click **OK** to accept the dataset and return to the **Core Loss versus**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Frequency window. Continue adding dataset information until all frequencies have datasets defined.

5. Click **OK** when all frequencies have valid data to complete the core loss calculation and return to the **View/Edit Material** window.

Computation of Power Ferrite Core Loss from Loss Curves

The principles of the computation algorithm are summarized as follows.

The iron-core loss is expressed as

$$p_v = C_m f^x B_m^y$$

or

$$\mathbf{log}\,(p_v) = c + x \cdot \mathbf{log}\,(f) + y \cdot \mathbf{log}(B_m)$$

where

$$c = \mathbf{log}(C_m)$$

Minimize the quadratic form to obtain *c*, *x* and *y:*

$$err\left(c, x, y\right) = \sum_{i=1}^{m} \sum_{j=1}^{n_i} \left[\mathbf{log}\left(p_{vij}\right) - \left(c + x \cdot \mathbf{log}\left(f_i\right) + y \cdot \mathbf{log}\left(B_{mij}\right)\right)\right]^2 = min$$

where m - the number of loss curves, n_i - the number of points of the i-th loss curve, and $P_{vij} = f(f_i, B_{mij})$ - two dimensional lookup table for multi-frequency loss curves. Then C_m is calculated from the equation above.

Mass Density for RMxprt Material

Mass density for RMxprt materials is measured in kg/m^3.

Composition for RMxprt Material

RMxprt material has the composition of **Solid** or **Lamination**.

If **Lamination**, specify the following:

- **Stacking Factor** - takes a simple value
- **Stacking Direction** - has a drop down menu lets you select V(1), V(2), or V(3).

> **Note:** This property is not used in RMxprt design analysis, but it will be transferred to a Maxwell 3D design automatically when the design is created by RMxprt.

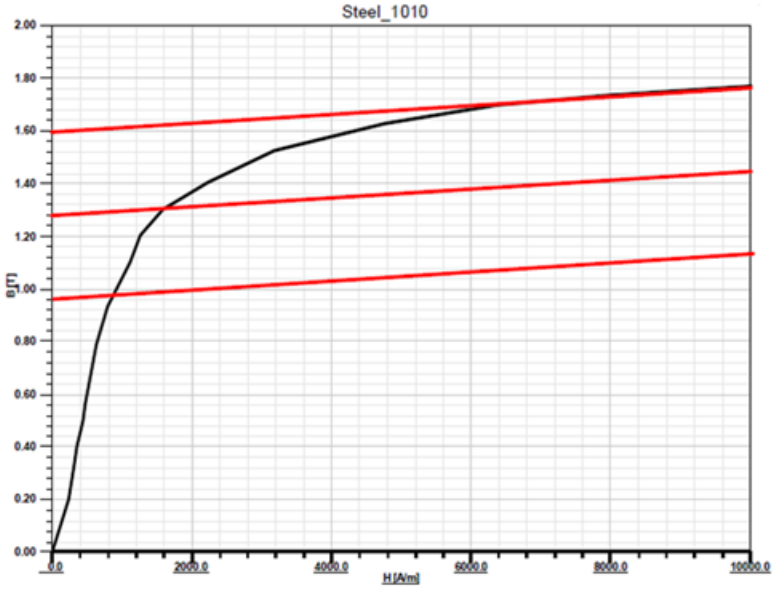
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Permanent Magnet Materials in RMxprt

A permanent magnet is defined as a material that generates a magnetic flux due to permanent magnetic dipoles in that material.

Compute Remanent B_r from B-H Curve

The value of the remanent B_r of the individual element after the magnetization field is computed is determined in such a way: after having located the operating point on the original non-remanent B-H curve, draw a line which is parallel to the original recoil curve with the slope of $\mu_0\mu_r$ and passes the operating point, the intersection of this line with B-axis is the remanent B_r as the result of the applied magnetization field.



Calculating the Properties for a Nonlinear Permanent Magnet in RMxprt

Nonlinear permanent magnet properties may be specified in one of two ways.

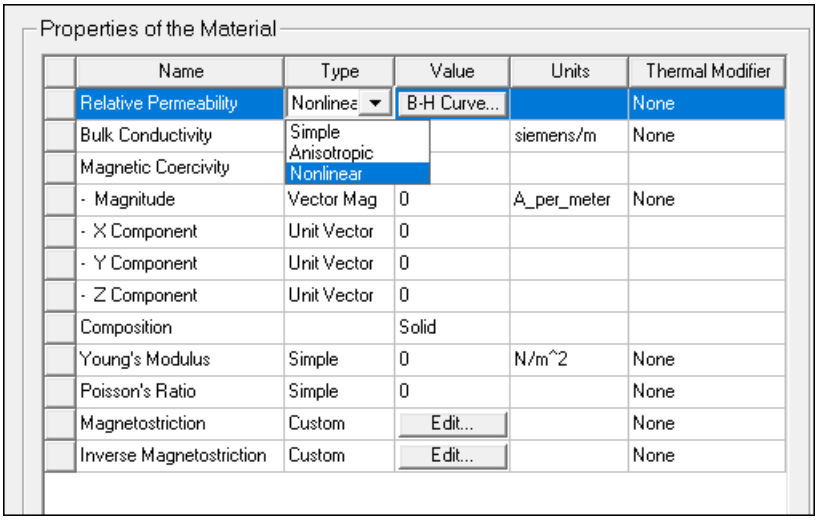
First, a B-H curve may be input directly as follows:

1. Click the **View/Edit Materials…** button in the **Edit Libraries** window.

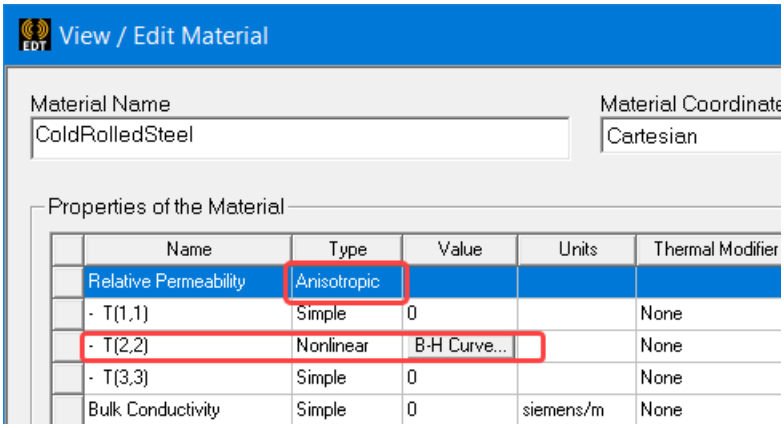
   The **View/Edit Material** window appears.

2. The nonlinear B-H curve is defined by setting the **Relative Permeability** **Type** either to **Nonlinear** or **Anisotropic.**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you select Anisotropic, each of its components can be selected **Nonlinear** and can be specified by a **B-H Curve**.

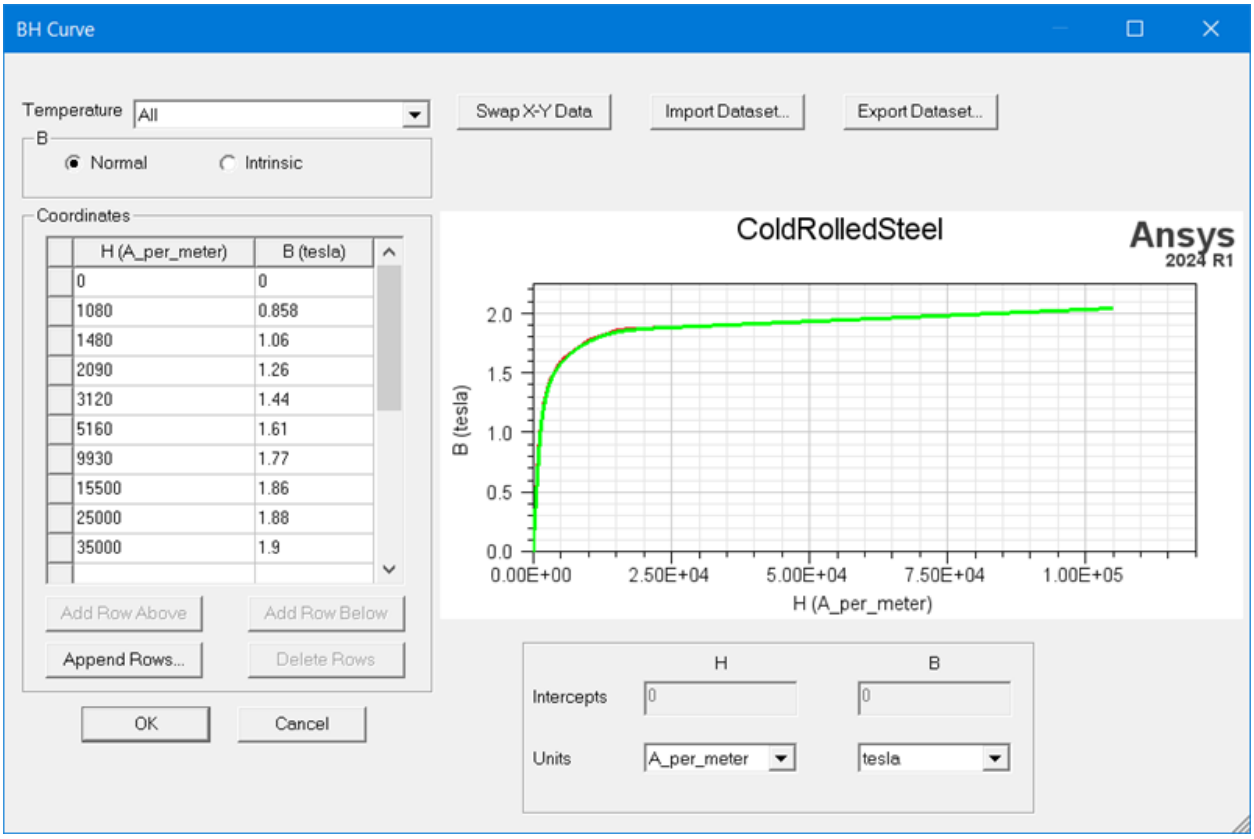


A **B-H Curve** button appears in the nonlinear property's **Value** column.

3. Input the B-H curve by clicking the **B-H Curve** button in the property **Value** column.

   This opens the **BH Curve** dialog box in which you can input (or modify) curve data. (Refer to Adding Datasets for general information on working with datasets.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- For a **Normal** BH curve, the slope of the curve can not be less than that of free space anywhere along the curve.
- For an **Intrinsic** BH curve, the slope of the curve can not be less than 0.

Note:

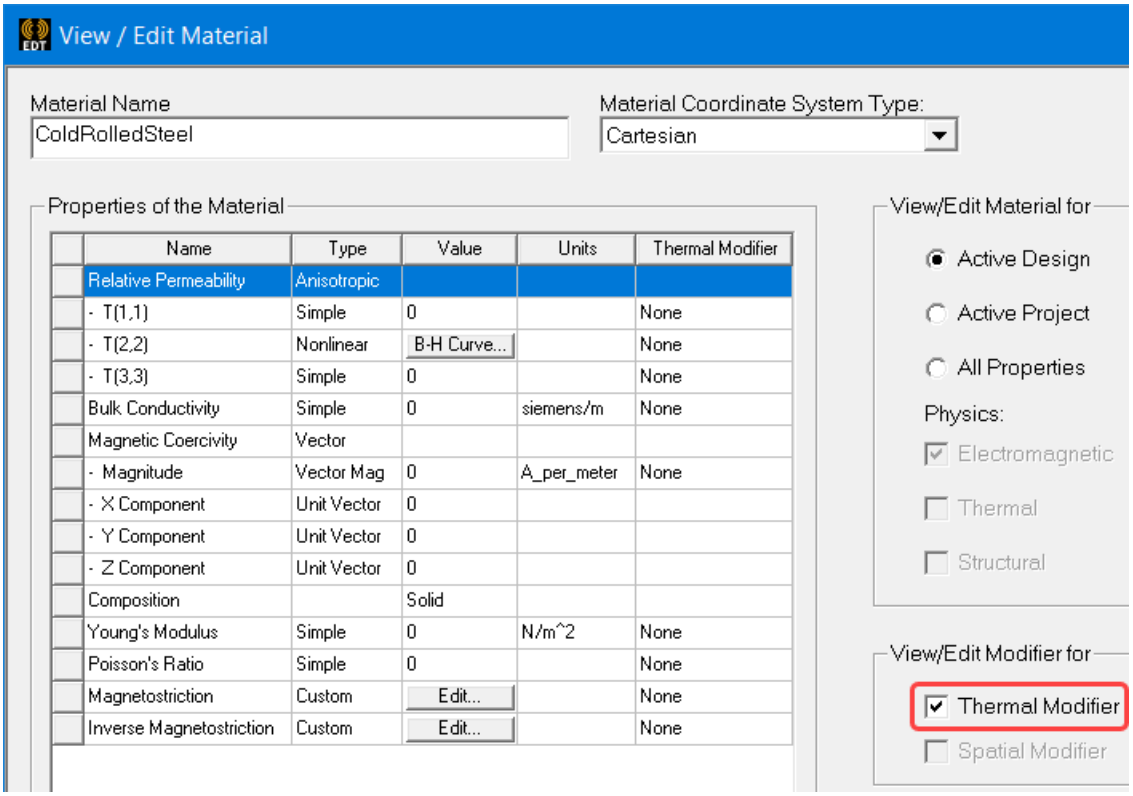
- The **Intrinsic** B-H curve is supported only in Maxwell 2D/3D magnetostatic and transient design types. A material property defined using an Intrinsic B-H curve will fail the validation check in all the other product/design types.
- When an **Intrinsic** B-H curve is added, the **Relative Permeability Value** button label changes to **Bi-H Curve** as visual indication of the type of curve currently defined for the material.

When you click **OK** in the dialog box, an error message displays if a slope is out of tolerance, identifying the data points between which the slope is less than that of free space.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The operations to input a nonlinear demagnetization curve are the same as entering a B-H curve for Steel material. When a B-H curve goes through the second quadrant or third quadrant, the curve is treated as a demagnetization curve.

4. To model *temperature dependency* for a nonlinear permanent magnet you must
   - use an **Intrinsic** B-H curve to model the **Relative Permeability**.
   - Specify a Thermal Modifier for both **Relative Permeability** and the **Magnitude** of **Magnetic Coercivity**. Apply a thermal Modifier by selecting the **Thermal Modifier** check box. Checking this box causes the **Thermal Modifier** column to display at the right side of the **Properties of the Material** table. Selecting **Edit** rather than **None** causes display of the **Edit Thermal Modifier** dialog box.



Alternatively, a nonlinear B-H curve can be modeled by the following four parameters:

- residual flux density B_r
- coercive field force H_c
- maximum energy product $(BH)_{max}$
- relative recoil permeability μ_r

From the **View/Edit Materials** window:

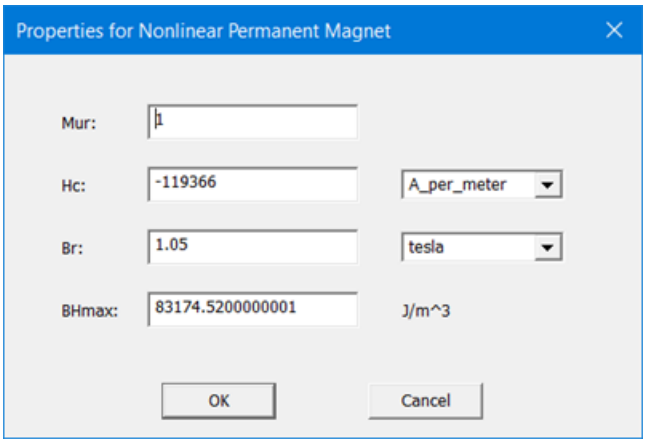
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Set the **Relative Permeability** to **Nonlinear**.

   This enables the **Calculate Properties for...** drop-down menu at the bottom of the window.

2. Click **Nonlinear Permanent Magnet** from the drop-down menu.

   This displays the **Properties for Nonlinear Permanent Magnet** dialog box, which contains the following fields into which you enter the appropriate values.



Mur	Provide a value for relative permeability.
Hc	Coercive field force H_c in the units specified. Provide a value and select units from the drop down menu.
Br	Residual flux density B_r in Tesla. If enabled, provide a value and select units from the drop down menu.
BH max	Maximum magnetic energy product $(BH)_{max}$ If enabled, provide a value and select units form the drop down menu.

3. Click **OK** to close the dialog and return to the **View/Edit Materials** window.

The values for **Relative Permeability** and **Magnitude** under **Magnetic Coercivity** are updated as new default values. Maxwell will create a look-up table based on the Four-Parameter Curve Fitting algorithm, and update the coordinates of the B-H curve automatically as long as the input data of four parameters pass the validation check.

Calculating the Properties for a Linear Permanent Magnet

Edit a linear demagnetization curve is simple. From the **View/Edit Materials** window:

1. Set the **Relative Permeability** to **Simple**.

   This enables the **Calculate Properties for Permanent Magnet** drop-down menu at the bottom of the window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **Permanent Magnet** from the drop-down menu.

   This displays the **Properties for Permanent Magnet** window. This contains the following fields.

Mu (enabled by default)	Provide a value.
Hc (enabled by default)	Coercive field force H_c in the units specified. Provide a value and select units from the drop down menu.
Br/Mp (disabled by default)	Checking this enables the radio buttons that let you specify either Br or Mp.
Br	Residual flux density B_r, in Tesla. If enabled, provide a value and select units from the drop down menu.
Mp	If enabled, provide a value and select units form the drop down menu.

3. Click **OK** to close the dialog and return to the **View/Edit Materials** window.

The values for **Relative Permeability** and **Magnitude** under **Magnetic Coercivity** are updated as new default values.

> **Note:** The accuracy in inputting the characteristic parameters for the permanent-magnetic material has significant effect on the correctness of the analyses to the electromagnetic characteristics of the electric machine. It is suggested that users should input the characteristic parameters of the permanent-magnetic material according to the accurate data provided by the manufacturers of materials. **RMxprt** provides a few characteristic parameters of permanent-magnetic materials for reference.

Using Demagnetization Curves

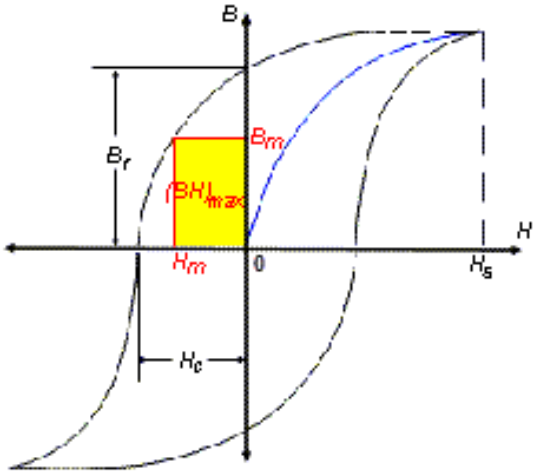
Many permanent magnet manufactures directly provide demagnetization curves for their products, but in most cases, manufactures provide some main parameters, such as residual flux density *Br*, coercive field force *Hc* and maximum magnetic energy product $(BH)_{max}$, and relative recoil magnetic permeability μ_r. This section and the next section describe the basic parameters for the demagnetization curve of permanent magnets and the curve fitting based on these parameters.

Hysteresis Loop

The permanent-magnetic material belongs to hard-magnetic material. It is characterized with "fat" hysteresis loop, which encloses large area as shown in the figure. When magnetized, it

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

keeps high magnetic property with the external magnetic field removed, therefore is used in the permanent-magnet electric machine to produce magnetic field. The characteristics of the permanent-magnetic material are represented with its main parameters: residual flux density B_r, coercive field force H_c and maximum magnetic energy product $(BH)_{max}$.

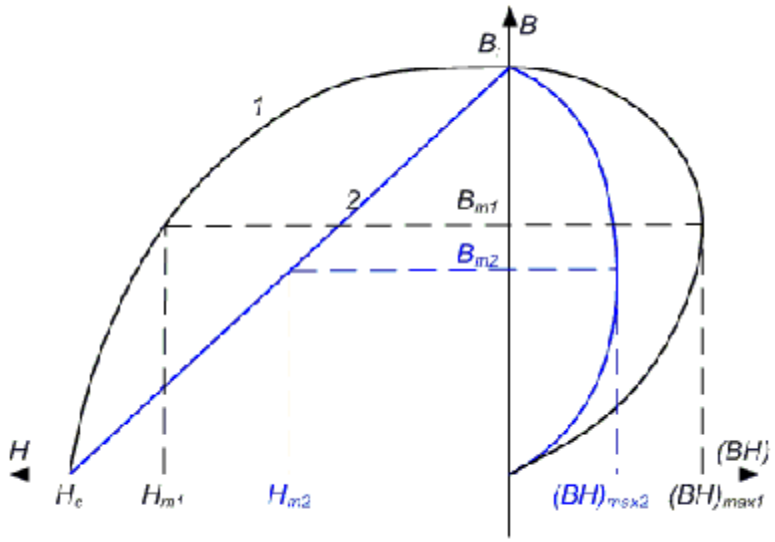


For materials with nonlinear B-H curve, the hysteresis core loss can be modeled by a hysteresis loop. In Maxwell, the hysteresis loop can be created from a normal B-H curve (starts at (0,0)) and hysteresis parameters (H_c, B_r).

Demagnetization Curve

The part of the maximum hysteresis loop of the permanent-magnetic material in the second quadrant is called the demagnetization curve as shown in the next figure. It is the basic characteristics curve of the permanent-magnetic material. On the demagnetization curve, the magnetic flux density has positive values, but the magnetic field intensity has negative values. It means that the permanent-magnetic material is applied with demagnetization magnetic field intensity. Because H_m and B_m are in opposite directions, when the magnetic flux goes through the permanent-magnetic material, the magnetic potential difference along the direction of the magnetic flux does not drop, but rises. Therefore, the permanent-magnetic material is a magnetic source, similar to the electric source in the electric circuit.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

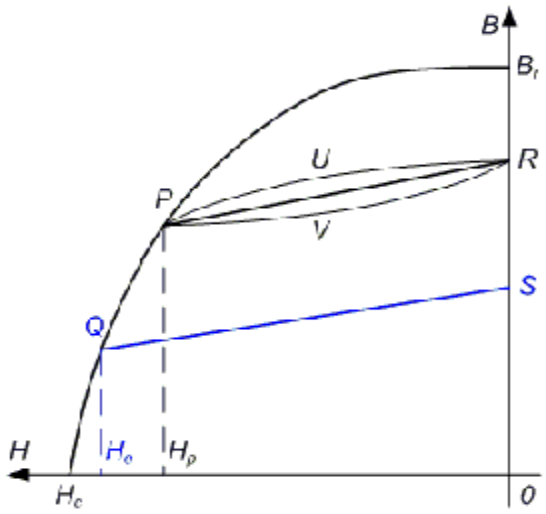


The two extreme positions on the demagnetization curve are the two significant parameters to represent the magnetic characteristics of the permanent-magnetic material. On the demagnetization curve, the value of the magnetic flux density corresponding to zero magnetic field intensity H is termed residual flux density B_r, on the other hand, the value of the magnetic field intensity corresponding to zero magnetic flux density B is termed coercive field force H_c. The produce of the magnetic flux density and the magnetic field intensity at any point on the demagnetization curve is termed magnetic energy product (BH), which is proportional to the magnetic energy density possessed by the permanent magnet at the given operating situation. At the two extreme positions ($B = B_r$, $H = 0$) and ($B = 0$, $H = H_c$), the magnetic energy product is equal to zero. Somewhere at an intermediate position, the magnetic energy reaches its maximum value and is termed maximum magnetic energy product $(BH)_{max}$, which is another significant parameter to represent the magnetic characteristics of the permanent-magnetic material. To some permanent-magnetic materials with linear demagnetization curve, it is obvious that at ($B = B_r / 2$, $H = H_c / 2$), the magnetic energy product reaches its maximum value, i.e. $(BH)_{max} = B_r H_c / 4$.

Recoil Lines

The relationship between the magnetic flux density and the magnetic field intensity represented by the demagnetization curve only exists when the magnetic field intensity varies in the same direction. In fact, when the permanent magnet electric machine is working, the demagnetization field intensity varies repeatedly in both directions. When demagnetization field is applied to the magnetized permanent magnet, the magnetic flux density decreases along the curved segment B_rP on the demagnetization curve as shown in the figure:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If the external demagnetization field intensity H_p is removed when the magnetic flux density reaches the point *P*, the magnetic flux density will increase not along the original demagnetization curve, but along another curve *PVR*. If the external demagnetization field intensity is reapplied, the magnetic flux density will decrease along the new curve *RUP*. By repeatedly applying the demagnetization field intensity, a localized loop is formed and is termed local hysteresis loop. On the local hysteresis loop, the rising segment and the dropping segment are quite close to each other, therefore can be approximated by the straight line *PR*, which is termed recoil line with *P* as the starting point. If demagnetization field with intensity H_q not exceeding the original value H_p is applied thereafter, the magnetic flux density will vary reversibly along the recoil line *PR*. If $H_q > H_p$, the magnetic flux density drops to a new starting point *Q*. It will vary along the new recoil line *QS*, but not the previous one *PR*. This sort of irreversible variation in magnetic flux density causes instability in the characteristics of electric machines and complicates the design computation of permanent magnet electric machines, therefore should be avoided as possible.

Recoil Magnetic Permeability

The ratio of the average slope of the recoil line to the magnetic permeability in vacuum μ_0 ($\mu_0 = 4 \times 10^{-7}$ H/m) is termed relative recoil magnetic permeability or recoil magnetic permeability for short, μ_r:

$$\mu_r = \frac{1}{\mu_0} \left| \frac{\Delta B}{\Delta H} \right|$$

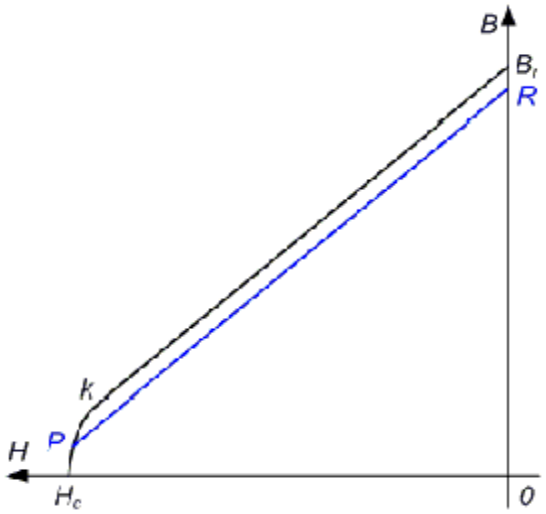
If the demagnetization curve is curly, the value of μ_r depends on the location of the starting point and is a variable, but typically varies within a small range. Therefore, it is approximated as a constant and is taken as the slop of the tangent to the point (B_r, 0) on the demagnetization curve.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In other words, the recoil lines at different starting points are approximated as a family of parallel lines, which are all parallel to the tangent to the point (B_r, 0) on the demagnetization curve.

Inflection Point

Some permanent-magnetic materials, such as some ferrite permanent-magnetic materials, show straight upper segment on the demagnetization curve. When the demagnetization field intensity drops to a given value, the demagnetization curve turns to decrease rapidly. The turning point is termed inflection point. If the demagnetization field intensity does not exceed the inflection point *k*, the recoil line coincides with the straight segment on the demagnetization curve. If the demagnetization field intensity exceeds the inflection point *k*, the new recoil line *RP* no longer coincides with the straight segment on the demagnetization curve.



Some permanent-magnetic materials, such as most of the rare-earth permanent-magnetic materials, show straight demagnetization curve in the whole range. In those cases, the recoil line coincides with the demagnetization curve. This makes the magnetic property keep stable while the permanent-magnet electric machine is working. This is the best ideal demagnetization curve for electric machine applications.

Curve Fitting of Demagnetization Curves

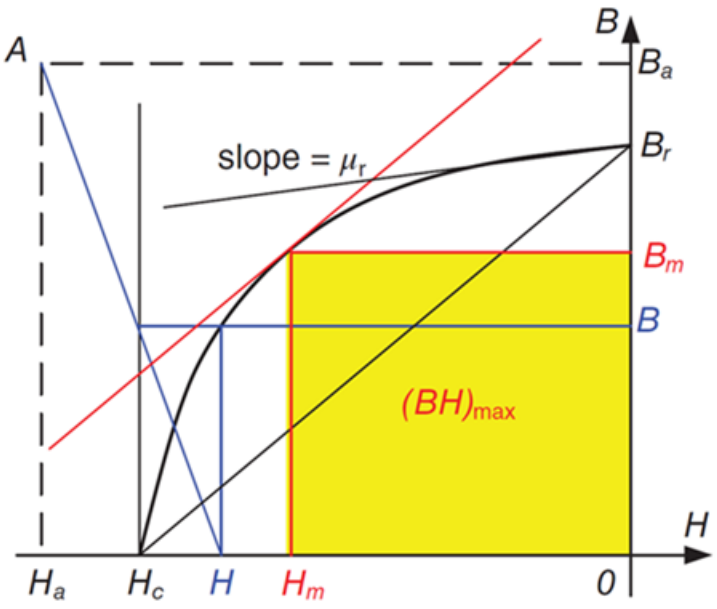
RMxprt fits the demagnetization curve according to the given characteristic parameters B_r, H_c, $(BH)_{max}$ and μ_r.

Three-Parameter Curve Fitting

Given the three characteristic parameters *Br*, *Hc* and $(BH)_{max}$, the principles of the three-parameter curve fitting algorithm are summarized as follows. Using the following figure as a

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

reference:



$$H_a = \frac{H_c}{a}$$

and

$$B_a = \frac{B_r}{a}$$

where $a < 1$.

Any magnetic flux density B in the interval $0 \le B \le B_r$ corresponds to the magnetic field intensity H:

$$H = -H_c + \frac{H_a - H_c}{B_a - B} B = -H_c + \frac{H_c(1-a)}{B_r - aB} B = -H_c \frac{B_r - B}{B_r - aB}$$

The tangent at any point is given by

$$\frac{dB}{dH} = \left(\frac{dH}{dB}\right)^{-1} = \frac{\left(1 - a\frac{B}{B_r}\right)^2}{1-a} \frac{B_r}{H_c}$$

The magnetic flux density B_m and the magnetic field intensity H_m corresponding to the maximum magnetic energy product satisfy the following relationship:

$$\left.\frac{dB}{dH}\right|_{B=B_m} = \frac{B_r}{H_c}$$

Solving yields

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$B_m = \frac{B_r}{1+\sqrt{1-a}} \quad \text{and} \quad H_m = -\frac{H_c}{1+\sqrt{1-a}}$$

Let the magnetic energy product at the point equal to $(BH)_{max}$ be

$$|B_m H_m| = \frac{B_r H_c}{\left(1+\sqrt{1-a}\right)^2} = (BH)_{max}$$

Solving yields

$$a = 2\sqrt{\left|\frac{B_r H_c}{(BH)_{max}}\right|} - \left|\frac{B_r H_c}{(BH)_{max}}\right|$$

The relative recoil magnetic permeability μ_r is calculated as

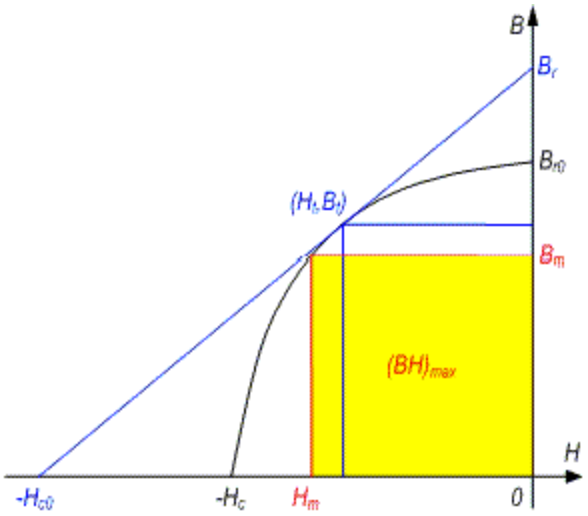
$$\mu_r = \frac{1}{\mu_0}\frac{dB}{dH}\Big|_{B=B_r} = \left(1-a\right)\frac{B_r}{\mu_0 H_c}$$

Four-Parameter Curve Fitting

The three-parameter curve fitting technique fits the demagnetization curve well. For the nonlinear permanent-magnetic material, the real operating point lies often not on the demagnetization curve, but on the recoil line. The relative recoil magnetic permeability calculated with the three-parameter curve fitting technique will cause deviation, therefore RMxprt employs a more accurate fitting technique: four-parameter curve fitting technique, as introduced below.

Given the four characteristic parameters B_r, H_c, $(BH)_{max}$ and μ_r, the principles of the four-parameter curve fitting algorithm are summarized as follows:

1. Draw a line through the point (0, B_r) with the slope equal to $-\mu_r\mu_0$ as shown in the figure. The segment of this line in the second quadrant is termed the ideal recoil line.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Find the virtual magnetic flux density B_{r0}.
3. With B_{r0}, H_c, and $(BH)_{max}$, draw the demagnetization curve with the three-parameter curve fitting technique. The curve should touch the ideal recoil line at the tangent point (H_t, B_t).
4. Any magnetic flux density B in the interval $0 \le B \le B_r$ corresponds to the magnetic field intensity H:

$$H = \begin{cases} -H_c \frac{B_{r0} - B}{B_{r0} - a_0 B} & B \le B_t \\ H_t + \frac{B - B_t}{\mu_r \mu_0} & B \ge B_t \end{cases}$$

5. The virtual magnetic flux density B_{r0} is found by iteration:
   a. Start from the initial guess for the lower and the upper bounds for the virtual magnetic flux density B_{r0}:

$$B_0 = \max\left(\mu_r \mu_0 H_c, \frac{(BH)_{max}}{H_c}\right)$$

$$B_1 = B_r$$

   b. Let

$$B_{r0} = \frac{B_0 + B_1}{2}$$

6. With B_{r0}, H_c, and $(BH)_{max}$, draw the demagnetization curve with the three-parameter curve fitting technique.

$$a_0 = 2\sqrt{\left|\frac{B_{r0} H_c}{(BH)_{max}}\right|} - \left|\frac{B_{r0} H_c}{(BH)_{max}}\right|$$

7. The curve should touch a line parallel to the ideal recoil line at the tangent point (H_t, B_t).

$$B_t = \begin{cases} B_{r0} & a_0 = 0 \\ \frac{B_{r0} - \sqrt{\mu_r \mu_0 H_c (1 - a_0) B_{r0}}}{a_0} & a_0 > 0 \end{cases}$$

   and

$$H_t = -H_c + \frac{B_{r0} - B_t}{B_{r0} - a_0 B_t}$$

8. For any magnetic flux density B in the interval $0 \le B \le B_r$, the corresponding magnetic field intensity H will be calculated by

$$H = \begin{cases} -H_c \frac{B_{r0} - B}{B_{r0} - a_0 B} & B \le B_t \\ H_t + \frac{B - B_t}{\mu_r \mu_0} & B \ge B_t \end{cases}$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. Calculate the value of H_r corresponding to B_r using:

$$H_r = H_t + \frac{B_r - B_t}{\mu_r - \mu_0}$$

10. If $H_r > 0$, the assumed virtual B_{r0} is too small, the lower bound of the interval needs to be increased, so let $B_0 = B_{r0}$. If, however, $H_r < 0$, the assumed B_{r0} is too big, the upper bound of the interval needs to be decreased, so let $B_1 = B_{r0}$.
11. Repeat steps until H_r converges to 0 within satisfactory precision.

Conductor Data

When a material is assigned to a machine part, RMxprt checks that the material is appropriate for the machine part based upon whether it is a conductor or other material type. RMxprt distinguishes conductors based on material threshold settings. RMxprt's treatment of conductors may be set by the used by adjusting the **Material Threshold**.

Editing Conductivity Properties in RMxprt

1. Click **Tools** > **Edit Libraries** > **Materials** to open the **Edit Libraries** window.
2. Select the material in the list whose conductivity properties you wish to edit and click the **View/Edit Materials** button.
3. In the **View/Edit Materials** window, **Bulk Conductivity** has two property types in the **Type** drop-down menu.
    - **Simple**: For this type, you must enter a simple value for the property value.
    - **Anisotropic**: For this type, you must specify material properties for three principal directions:
        a. **T(1,1)**
        b. **T(2,2)**
        c. **T(3,3)**

    > **Note:** The Anisotropic type is not used in RMxprt design analysis, but it will be transferred to Maxwell 3D Design automatically when the design is created by RMxprt.

4. **Mass Density** is a Simple parameter.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specifying RMxprt Solution Settings

Specify how RMxprt computes a solution by adding a *solution setup* to the design. You can define more than one solution setup per design.

To add a solution setup to a design:

1. Select a design in the Project Manager tree.
2. Click **RMxprt** > **Analysis Setup** > **Add Solution Setup**.

   Alternatively, right-click **Analysis** in the Project Manager tree, and then click **Add Solution Setup** on the shortcut menu.

   The **Solution Setup** window appears. It is divided among the following tabs:

General	Includes general solution settings, including rated output power, speed, operating temperature, etc.
<machine type>	Includes settings specific to the selected machine type.
Defaults	Includes settings to save and clear user-defined default values.

3. Click the **General** tab.
4. If available for the machine you are using, select an **Operation Type** from the drop-down menu. The options are **Motor** or **Generator**.

   > **Note:** To enable selection of **Generator** for Adjust-Speed Synchronous Machines, the machine **Control Type** must be set to **AC** in its Properties window.

   When you make the selection, this makes a difference in the **Load Type** available.

5. Select a **Load Type** from the drop-down menu. For **Motor** operation, the options are
   - **Const Speed**
   - **Const Power**
   - **Const Torque**
   - **Linear Torque**
   - **Fan Load**

   If the model has an **Operation Type**, and you select **Generator**, the **Load Type** options are

   - **Infinite Bus**
   - **Independent Generator**
6. Type the **Rated Output Power**, and select the units.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Type the **Rated Voltage**, and select the units.
8. Type the **Rated Speed**, and select the units.
9. Type the **Operating Temperature**, and select the units.
10. Click the **<machine type>** tab (if any for this machine).

    The options vary depending on the machine. For example, the 3-Phase Induction Motor includes options for

    - Frequency and Units
    - Winding Connection (Wye or Delta).

    The 3-Phase Synchronous Machine includes options for

    - Rated Power Factor
    - Winding Connection (Wye or Delta)
    - Exciter Efficiency
    - Input Exciting current and units

    The Brushless DC Motor does not use the <**machine type**> tab.
11. Specify the desired settings, based on the machine type you have selected.
12. Click **OK**.

> **Note:** To edit a setup after it has been created, right-click the specific setup (for example, **Setup1**), under **Analysis** in the Project Manager tree, and then click **Properties** on the shortcut menu.

Generating a Custom Design Sheet for RMxprt

RMxprt allows users to import all the data items of **Design Output** into a **Microsoft Excel** worksheet in order for users to design **Design Sheet** of their own styles according to their own requirements using **Microsoft Excel**.

Before you can specify a design sheet, you first need to customize a template in Microsoft Excel and to set the export options.

To set the export options:

1. Click **RMxprt** > **Design Settings**.

   The **Design Settings** dialog box appears. Select the **Export Options** tab.
2. In the **Design Sheet** section, specify an **Excel Template** by clicking the **...** button, selecting the template you want to use, and clicking **Save**.
3. Click **OK**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To create a design sheet based on a previously-customized template after the design has been analyzed:

1. Click **RMxprt** > **Analysis Setup** > **Export**.
   The **Export** dialog box appears.
2. Select **Customized Design Sheet** from the **Type** drop-down menu.
3. Choose the **Solution Setup** from the drop-down menu.
4. In the **Path** field, enter the location to export the files to, or use the ellipsis (...) button to browse to the desired location.
5. Click **OK**.

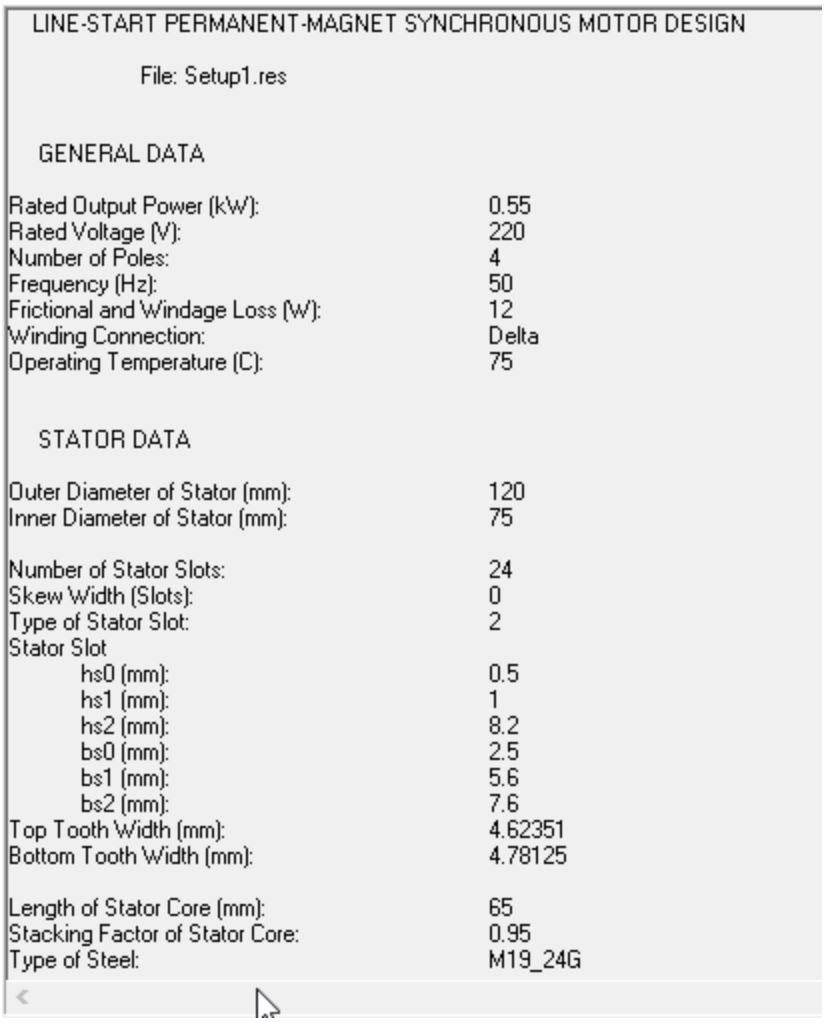
RMxprt will connect to **Microsoft Excel** according to the set path and automatically import the relevant data from **Design Output** into a copy of the customized **Design Sheet Template**. Specific data not available in **Design Output** is shown as **N/A**, for instance, **RS** in **Slot Sizes** is *N/A* **(mm)** when stop type is 1. The postprocessing of data is automatically performed, for instance, **Winding Weight** is *0.947* **(kg)**.

> **Note:** Several **Design Sheet** examples are shipped with RMxprt in the examples subdirectory of your installation.

Key Words in Output Data for RMxprt

In **Design Output** of RMxprt, the literal expressions for various parameters are termed key words of output data. For example, the **Design Output** for Line-start Permanent-magnet Synchronous Motor (*lssm*) example project sm-1.aedt is shown below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



LINE-START PERMANENT-MAGNET SYNCHRONOUS MOTOR DESIGN

File: Setup1.res

GENERAL DATA

Rated Output Power (kW):	0.55
Rated Voltage (V):	220
Number of Poles:	4
Frequency (Hz):	50
Frictional and Windage Loss (W):	12
Winding Connection:	Delta
Operating Temperature (C):	75

STATOR DATA

Outer Diameter of Stator (mm):	120
Inner Diameter of Stator (mm):	75
Number of Stator Slots:	24
Skew Width (Slots):	0
Type of Stator Slot:	2
Stator Slot	
hs0 (mm):	0.5
hs1 (mm):	1
hs2 (mm):	8.2
bs0 (mm):	2.5
bs1 (mm):	5.6
bs2 (mm):	7.6
Top Tooth Width (mm):	4.62351
Bottom Tooth Width (mm):	4.78125
Length of Stator Core (mm):	65
Stacking Factor of Stator Core:	0.95
Type of Steel:	M19_24G

In the Design Sheet, "Rated Output Power", "Rated Voltage", "Number of Poles", "Frequency", "Frictional and Windage Loss", *etc.* are all key words of output data.

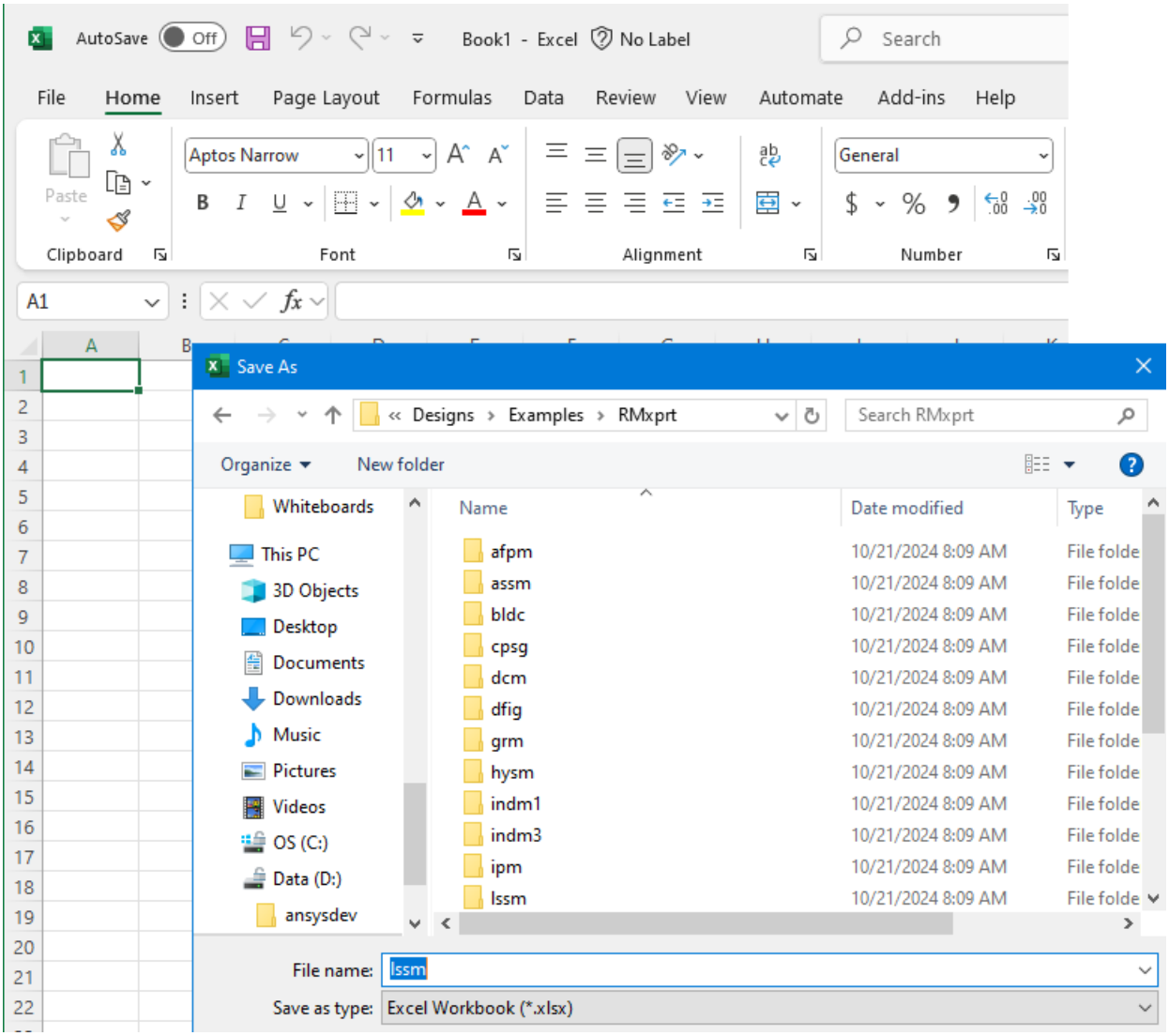
Creating RMxprt Customized Design Sheet Template

As a sample example, the Line-start Permanent-magnet Synchronous Motor (*lssm*) is used to demonstrate the process for creating a **CustomizedDesign Sheet template**.

Start **Microsoft Excel**, rename the blank worksheet file as "*lssm*" and save it under the path *\\Examples\RMxprt\lssm* as shown below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: In the example below, the *\\Examples\RMxprt* directory has been copied from *"C:\Program Files\ANSYS Inc\v252\AnsysEM\Examples\RMxprt"* directory to the user's personal Design directory. By copying the original files to another location, you can save the original files included with the software in case you need access to them at a later date.

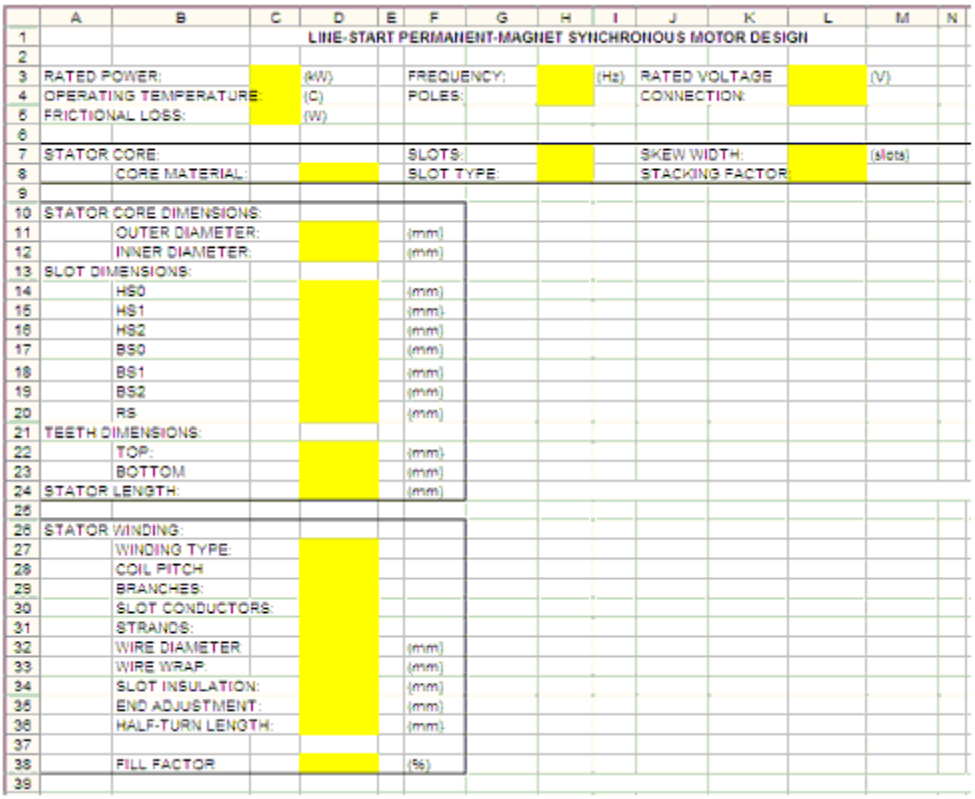


Design Template of Microsoft Excel Worksheet in Preferred Styles

According to special requirements and preferences, select relevant parameters, arrange the page formats of worksheet. Taking as example a part of the parameters of the Design Sheet of

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the Line-start Permanent-magnet Synchronous Motor (*lssm*), the designed worksheet template is shown below.



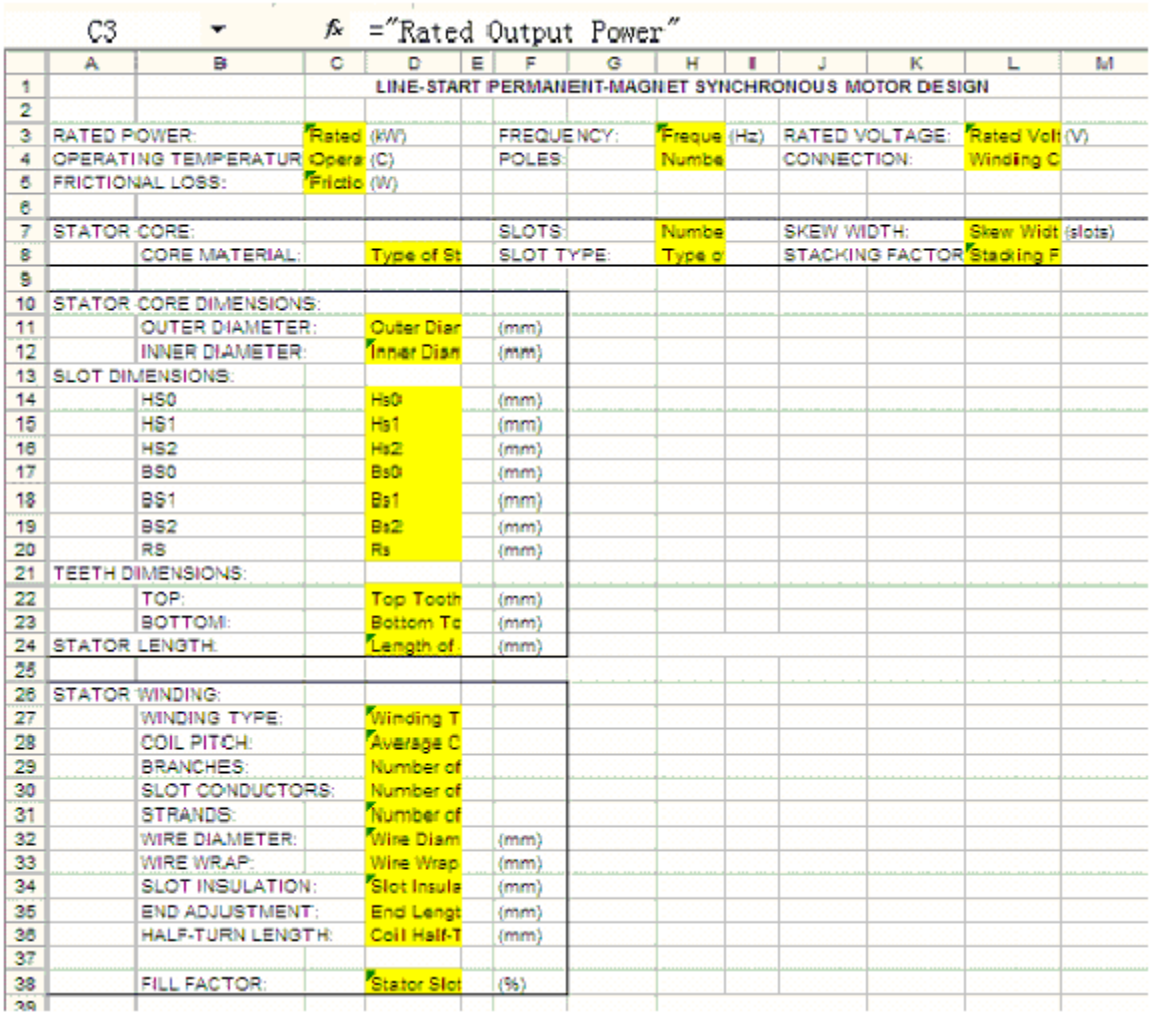
	A	B	C	D	E	F	G	H	I	J	K	L	M	N
1	LINE-START PERMANENT-MAGNET SYNCHRONOUS MOTOR DESIGN													
2														
3	RATED POWER:			(kW)		FREQUENCY:			(Hz)	RATED VOLTAGE			(V)	
4	OPERATING TEMPERATURE:			(C)		POLES:				CONNECTION:				
5	FRICTIONAL LOSS:			(W)										
6														
7	STATOR CORE:					SLOTS:				SKEW WIDTH:			(slots)	
8		CORE MATERIAL:				SLOT TYPE:				STACKING FACTOR:				
9														
10	STATOR CORE DIMENSIONS:													
11		OUTER DIAMETER:				(mm)								
12		INNER DIAMETER:				(mm)								
13	SLOT DIMENSIONS:													
14		HS0				(mm)								
15		HS1				(mm)								
16		HS2				(mm)								
17		BS0				(mm)								
18		BS1				(mm)								
19		BS2				(mm)								
20		RS				(mm)								
21	TEETH DIMENSIONS:													
22		TOP:				(mm)								
23		BOTTOM				(mm)								
24	STATOR LENGTH:					(mm)								
25														
26	STATOR WINDING:													
27		WINDING TYPE:												
28		COIL PITCH												
29		BRANCHES:												
30		SLOT CONDUCTORS:												
31		STRANDS:												
32		WIRE DIAMETER				(mm)								
33		WIRE WRAP:				(mm)								
34		SLOT INSULATION:				(mm)								
35		END ADJUSTMENT:				(mm)								
36		HALF-TURN LENGTH:				(mm)								
37														
38		FILL FACTOR				(%)								
39														

In the template, the yellow-colored areas are reserved for importing data.

Resort to Keywords in Design Output

In the spaces for importing data in the template (shown in yellow), key in = *"xxxxx"*. Within the double quotation marks, *xxxxx* stands for the relevant keywords as shown below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



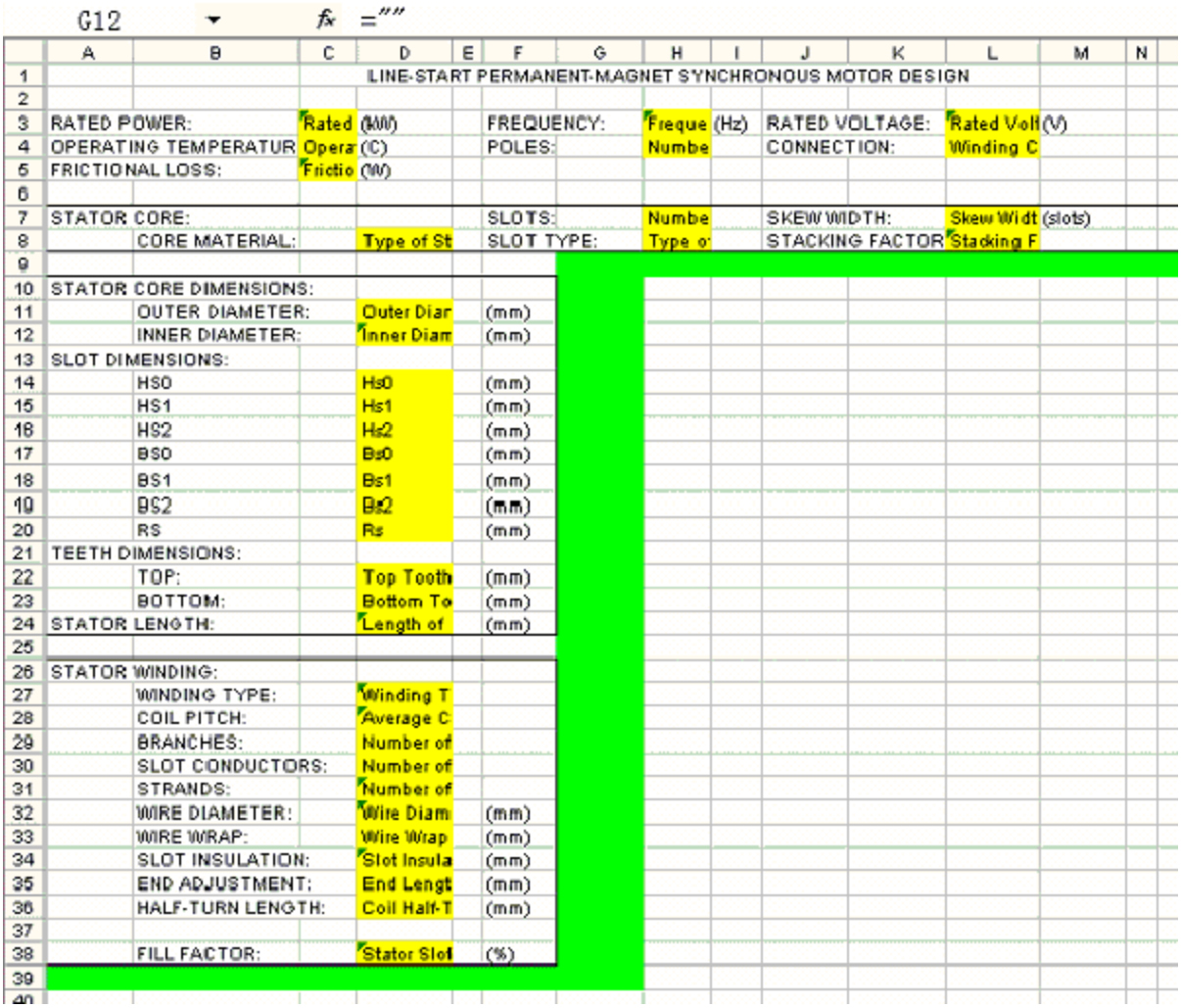
C3 fx ="Rated Output Power"

	A	B	C	D	E	F	G	H	I	J	K	L	M
1	LINE-START PERMANENT-MAGNET SYNCHRONOUS MOTOR DESIGN												
2													
3	RATED POWER:		Rated	(kW)		FREQUENCY:		Freque	(Hz)	RATED VOLTAGE:		Rated Volt	(V)
4	OPERATING TEMPERATUR		Opera	(C)		POLES:		Numbe		CONNECTION:		Winding C	
5	FRICTIONAL LOSS:		Frictio	(W)									
6													
7	STATOR CORE:					SLOTS:		Numbe		SKEW WIDTH:		Skew Widt	(slots)
8		CORE MATERIAL:		Type of St		SLOT TYPE:		Type o		STACKING FACTOR:		Stacking F	
9													
10	STATOR CORE DIMENSIONS:												
11		OUTER DIAMETER:		Outer Diar		(mm)							
12		INNER DIAMETER:		Inner Diar		(mm)							
13	SLOT DIMENSIONS:												
14		HS0		Hs0		(mm)							
15		HS1		Hs1		(mm)							
16		HS2		Hs2		(mm)							
17		BS0		Bs0		(mm)							
18		BS1		Bs1		(mm)							
19		BS2		Bs2		(mm)							
20		RS		Rs		(mm)							
21	TEETH DIMENSIONS:												
22		TOP:		Top Tooth		(mm)							
23		BOTTOM:		Bottom To		(mm)							
24	STATOR LENGTH:			Length of		(mm)							
25													
26	STATOR WINDING:												
27		WINDING TYPE:		Winding T									
28		COIL PITCH:		Average C									
29		BRANCHES:		Number of									
30		SLOT CONDUCTORS:		Number of									
31		STRANDS:		Number of									
32		WIRE DIAMETER:		Wire Diam		(mm)							
33		WIRE WRAP:		Wire Wrap		(mm)							
34		SLOT INSULATION:		Slot Insula		(mm)							
35		END ADJUSTMENT:		End Lengt		(mm)							
36		HALF-TURN LENGTH:		Coil Half-T		(mm)							
37													
38		FILL FACTOR:		Stator Slot		(%)							
39													

Set Boundary for Data Imported into Worksheet for the RMxprt App

The RMxprt application automatically searches for matching key words while importing data into the Microsoft Excel® worksheet. To reduce the searching space, and therefore the searching time, it is possible to set the searching boundary. RMxprt defines = "" as identification of boundary. In the following figure, for instance, the cells in the green-colored area are all keyed in with = "" to form a boundary. While importing data, the RMxprt application will not search below or to the right of the boundary for matching key words.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

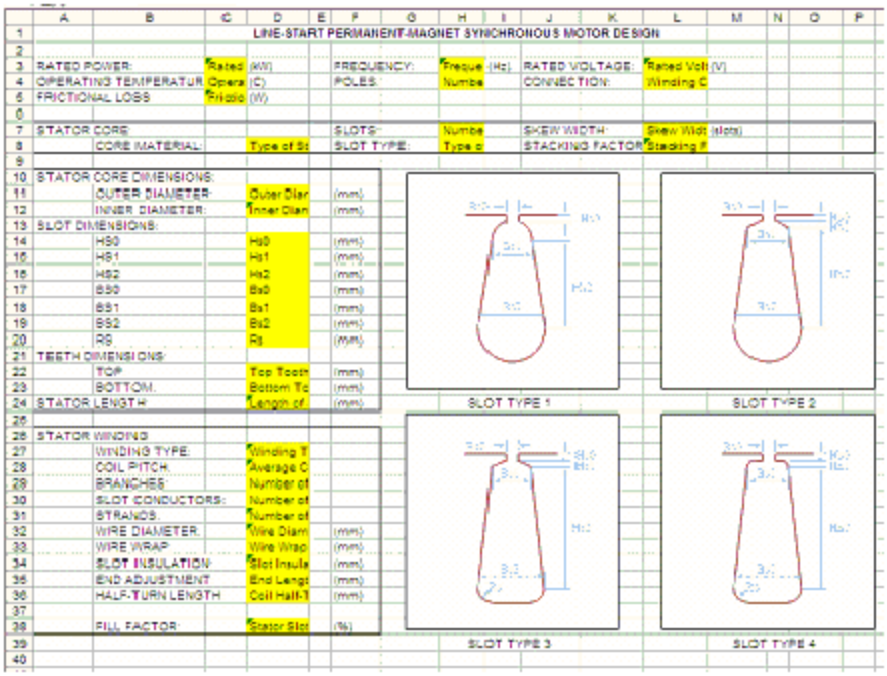


	A	B	C	D	E	F	G	H	I	J	K	L	M	N
1	LINE-START PERMANENT-MAGNET SYNCHRONOUS MOTOR DESIGN													
2														
3	RATED POWER:		Rated	(W)		FREQUENCY:		Freque	(Hz)	RATED VOLTAGE:		Rated Volt	(V)	
4	OPERATING TEMPERATUR		Opera	(C)		POLES:		Numbe		CONNECTION:		Winding C		
5	FRICTIONAL LOSS:		Frictio	(W)										
6														
7	STATOR CORE:					SLOTS:		Numbe		SKEW WIDTH:		Skew Widt	(slots)	
8		CORE MATERIAL:		Type of St		SLOT TYPE:		Type o		STACKING FACTOR		Stacking F		
9														
10	STATOR CORE DIMENSIONS:													
11		OUTER DIAMETER:		Outer Diar		(mm)								
12		INNER DIAMETER:		Inner Diam		(mm)								
13	SLOT DIMENSIONS:													
14		HS0		Hs0		(mm)								
15		HS1		Hs1		(mm)								
16		HS2		Hs2		(mm)								
17		BS0		Bs0		(mm)								
18		BS1		Bs1		(mm)								
19		BS2		Bs2		(mm)								
20		RS		Rs		(mm)								
21	TEETH DIMENSIONS:													
22		TOP:		Top Tooth		(mm)								
23		BOTTOM:		Bottom To		(mm)								
24	STATOR LENGTH:			Length of		(mm)								
25														
26	STATOR WINDING:													
27		WINDING TYPE:		Winding T										
28		COIL PITCH:		Average C										
29		BRANCHES:		Number of										
30		SLOT CONDUCTORS:		Number of										
31		STRANDS:		Number of										
32		WIRE DIAMETER:		Wire Diam		(mm)								
33		WIRE WRAP:		Wire Wrap		(mm)								
34		SLOT INSULATION:		Slot Insula		(mm)								
35		END ADJUSTMENT:		End Lengt		(mm)								
36		HALF-TURN LENGTH:		Coil Half-T		(mm)								
37														
38		FILL FACTOR:		Stator Slot		(%)								
39														

Insert Figures into Template for RMxprt

With the **Paste** function, it is possible to insert desired figures into the worksheet template. In the following example, for instance, four figures of slot types are inserted.

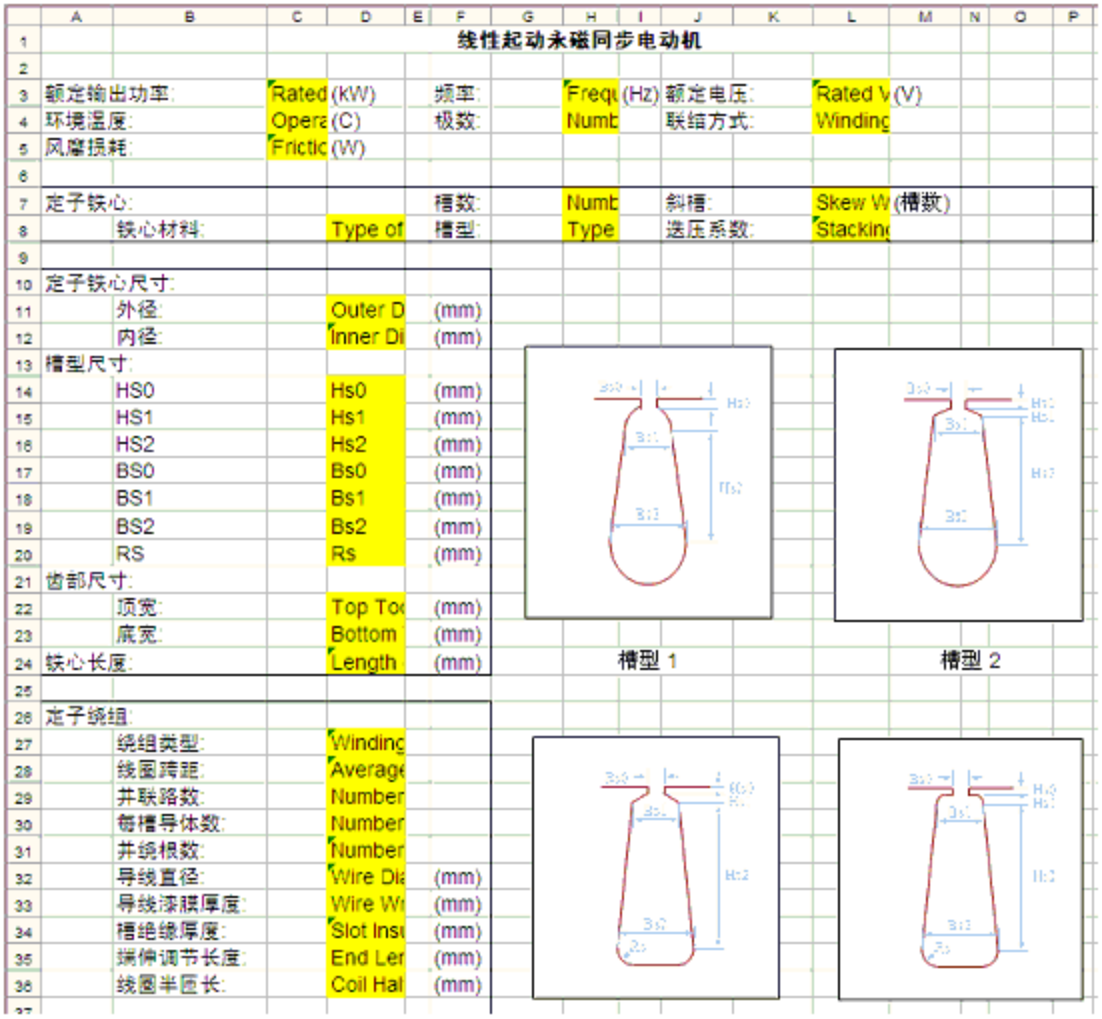
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Use Different Languages for RMxprt Design Sheets

It is possible to use a preferred language other than English in the Microsoft Excel worksheet template. In the following example, for instance, Simplified Chinese is used.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Warning: Keywords must be in English.

Postprocess Data for RMxprt

Using the relevant functions of **Microsoft Excel**, it is possible to postprocess data in the worksheet template. For example, calculate the weight of a winding using the following formula:

windingweight = *number of slots* x *number of conductors per slot* x *number of parallel wires* x *length of half turns of coil* x *sectional area of wire* x *specific weight of wire*

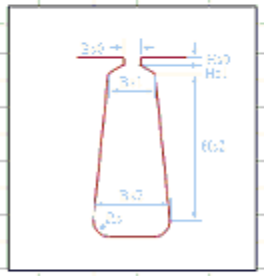
In the following figure, the formula is entered into the relevant cell as:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

=H7*D30*D31*D36*3.14*D32*D32*0.0078/4

SUM =H7*D30*D31*D36*3.14*D32*D32*0.0078/4

	A	B	C	D	E	F	G	H	I	J	K
26	定子绕组:										
27		绕组类型:		Winding							
28		线圈跨距:		Average							
29		并联路数:		Number							
30		每槽导体数:		Number							
31		并绕根数:		Number							
32		导线直径:		Wire Di		(mm)					
33		导线漆膜厚度:		Wire Wi		(mm)					
34		槽绝缘厚度:		Slot Insu		(mm)					
35		端伸调节长度:		End Ler		(mm)					
36		线圈半匝长:		Coil Hal		(mm)					
37											
38		槽满率:		Stator S		(%)					
39											
40		绕组重量	=H7*D30*D31*D36*3.14*D32*D32*0.0078/4								
41											



槽型 3

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Running an RMxprt Simulation

After you specify how the RMxprt application will compute the solution, begin the solution process.

1. Select a solution setup in the Project Manager tree.
2. Click **RMxprt** > **Analyze**.

To run more than one analysis at a time, follow the same procedure while a simulation is running. The next solution setup will be solved when the previous solution is complete.

To solve *every solution setup in a design*:

1. In the Project Manager tree, under the design you want to solve, select **Analysis**.
2. Click **RMxprt** > **Analyze All.**

Each solution setup is solved in the order it appears in the Project Manager tree.

Aborting RMxprt Analyses

To end the solution process before it is complete, right-click in the **Progress** window, and select **Abort** from the shortcut menu.

The analysis stops immediately.

If you aborted the solution in the middle of an adaptive pass, the data for that pass or current frequency point is deleted. Any solutions that were completed prior to the one that was aborted are still available.

Re-solving an RMxprt Problem

If you modify a design after generating a solution, the solution in memory will no longer match the design. The solution setup with the invalid solution is marked with an *X* in the Project Manager tree and in the **Results** window.

To generate a new solution after modifying a design, follow the procedure for running a simulation:

1. Select a solution setup in the Project Manager tree.
2. Click **RMxprt** > **Analyze**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Postprocessing and Generating Reports in RMxprt

When RMxprt has completed a solution, you can display and analyze the results in the following ways:

- View solution data
- Specify output variables
- Export a model to be used in Maxwell 2D, Maxwell 3D, or Twin Builder
- Create a Customized Design Sheet
- Create Reports

Viewing RMxprt Solution Data

To access the **Solutions** dialog box, do one of the following:

- Click **RMxprt** > **Results** > **Solution Data**.
- Right-click **Results** in the project tree, and then click **Solution Data** on the shortcut menu.
- Right-click **Setup1** in the Project tree, and then click **Performance** on the shortcut menu.
- Click the **Solution Data** icon on the **Results** ribbon.

The **Solutions** dialog contains three tabs:

- **Performance** – this contains a Data field with a drop-down menu that allows you to view many different data tables, which vary with the machine type. Some examples are:
  - Aux Winding
  - Full Load Operation
  - Material Consumption
  - No Load Operation
  - Permanent Magnet
  - Rotor Data
  - Rated Parameters
  - Stator Slot
  - Stator Winding
  - Steady State Parameters
- **Design Sheet** – this displays the contexts of the **.res** file for the current setup. The file contains tables with information for such things as (depending on the machine type):
  - General Data
  - Stator Data
  - Rotor Data

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

    - Permanent Magnet Data
    - Material Consumption
    - Rated Operation
    - No-Load Operation
    - Steady State Parameters
    - No Load Magnetic Data
    - Full Load Data
    - Winding Arrangement
    - Transient FEA Input Data
- **Curves** – this displays the plots that were automatically generated by the solver. You can select these from a drop-down menu (menu contents vary with the machine type):
    - Input DC Current vs Speed
    - Efficiency vs Speed
    - Output Power vs Speed
    - Output Torque vs Speed
    - Cogging Torque in Two Teeth
    - Induced Coil Voltage at Rated Speed
    - Air Gap Flux Density
    - Induced Winding Phase Voltage at Rated Speed
    - Winding Currents Under Load
    - Phase Voltage Under Load

> **Note:** You can also open the **Solution Data** dialog box directly by clicking the **Solution Data** icon on the Results ribbon.

To print the design sheet:

1. From the **Design Sheet** tab, Right-click the design sheet to display the context menu.
2. Select **Print** from the shortcut menu.

   The **Print** dialog box appears.
3. Select the printer, and click **OK** to print.

To print plots from the **Curve** tab:

1. Right-click on the desired plot to display the context menu.
2. Click **Print** on the menu to display the **Print** dialog box.
3. Select the printer, and click **OK** to print.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** The context menu also provides commands that allow the user to change various plot characteristics such as: trace type and properties such as color; add/edit data markers and labels; edit axis, legend, and title properties. The plot image can also be copied to the clipboard for pasting in another application. Plot data can also be exported to various formatted text files that can then be imported into spreadsheets, *etc*. Refer to *Modifying the Background Properties of a Report* for additional information.

Browse Solutions in RMxprt

If you have run different solutions on a design, you can use the **Browse Solutions** option to look through and manage them.

1. Click **RMxprt** > **Results** > **Browse Solutions**

   This displays the **Solutions** dialog with the **Browse** tab selected. From here you can select a design, and view the Setup, Solution and State tables.

2. Click the **Properties** button to view the **Solution Browser** properties dialog box.

   This contains radio buttons for you to select the tree view. It can be organized as:

   - **Variation / Setup / Solution**
   - **Setup / Solution / Variation**
   - **Setup /Variation / Solution**

   Click **OK** to accept your selection and close the dialog box.

3. From the **Solutions** dialog box, you can also select and delete solutions.

Exporting a Simplorer Model or Customized Design Sheet

To export the model to a Simplorer model or Customized Design Sheet:

1. First, solve the design for the specific solution setup from which the export is desired.

   > **Note:** Attempting to export without the requisite files present automatically launches simulation to generate them.

2. Click **RMxprt** > **Analysis Setup** > **Export** to open the **Export** dialog box.

   You can also right-click on **Analysis** or **Analysis** > **Setup** in the Project Manager and select **Export**.

3. Select one of the following from the Type drop-down menu:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Simplorer Model**
- **Customized Design Sheet**

4. Select the setup you want to export from the **Solution Setup** drop-down menu.
5. Optionally, click the Variation ellipsis [**...**] button to specify a design **Variation to use during export**.
6. Specify the **Path** to store the exported files in by clicking the Path ellipsis [...] button and browsing to the desired directory. You can also create a new directory, if desired.
7. Click **OK**.

For **Customized Design Sheets**, a <*design_sheet_name*>**.xls** file will be generated.

For **Simplorer** models, a <*project name*>_<*design name*>_<*setup name*>.sml file will be exported.

You can now work with the model in the product to which you exported it, and perform other kinds of analyses.

Creating a Maxwell Design from RMxprt

RMxprt provides a way to export solved models as either a Maxwell 2D or Maxwell 3D design.

- All the RMxprt machine types are supported.
- Setups for boundaries, excitations, parameters, etc. are automatically created.
- Variable definitions and assignments are automatically mapped from the solved RMxprt design to the Maxwell design.

> **Note:** Creating a Maxwell 2D/3D design from an old version (version 15 or older) of a solved RMxprt design results in the Maxwell 2D/3D design being created *without* any variables even if the original RMxprt design uses variables. In such cases, the **Message Manager** displays a warning message informing the user that the Maxwell design being created does not contain any variable assignments from the RMxprt design; and recommends re-solving the setup to get the variable assignments.

To export the model to a Maxwell 2D Design or Maxwell 3D Design:

1. Click **RMxprt** > **Analysis Setup** > **Create Maxwell Design**

   The **Create Maxwell Design** dialog box appears.

2. Select one of the following from the **Type** drop-down menu:
   - **Maxwell 2D Design**
   - **Maxwell 3D Design**
3. Select the setup you want to export from the **Solution Setup** drop-down menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Optionally, click the **[...]** button to specify a **Variation** in the **Set Design Variation** dialog box. This allows the user to choose the default variable values to be used in the Maxwell design being created with variables.
5. Click **OK** to create the Maxwell design.

You can now work with the model in the Maxwell 2D or Maxwell 3D design, add boundaries and excitations, perform analyses, and analyze results.

Note:

- For Maxwell 2D/3D designs for motor applications, the computed output mechanical power at a given constant speed may not reach the desired rated power. Therefore, it is useful to apply constant power as the mechanical load. In Maxwell 2D/3D designs created by RMxprt, the mechanical transient is setup according to the rated mechanical output power, and equivalent damping is added to accelerate the process approaching to the rated output power. This setup is visible when **Consider Mechanical Transient** on the Mechanical tab of the Motion Setup dialog box is checked.
- For Maxwell 2D/3D designs of AC machines, RMxprt automatically enables an alternating flux (AF) model in the voltage definition for each winding to reduce transient solution time to reach steady-state when the Maxwell 2D or 3D model is created.
- For Maxwell 2D designs of double-cage induction machines with separate end rings, and of synchronous machines with separate dampers for each pole, RMxprt automatically creates multiple end connection excitations in the Maxwell 2D design. For synchronous machines, if the created geometry model has only one pole, it is changed to two poles, with two separate end connection excitations. This enables simulation of double-cage induction machines with separate rings, as well as synchronous machines with separate dampers for each pole.

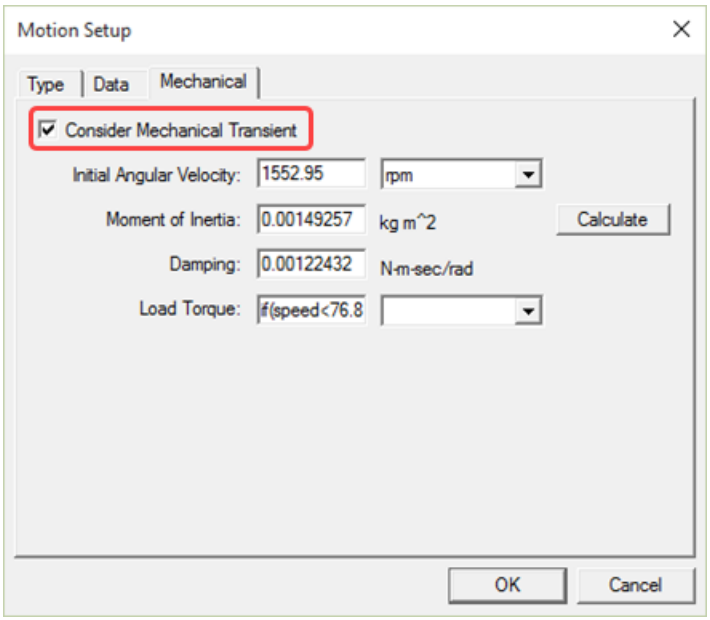
Speed Adjustment to Achieve to Rated Output Power

The speed setup in all RMxprt-created Maxwell designs is the estimated rated speed, which is computed based on the rated output power using the Equivalent Magnetic Circuit (EMC) method in RMxprt. The EMC method may not be very accurate in some cases, especially when some part of the magnetic circuit is highly saturated. In order to obtain the correct rated performance in Maxwell designs, the RMxprt-based speed setup should be adjusted so that the simulated mechanical output power matches the specified rated output power.

Warning: All rated performance data, such as torque, speed and current, must be obtained at the rated output power!

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For this purpose, all RMxprt-created Maxwell designs have an initial mechanical transient setup to automatically adjust the speed to approach the rated mechanical output power. This initial transient mechanical setup will be lost if you close the project without checking the **Consider Mechanical Transient** check box as shown in the following figure:



However, using the mechanical transient to reach to the rated output power may need much longer simulation time especially when the rotor moment of inertia is large. But reducing the value of the moment of inertia may cause speed oscillation.

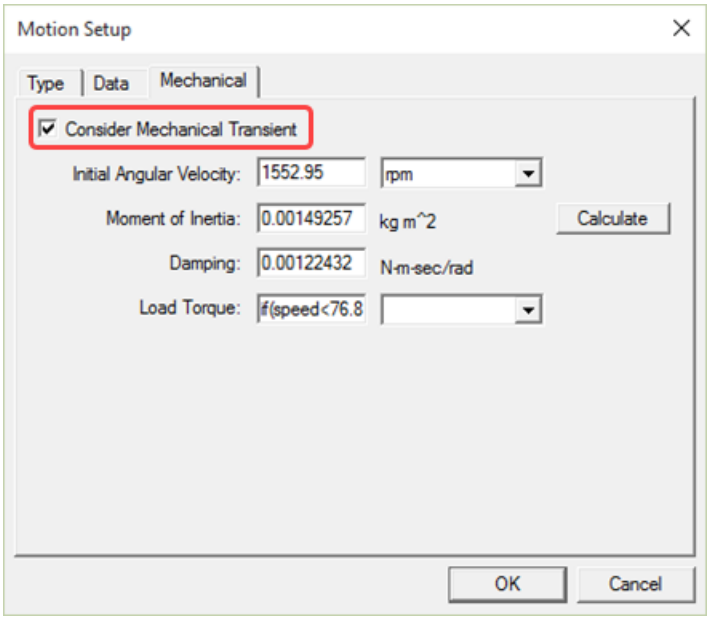
The following two methods may be used to adjust the speed to achieve to the rated output power:

- Using the initial mechanical transient setup by RMxprt
- Adjusting speed manually for a constant speed setup

Using the Initial Mechanical Transient Setup by RMxprt

To use this method, you need to check **Consider Mechanical Transient** in the **Mechanical** tab of the **Motion Setup** dialog box immediately after the design is created from RMxprt. An example showing the initial values for mechanical transient setup is shown below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note that if you close the design without checking it, and open the design later, this initial mechanical transient setup will be lost.

The method for using the mechanical transient setup is as follows:

In the Mechanical tab of the Motion Setup panel, when **Consider Mechanical Transient** is checked, the displayed **Load Torque** is actually the "Driving Torque" because the positive torque is defined in the positive speed direction. Therefore, the real positive Load Torque must be defined with a negative value. In order to automatically adjust the speed for the motor to generate the specified output power, you need to set the load torque curve with Constant Power, that is:

Tload = – Pspec / speed

where **speed** is the real rotor speed, and **Pspec** is a constant value.

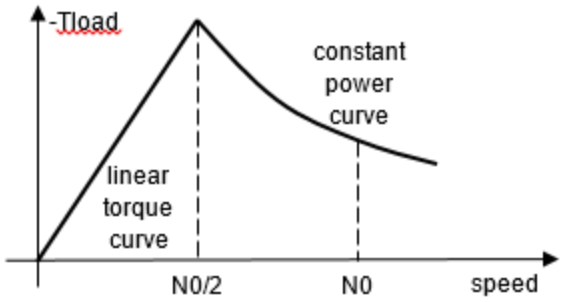
In order to apply this load torque curve for any initial speed, the load torque curve is defined as:

Tload = if(speed<N0/2, -4*Pspec/N0/N0*speed, -Pspec/speed)

where **N0** is the *synchronous speed* for synchronous motors, or the *rated speed* for other motor types in (rad/s).

The load torque is shown below, which is applicable for all motor types excluding synchronous motors.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

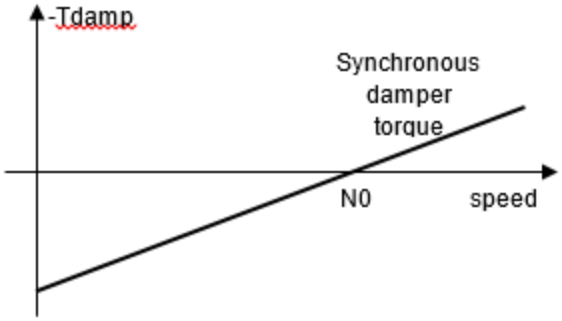


For synchronous motors, the motor driving torque varies with torque angle which is the difference between the phase angles of the applied and the induced voltages. If the initial speed is set as the synchronous speed, and the initial torque angle is set smaller than the real value, the motor driving torque will be smaller than the specified load torque. With mechanical transient, the speed will decrease, which makes the torque angle and driving torque increase. When the driving torque equals the load torque, the speed, which is lower than the synchronous speed, will stay constant temporarily. Since the speed is lower than the synchronous speed, the torque angle continues to increase, which makes the driving torque greater than the load torque, and therefore the speed increases. When the speed reaches the synchronous speed, the torque angle will be too large, which will increase the speed to exceed the synchronous speed. Therefore, the speed will oscillate about the synchronous speed. This oscillation will decay to zero if the synchronous motor has a damper cage winding. The decaying time depends on the strength of the damping effects.

In order to speed up the decaying process, an additional load torque component to model the damping effects around the synchronous speed is set for all types of synchronous motors, as expressed below:

Tdamp = - Tst * (speed/N0 - 1)

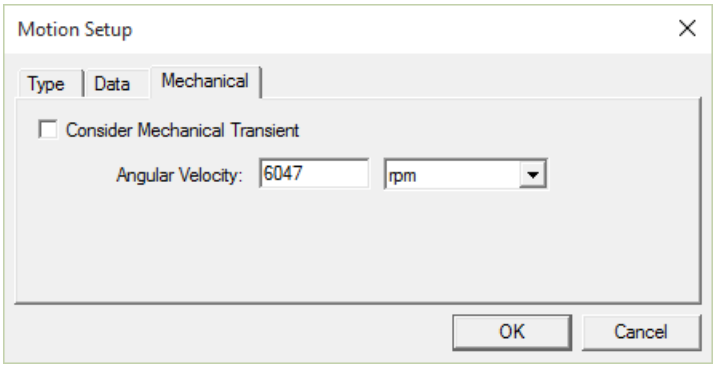
where **Tst** is set as the rated torque. You may increase **Tst** to increase the damping effects. The damping torque curve is shown below.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adjusting Speed Manually for a Constant Speed Setup

The specified rated output power can be reached by manually adjusting the input speed without checking **Consider Mechanical Transient**.



This adjustment method may require iteration, as described below.

Determine Solution at RMxprt Setting Speed

Using a Permanent Magnet DC motor as an example, assume that the solution data for the Maxwell 2D design originally created by RMxprt is as follows:

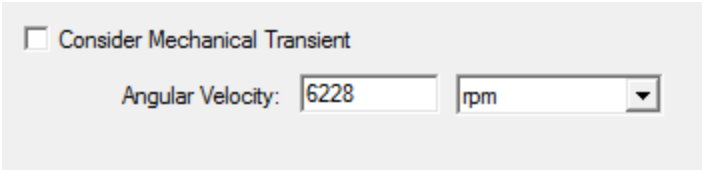
- The period is 4.96 ms based on the original setting speed *n0* = 6047 rpm for a four-pole motor.
- The average torque in the last period, from 5.04 ms to 10 ms, is *T0* = 1.4044 Nm
- The simulated output power is 889.3 W.

If the specified rated output power is 200 W, increase the motor speed in order to reduce the motor driving output power.

For the next iteration, try increasing the speed by 3%, that is: *n1 = n0 * (1+3%)* = 6228 rpm.

First Iteration – Determine Solution at First adjusted Speed

When the speed is set to be n1 = 6228 rpm:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The period for this speed is 4.82 ms.
- The new simulated average torque in the last period is *T1* = 0.7276 Nm
- The output power is *P1 = 0.7276 * 6228 * 2*pi/60* = 474.5 W.

If the output power of 474.5 W is acceptable, you can stop here. Otherwise, you need to continue to adjust the speed as described below.

Second Iteration – Determine Solution at Second Adjusted Speed

Now, there are two solutions of (*n0*, *T0*) = (6047 rpm, 1.4044 Nm) and (*n1*, *T1*) = (6228 rpm, 0.7276 Nm). Assuming the motor driving torque *T* at any speed *n* can be obtained from linear interpolation and extrapolation, the motor driving torque can be expressed as:

*T = T0 + (T1-T0) / (n1-n0) * (n-n0)* = 1.4044 – 0.003739 * (*n* – 6047) = 24.0155 – 0.003739 * *n*

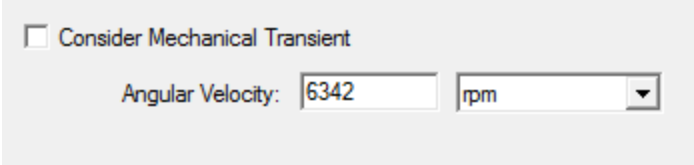
In order to obtain the specified rated output power, the load torque can expressed as:

*T = 200 / n * 60/(2 pi)* = 1910 / *n*

The new adjusted speed *n2* can be obtained by solving the above two equations, the resulting value being:

n2 = 6342 rpm

Thus, with the speed set to *n2* = 6342 rpm,



- the period for this speed is 4.73 ms,
- the new simulated average torque in the last period is *T2* = 0.3065 Nm,
- and the output power is: *P2 = 0.3065 * 6342 * 2*pi/60* = 203.6 W, which is very close to the specified rated output power of 200 W.

Comparing the discrepancy between RMxprt and Maxwell 2D results after this second iteration:

	RMxprt	**Maxwell 2D**	**Error**
Speed	6047 rpm	6342 rpm	-4.65%
Torque	0.3216 Nm	0.3065 Nm	4.93%
Output Power	200 W	203.6 W	-1.77%

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The above table shows that for a small change of speed in Maxwell setup, the torque change may be significant. In the example, the Maxwell 2D result of torque at *n0* = 6047 rpm is *T0* = 1.4044 Nm, while the torque at *n2* = 6342 rpm is *T2* = 0.3065 Nm. The speed change of 4.65% results in a torque change of 358%.

In the above method, the algorithm for the speed adjustment is suitable for most motor types including (PM)DC motors, BLDC motors, universal motors, induction motors, and switched reluctance motors. For induction motors, the first adjusted speed can be obtained by increasing or decreasing the speed slip with certain percentage.

Note:

- The speed setup by RMxprt is just an initial setup. Users must adjust the speed if the output power is far away from the specified rated output power.
- This method is suitable for all (PM)DC motors, BLDC motors, universal motors, induction motors, and switched reluctance motors. For all synchronous machines, users must adjust the power angle or the torque angle if the output power is far away from the rated output power.

Creating an Icepak Design

RMxprt provides a way to export solved models as Icepak designs.

To export the model as an Icepak Design:

1. Click **RMxprt** > **Analysis Setup** > **Create Icepak Design**; or right-click on the analysis setup in the Project tree and select **Create Icepak Design** from the context menu.

   The **Create Icepak Design** dialog box appears.

2. Select the setup you want to export from the **Solution Setup** drop-down menu.
3. Optionally, click the **[...]** button to specify a **Variation** in the **Set Design Variation** dialog box. This allows the user to choose either the nominal design or to choose the variation to be used in the Icepak design being created.
4. Enter or choose the **Path** where the Icepak project is to be output.
5. Do one of the following:
   a. Click **OK** without selecting **Launch Icepak application** to generate the Icepak project and put it in the designated folder.
   b. Select **Launch Icepak application**, and click **OK** to generate the Icepak project in the designated folder and automatically open the Icepak design in the Icepak application.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** If you want to launch the Icepak application, ensure that the Workbench path is set in the Tools/Options/General Option/Miscellaneous tab. Refer to the Icepak documentation for information on working with Icepak designs.

Creating Reports in RMxprt

After RMxprt has generated a solution, all of the results for that solution are available for analysis. One of the ways you can analyze your solution data is to create a report, or graphical representation, that displays the relationship between a design's values and the corresponding analysis results. Reports are created using the **RMxprt** > **Results** > **Create RMxprt Report** > ***<report_type>*** command. The available options in the resulting **Reports** dialog box depend on the report type you create and the available solution data.

You can also create reports using the **RMxprt** > **Results** > **Create Quick Report** command. The **Quick Report** feature lets you select from a list of predefined categories (such as coil voltage or torque) from which to create a rectangular plot.

If you have created custom report templates (for example, including your company name or other format changes), you can also create a report based on that template by selecting **RMxprt** > **Results** > **Report Templates > *<templateName>***.

You can also create a report from a Report Data (`.rdat`) file by selecting **RMxprt** > **Results** > **Create Report from File**, then browse to the desired `.rdat` file and open it. (Refer to *Performance Computation Based on FEA Created ROM for Induction Machines* for an example.)

The report appears in the view window and is listed in the Project Manager tree.

Modifying Reports in RMxprt

To modify the data that is plotted in a report:

1. In the Project Manager tree, right-click the report you want to modify.
2. Select **Modify Report** from the shortcut menu.
3. Modify the selections in the Report dialog box as needed.
4. Click **Close** when you are finished modifying the report.

   The updated report appears in the view window.

To update all modified reports, click **RMxprt** > **Results** > **Update Reports**.

Opening All Reports in RMxprt

To open all reports for a project, select **RMxprt** > **Results** > **Open All Reports**. This opens all reports.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To simplify viewing and comparisons, it may be helpful to use **Window** > **Cascade** or **Window** > **Tile Horizontally** or **Window** > **Tile Vertically** commands.

To close all open reports, select **Window** > **Close All**.

Deleting All Reports in RMxprt

To delete all reports for a project, select **RMxprt** > **Results** > **Delete All Reports**.

This deletes all reports for the project.

Selecting the Display Type in RMxprt

The information in a report can be displayed in several formats. Select from the following **Display Type** formats in the **Create Report** dialog box:

Rectangular Plot	A 2D rectangular (x-y) graph.
Rectangular Stacked Plot	A series of 2D rectangular (x-y) graphs stacked vertically, sharing a common x-axis scale, but with each trace having its own y-axis scale.
3D Rectangular Plot	A 3D rectangular (x-y-z) graph.
3D Rectangular Bar Plot	A 3D rectangular (x-y-z) bar graph.
3D Polar Plot	A 3D circular plot divided by spherical coordinates.
Data Table	A grid with rows and columns that displays, in numeric form, selected quantities against a swept variable or another quantity.

Creating 2D Rectangular Plots in RMxprt

A rectangular plot is a 2D, xy graph of results.

1. Click **RMxprt** > **Results** > **Create RMxprt Report** > **Rectangular Plot**.

   The **Report** window appears.

2. In the **Context** section make selections from the following field or fields, depending on the design and solution type.
   a. Solution field with a drop down selection list. This lists the available solutions, whether sweeps or adaptive passes.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Domain field with a drop-down selection list containing options for plotting vs ElectricDegree, Speed, or a user-selected Parameter.

3. Under the **Trace** tab, **Y** component section, specify the information to plot along the y axis:

   a. In the **Category** list, click the type of information to plot.
   b. In the **Quantity** list, click the value to plot.
   c. In the **Function** list, click the mathematical function of the quantity to plot.
   d. Value field displays the currently specified Quantity and Function. You can edit this field directly.

      > **Note:** Color indicates if an expression is valid.

   e. **Range Function** button -- opens the **Set Range Function** dialog box. This applies currently specified Quantity and Function.

4. On the **Trace** tab, **X** (Primary sweep) line, specify the quantity to plot along the x axis in one of the following ways:
   - Select the sweep variable to use from the drop down list.
   - If sweeps are available, you can select the browse [...] button to display a dialog box that lets you select particular sweep values, specify a range of sweep values (for Time sweeps), or **Use all values** (the default setting). The quantity will be plotted against the primary sweep variable listed.
5. On the **Families** tab, confirm or modify the sweep variables that will be plotted.
6. Click **New Report.**

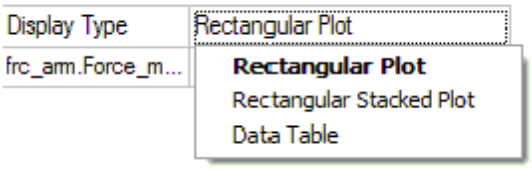
   This creates a new report in Project Manager tree, displays the report with the defined trace, and enables the **Add Trace** button on the **Report** dialog box. The default name is based on the Report Category you selected, (for example, Force Plot *n* or Output Variables Plot *n*). You can edit the plot names in the project tree and the plot header text in the report synchronizes.

   The function of the selected quantity will be plotted against the swept variable values or quantities you specified on an xy graph. The plot is listed under **Results** in the project tree and the traces are listed under the plot. When you select the traces or plots, their properties are displayed in the Properties window. These properties can be edited directly to modify the plot.

7. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

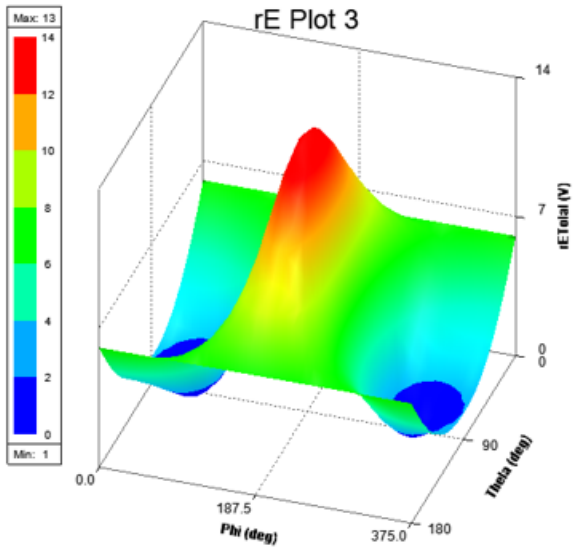
You can also modify the display type of an existing plot from the Properties window for that plot. Select the Report icon in the Project Manager tree to display the Properties window. Selecting the Display Type field displays a menu with selections available for that plot.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Creating 3D Rectangular Plots in RMxprt

Below is an example of a 3D rectangular plot.



You can rotate, zoom and pan a plot. When you rotate, the Cartesian grid responds so that the curve always remains in front and the grids behind.

Clicking on a plot entity selects it, highlighting the selected entity in bold.

Double-clicking anywhere in the plot brings up the Properties dialog box. the properties are grouped appropriately under various tabs, which correspond to plot entities:

- General: For general plot properties such as Visual Detail level and background color
- Header: Properties related to plot Header/Title.
- Axis [X|Y|Z]: Properties related to the 3 axes
- Grid [XY|YZ|ZX]: Properties related to the 3 grids
- ColorKey: Properties related to ColorKey, including borders, background, Min and Max, as well as number format and precision.
- Contour: Properties related to contouring of all curves/surfaces
- Surface: Properties related to the curve

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting a property also displays its properties in the Property window. You can edit the properties to customize the appearance of the plot. See "Controlling Visual Detail in a 3D Plot" on the facing page.

To create a 3D Rectangular Plot:

1. Click **RMxprt** > **Results** > **Create RMxprt Report** > **3D Rectangular Plot**.
2. In the **Context** section make selections from the following field or fields, depending on the design and solution type.
    a. Solution field with a drop-down selection list. This lists the available solutions, whether sweeps or adaptive passes.
    b. Domain field with a drop-down selection list containing options for plotting vs ElectricDegree, Speed, or a user-selected Parameter.
3. Under the **Z** tab, specify the information to plot along the z axis:
    a. In the **Category** list, click the type of information to plot.
    b. In the **Quantity** list, click the value to plot.
    c. In the **Function** list, click the mathematical function of the quantity to plot.
    d. The **Value** field displays the currently specified Quantity and Function. You can edit this field directly.

    > **Note:** Color indicates if an expression is valid.

    e. **Range Function** button -- opens the **Set Range Function** dialog box. This applies currently specified Quantity and Function.
4. On the **Trace** tab, **Y** (**Secondary Sweep**) lines, specify the information to plot along the y-axis in one of the following ways:
    - Select the sweep variable to use from the Secondary Sweep drop down list.
    - If sweeps are available, you can also select the browse [...] button to display a dialog box that lets you select particular sweep values, specify a range of sweep values (for Time sweeps), or **Use all values** (the default setting). The quantity will be plotted against the primary sweep variable listed.
5. On the **Trace** tab, **X** (**Primary Sweep**) lines, specify the information to plot along the x-axis in one of the following ways:
    - Select the sweep variable to use from the Primary Sweep drop-down list.
    - If sweeps are available, you can also select the browse [...] button to display a dialog box that lets you select particular sweep values, specify a range of sweep values (for Time sweeps), or **Use all values** (the default setting). The quantity will be plotted against the primary sweep variable listed.
6. On the **Families** tab, confirm or modify the sweep variables that will be plotted.
7. Click **New Report**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

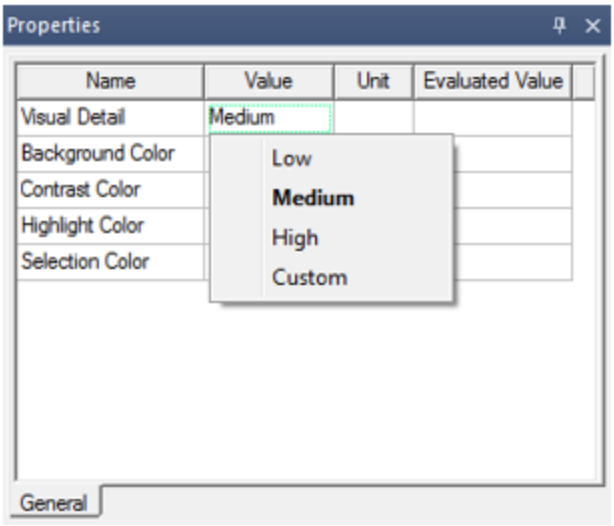
This creates a new report in Project tree, displays the report with the defined trace, and enables the **Add Trace** button on the **Report** dialog box. The default name is based on the Report Category you selected, (for example, Force Plot *n* or Output Variables Plot *n*). You can edit the plot names in the project tree and the plot header text in the report synchronizes.

The function of the selected quantity or quantities will be plotted against the values you specified on an x-y-z graph. The plot is listed under **Results** in the Project Manager tree. When you select the traces or plots, axis or grid labels, plot header, color key, or variable labels, their properties are displayed in the Properties window. The properties for each plot element can be edited directly to modify the plot content and appearance. See Modifying the Background Properties of a Report.

8. Optionally, add another trace to the plot by following the procedure above, using **Add Trace rather than New Report**.

Controlling Visual Detail in a 3D Plot

If a particular plot seems busy with information, you can edit plot properties, such as Axis and Grid Attributes for discrete levels of visual detail to improve readability. Double-click anywhere on a plot to display the Properties dialog box. The Visual Detail property on the General tab also provides control suited to different screen and plot sizes.

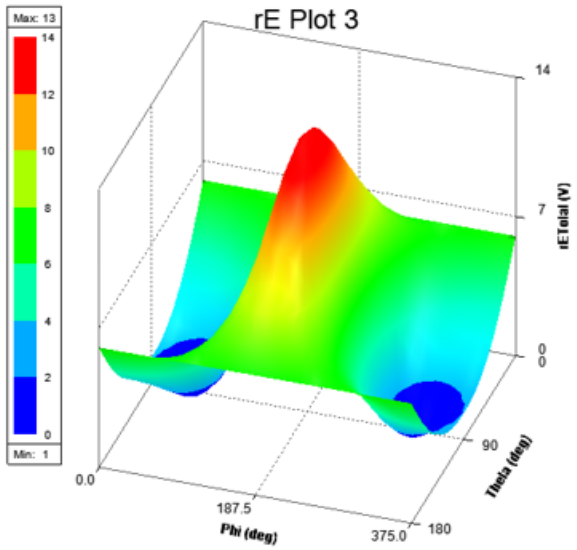


The Visual Detail menu has four options: Low, Medium (the default), High, and Custom. If you select any Visual Detail, the 3D plot is rendered according to the selected Visual Detail level and the properties reflect the values chosen for the selected visual detail level. From this predefined visual detail level, if you modify any properties, Visual Detail is automatically set to Custom (or to another predefined visual detail level if the edits happen to match the settings for that level).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also manually set Visual Detail to Custom. In such a case, Custom will inherit property values corresponding to the previous level. This ensures that you can customize settings starting from a baseline provided by the preconfigured Low, Medium or High Visual Detail levels.

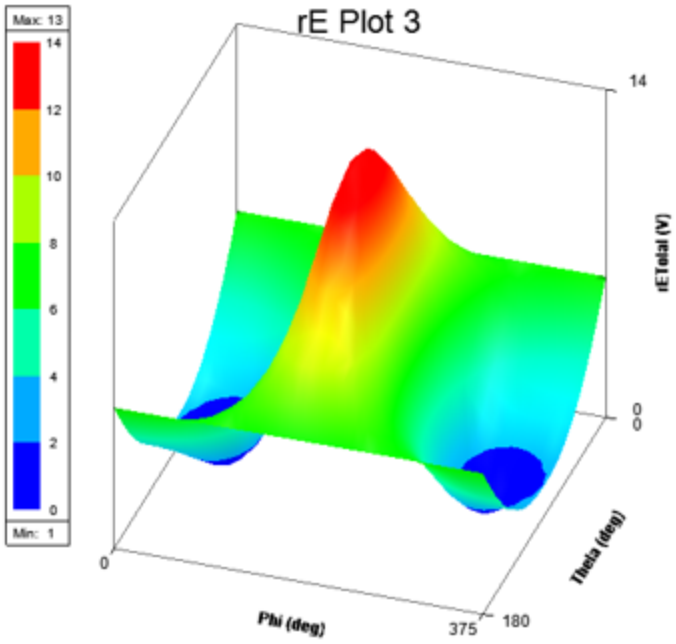
3D Rectangular Plot with Medium Visual Detail



On creation, a 3D Rectangular Plot has Visual Detail set to Medium and looks and feels as shown above. Specifically, under Medium Visual Detail level, a 3D Rectangular Plot has 3 ticks per axis (X, Y, Z axis) which will show min, max and middle value. This setting also shows axes labels.

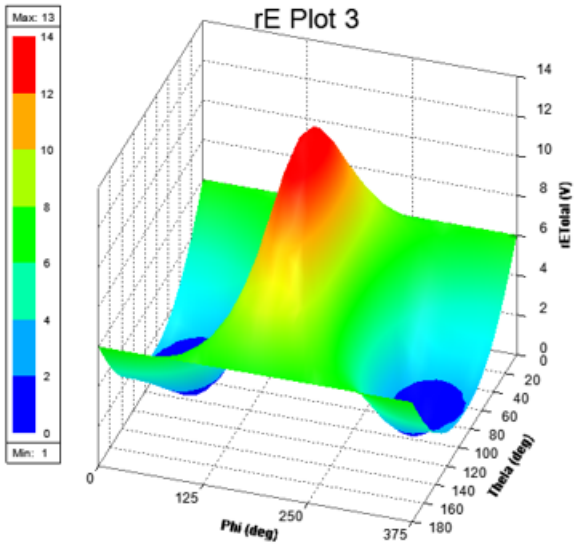
3D Rectangular Plot with Low Visual Detail

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



With Visual Detail set to Low, a 3D Rectangular Plot shows axes with 2 ticks corresponding to min and max values. It also shows axes labels and grid borders.

3D Rectangular Plot with High Visual Detail

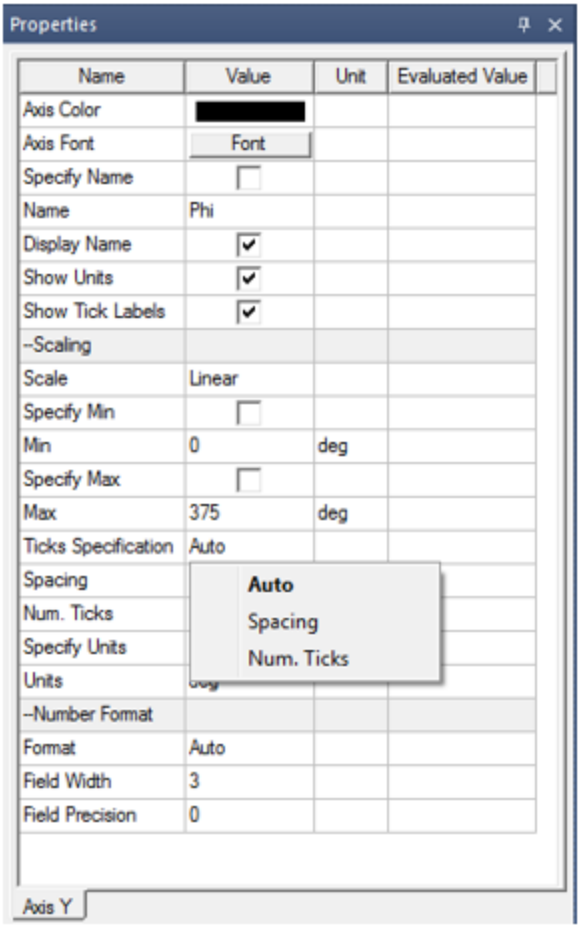


With Visual Detail set to High, a 3D Rectangular Plot shows all Cartesian axes and grids together with all ticks and axes labels.

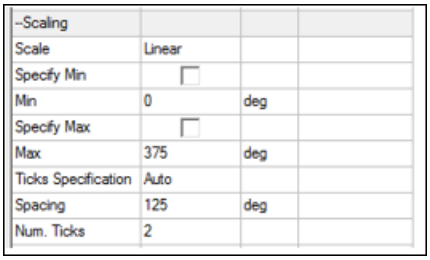
Axis Properties: Ticks Specification and Num. Ticks

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ticks Specification is available on Axis properties, as shown below:



Ticks Specification is a menu with possible values as Auto, Spacing, and Num. Ticks, with Auto being the default value. If Ticks Specification is Auto, then a spacing value is automatically calculated and used to calculate and display the tick labels. **Spacing** shows the calculated value, and Num. Ticks shows the number of ticks based on this spacing value, as shown below:



You can edit the **Spacing** field when Ticks Specification is set to Spacing; otherwise, it is read only.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

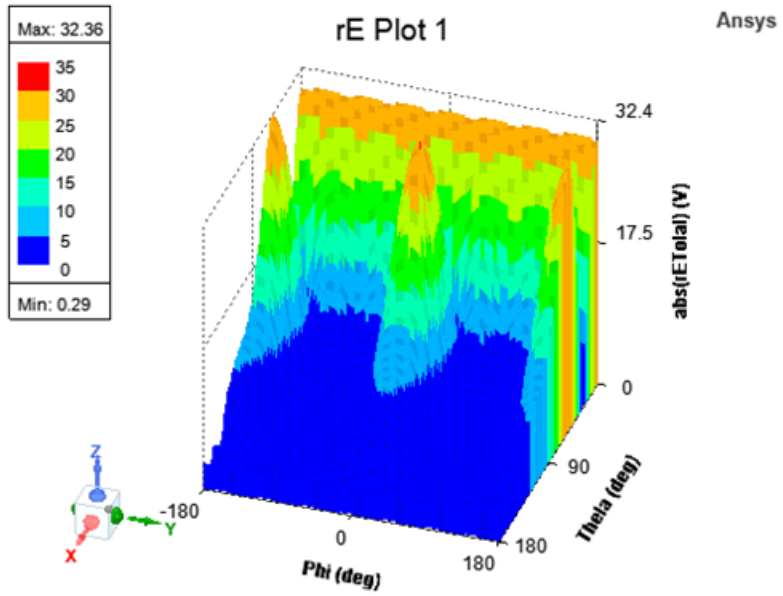
You can edit the **Num. Ticks** field when Ticks Specification is Num. Ticks; otherwise, it is read only.

Valid Num. Ticks are between 0 and 100, including 0 and 100. If you enter an invalid value, an error message is shown. If you enter a spacing value that results in number of ticks greater than 100, then an appropriate value is shown.

- If Num. Ticks is 0, then no ticks are shown on the axis.
- If Num. Ticks is 1, then only the max value tick is shown on the axis.
- If Num. Ticks is 2, then only the min and max value ticks are shown on the axis.
- If Num. Ticks is greater than 2, then evenly spaced ticks (including min and max) are shown on the axis.

Creating 3D Rectangular Bar Plots

This is a 3D graph of results as rectangular bars.



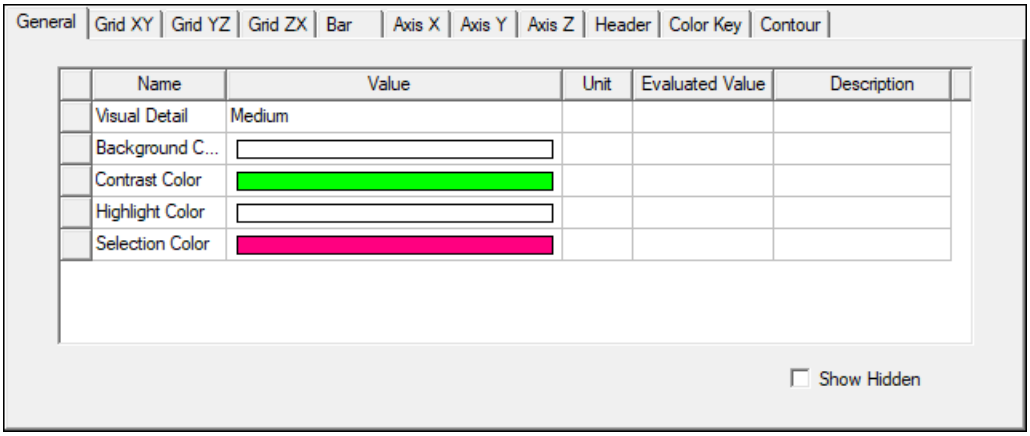
Working with a 3D Rectangular Plot

You can Rotate, Zoom, and Pan a plot. When you rotate, the Cartesian grid responds so that the curve always remains in front and the grids behind.

Clicking on a plot entity selects it, highlighting the selected entity in bold.

Double-clicking anywhere in the plot brings up the **Properties** dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The properties tabs for a 3D Rectangular Bar plot are

- **General**: For general plot properties such as Visual Detail level and background color
- **Grid [XY|YZ|ZX]**: Properties related to the three grids
- **Bar**: Properties related to rectangular bars.
- **Axis [X|Y|Z]**: Properties related to the three axes
- **Header**: Properties related to plot Header/Title.
- **ColorKey**: Properties related to ColorKey, including borders, background, Min and Max, as well as number format and precision.
- **Contour**: Properties related to contouring of all surfaces

Selecting a property also displays its properties in the Property window. You can edit the properties to customize the appearance of the plot. See "Controlling Visual Detail in a 3D Rectangular Bar Plot" on page 26-128.

Creating a 3D Rectangular Bar Plot

1. On the **Results** menu (under the **Maxwell** menu or right-click **Results** in the Project Manager), click **Create *<type>* Report**, and select **3D Rectangular Bar plot** from the report type menu, or select the Results tab of the ribbon, and select the 3D Rectangular Bar icon for the Report type you want to create.

   The *Report* dialog box appears.

2. In the **Context** section make selections from the following field or fields, depending on the design and solution type.
   a. Solution field with a drop-down selection list. This lists the available solutions, whether sweeps or adaptive passes.
   b. Geometry field with a drop-down selection list. For field and radiated field reports, this applies the quantity to a geometry or radiated field setup.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Under the **Trace** tab **Z** Component area, specify the information to plot along the z axis:
    a. In the **Category** list, click the type of information to plot. The category you select provides the default plot name.
    b. In the **Quantity** list, click the value to plot.
    c. In the **Function** list, click the mathematical function of the quantity to plot.
    d. The **Value** field displays the currently specified Quantity and Function. You can edit this field directly.

    > **Note:** Color indicates a valid expression.

    e. **Range Function** button -- opens the *Set Range Function* dialog box. This applies currently specified Quantity and Function.
4. On the **Trace** tab **Y** (Secondary sweep) lines, specify the information to plot along the y-axis in one of the following ways:
    - Select the sweep variable to use from the drop-down list.
    - If sweeps are available, you can select the browse button to display a dialog that lets you select particular values. The quantity will be plotted against the primary sweep variable listed.
5. On the **Trace** tab **X** (Primary sweep) lines, specify the information to plot along the x-axis in one of the following ways:
    - Select the sweep variable to use from the drop-down list.
    - If sweeps are available, you can select the browse button to display a dialog that lets you select particular values. The quantity will be plotted against the primary sweep variable listed.
6. Click **New Report.**

    This creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog box. The default name is based on the Report Category you selected, (for example, S Parameter Plot *n* or rE Plot *n*). You can edit the plot names in the project tree and the plot header text in the report synchronizes.

    The function of the selected quantity or quantities will be plotted against the values you specified on an x-y-z graph. The plot is listed under **Results** in the Project Manager. When you select the traces or plots, axis or grid labels, plot header, color key, or variable labels, their properties are displayed in the Properties window. The properties for each plot element can be edited directly to modify the plot content and appearance. See Modifying the Background Properties of a Report.

7. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

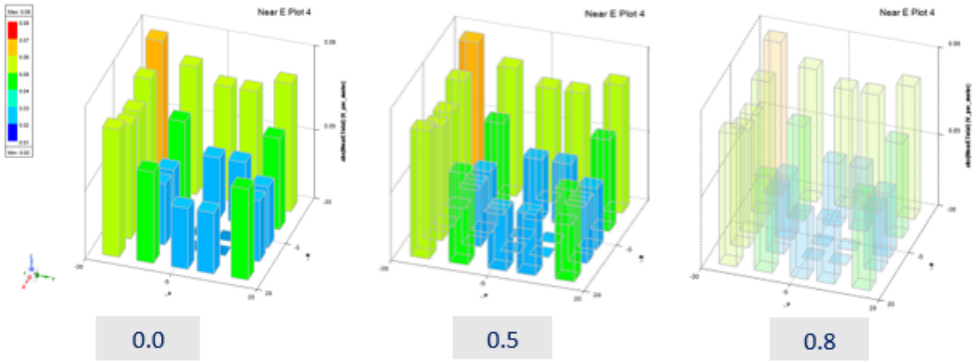
3D Rectangular Bar Customization

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

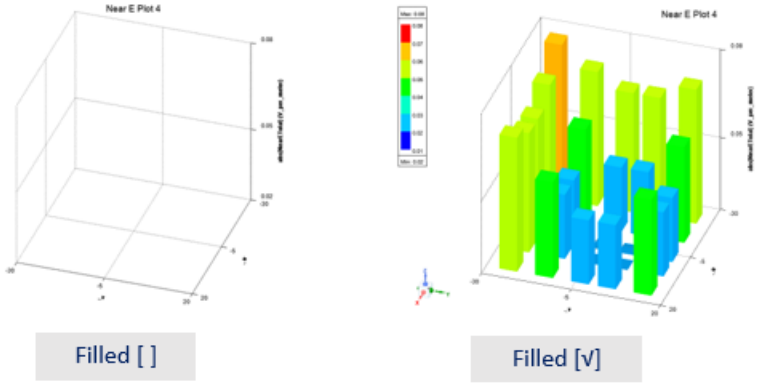
Selecting a bar in the plot shows the customizable attributes of bars in a 3D Rectangular Bar plot.

The customizable attributes of the bars in a 3D Rectangular Bar plot include

- Transparency

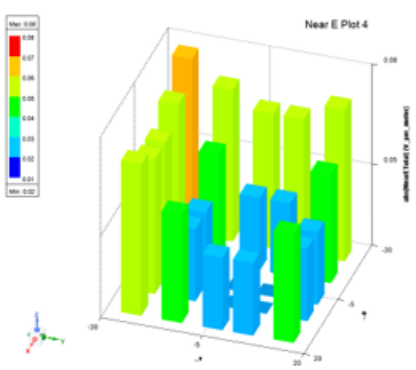


- Filled

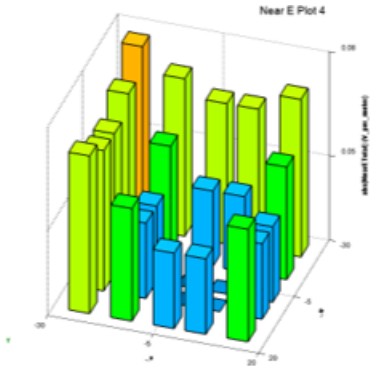


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Show Outline

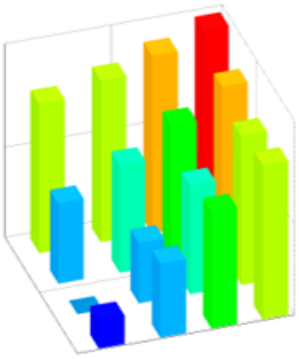


Show Outline []

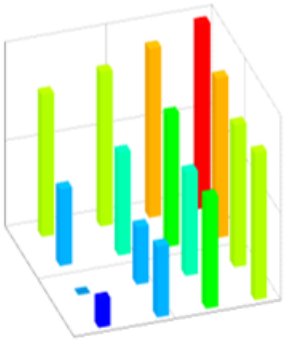


Show Outline [√]

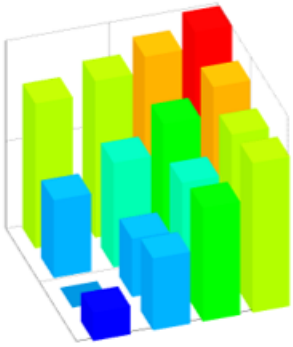
- Outline Color (customizable, see plot above)
- Outline Width (customizable, see plot above)
- Bar Width, controls the thickness of the bar. Medium (default), Wide and Narrow are the width that equals to ½, ¾ and ¼ of the min spacing between adjacent points respectively.



Medium



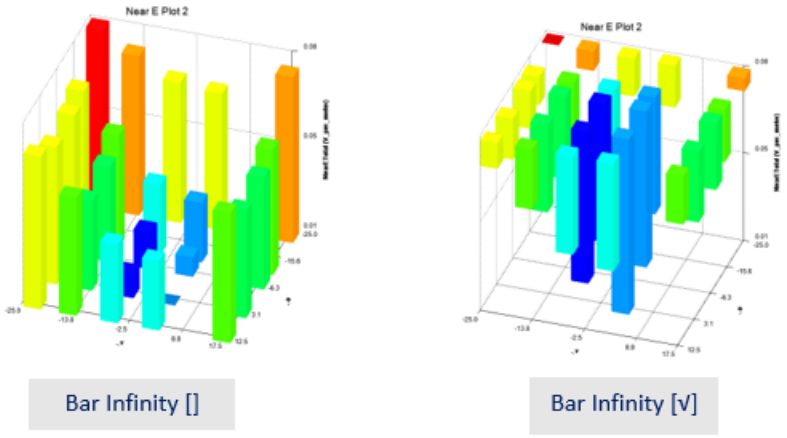
Narrow



Wide

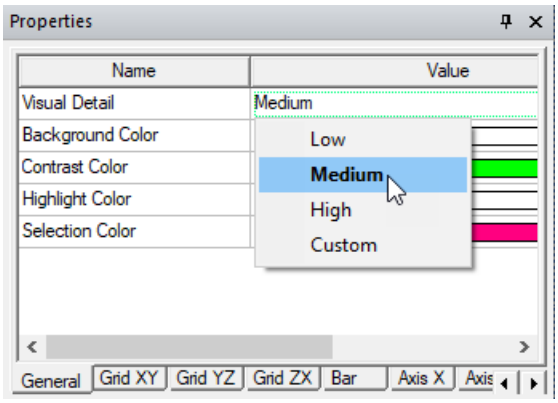
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Bar Infinity represents the third coordinate (z) using the bottom of the bar.



Controlling Visual Detail in a 3D Rectangular Bar Plot

If a particular plot seems busy with information, you can edit plot properties, such as Axis and Grid Attributes for discrete levels of visual detail to improve readability. Double-click anywhere on a plot to display the Properties dialog box. The Visual Detail property on the General tab also provides control suited to different screen and plot sizes.

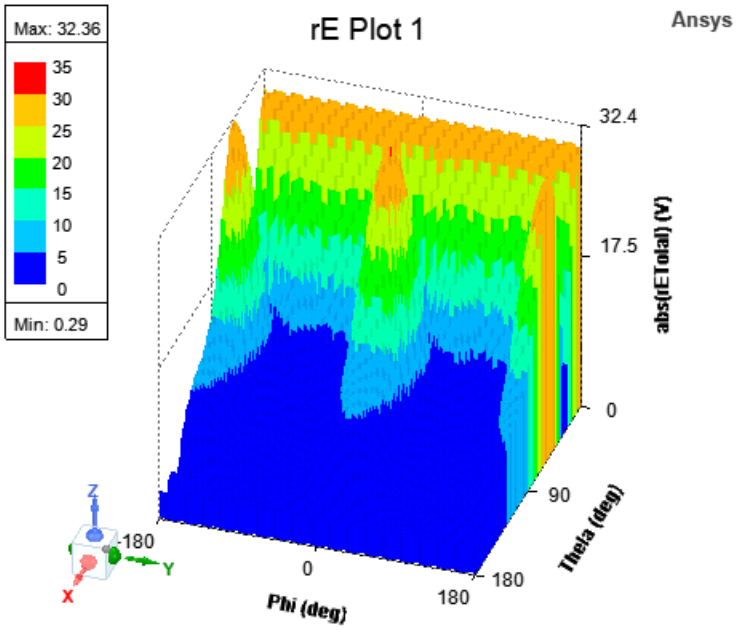


The Visual Detail menu has four options: Low, Medium (the default), High, and Custom. If you select any Visual Detail, the 3D plot is rendered according to the selected Visual Detail level and the properties reflect the values chosen for the selected visual detail level. From this predefined visual detail level, if you modify any properties, Visual Detail is automatically set to Custom (or to another predefined visual detail level if the edits happen to match the settings for that level).

You can also manually set Visual Detail to Custom. In such a case, Custom will inherit property values corresponding to the previous level. This ensures that you can customize settings starting from a baseline provided by the preconfigured Low, Medium or High Visual Detail levels.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

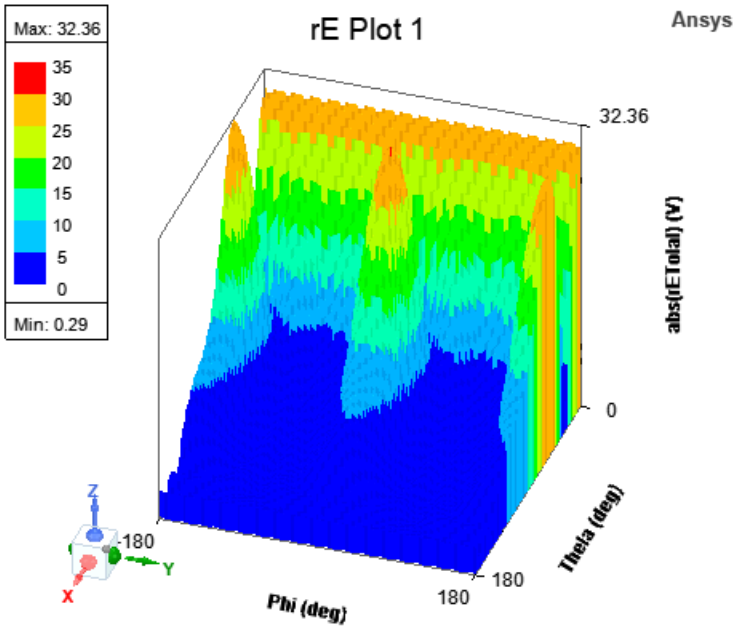
3D Rectangular Plot with Medium Visual Detail



On creation, a 3D Rectangular Plot has Visual Detail set to Medium and looks and feels as shown above. Specifically, under Medium Visual Detail level, a 3D Rectangular Plot has 3 ticks per axis (X, Y, Z axis) which will show min, max and middle value. This setting also shows axes labels.

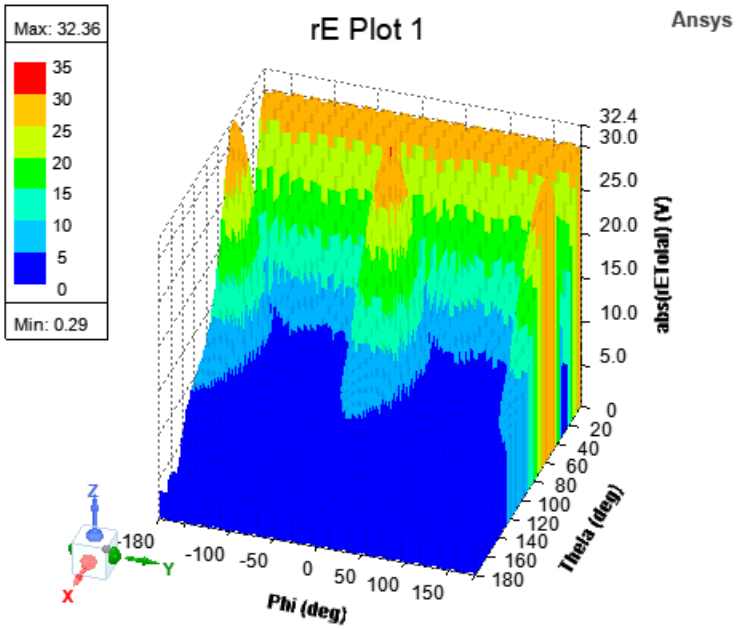
3D Rectangular Bar Plot with Low Visual Detail

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



With Visual Detail set to Low, a 3D Rectangular Plot shows axes with 2 ticks corresponding to min and max values. It also shows axes labels and grid borders.

3D Rectangular Bar Plot with High Visual Detail

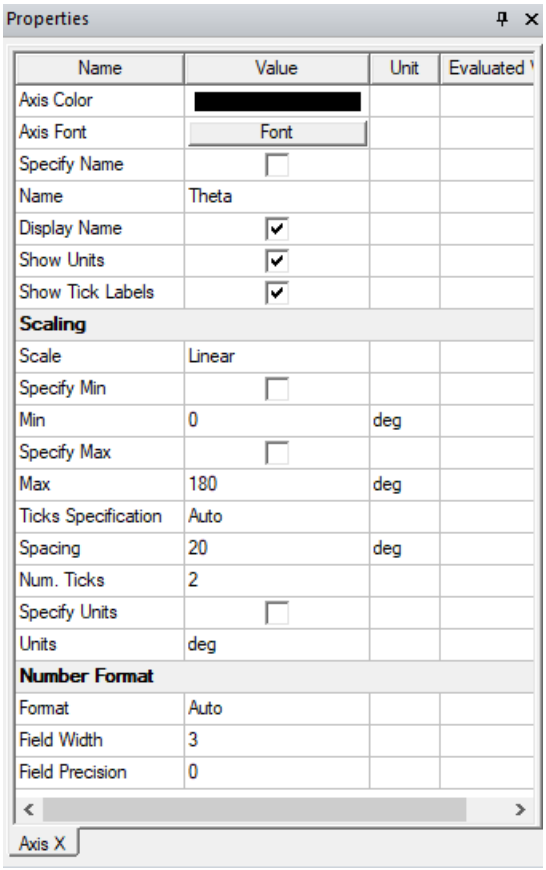


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

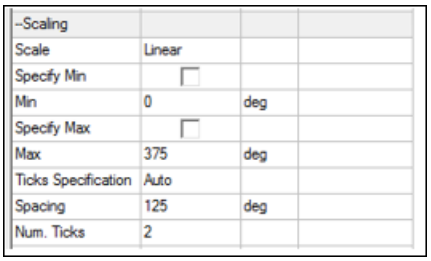
With Visual Detail set to High, a 3D Rectangular Plot shows all Cartesian axes and grids together with all ticks and axes labels.

Axis Properties: Ticks Specification and Num. Ticks

Ticks Specification is available on Axis properties, as shown below:



Ticks Specification is a menu with possible values as Auto, Spacing, and Num. Ticks, with Auto being the default value. If Ticks Specification is Auto, then a spacing value is automatically calculated and used to calculate and display the tick labels. **Spacing** shows the calculated value, and Num. Ticks shows the number of ticks based on this spacing value, as shown below:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can edit the **Spacing** field when Ticks Specification is set to Spacing; otherwise, it is read only.

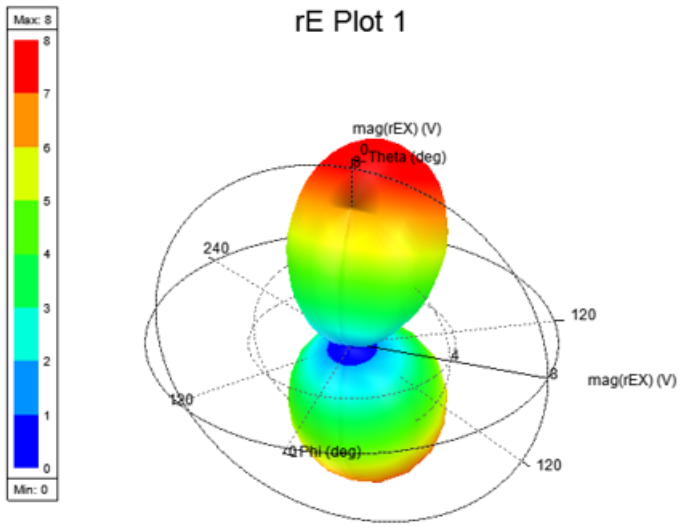
You can edit the **Num. Ticks** field when Ticks Specification is Num. Ticks; otherwise, it is read only.

Valid Num. Ticks are between 0 and 100, including 0 and 100. If you enter an invalid value, an error message is shown. If you enter a spacing value that results in number of ticks greater than 100, then an appropriate value is shown.

- If Num. Ticks is 0, then no ticks are shown on the axis.
- If Num. Ticks is 1, then only the max value tick is shown on the axis.
- If Num. Ticks is 2, then only the min and max value ticks are shown on the axis.
- If Num. Ticks is greater than 2, then evenly spaced ticks (including min and max) are shown on the axis.

Creating 3D Polar Plots in RMxprt

A 3D polar plot is a 3D circular chart divided by the spherical coordinates R, theta, and phi, where R is the radius, or distance from the origin, theta is the angle from the x-axis, and phi is the angle from the origin in the z direction.



Interacting with 3D Polar Plots

You can rotate, zoom and pan a plot.

Clicking on a plot entity selects it, highlighting the selected entity in bold.

Double-clicking anywhere in the plot brings up the Properties window. The properties are grouped appropriately under various tabs, which correspond to plot entities:

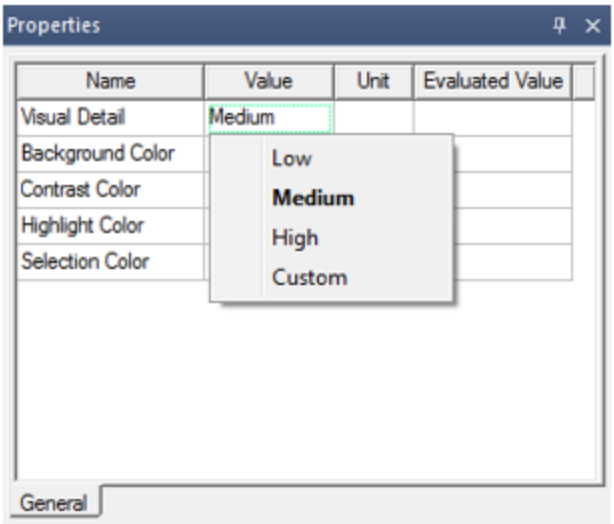
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **General**: For general plot properties such as Visual Detail level and background color
- **Header**: Properties related to plot Header/Title
- **Axis Phi**: Properties related to the circular axis which is in XY plane
- **Axis Theta**: Properties related to the circular axis which is in YZ plane
- **Axis Rho**: Properties related to the radial axis
- **Grid phi-rho-theta(0)**: Properties related to phi-rho grid at theta = 0 (XY plane)
- **Grid theta-rho-phi(90)**: Properties related to theta-rho grid at phi = 90 (YZ plane)
- **Color Key**: Properties related to the color key, including borders, background, Min and Max, as well as number format and precision.
- **Contour**: Properties related to contouring of all curves/surfaces
- **Surface**: Properties related to the curve

Selecting a property also displays its properties in the Property window. You can edit the properties to customize the appearance of the plot. See Controlling Visual Detail in a 3D Plot.

Controlling Visual Detail in a 3D Plot

If a particular plot seems busy with information, you can edit plot properties, such as Axis and Grid Attributes for discrete levels of visual detail to improve readability. Double-click anywhere on a plot to display the Properties window. The Visual Detail property on the **General** tab also provides control suited to different screen and plot sizes.

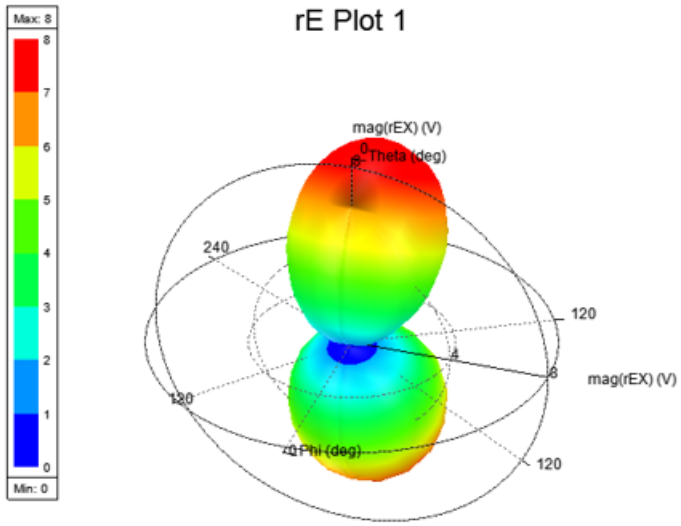


The Visual Detail menu has four options: **Low**, **Medium** (the default), **High**, and **Custom**. If you select any Visual Detail, the 3D plot is rendered according to the selected Visual Detail level and the properties reflect the values chosen for the selected visual detail level. From this predefined visual detail level, if you modify any properties, Visual Detail is automatically set to Custom (or to another predefined visual detail level if the edits happen to match the settings for that level).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

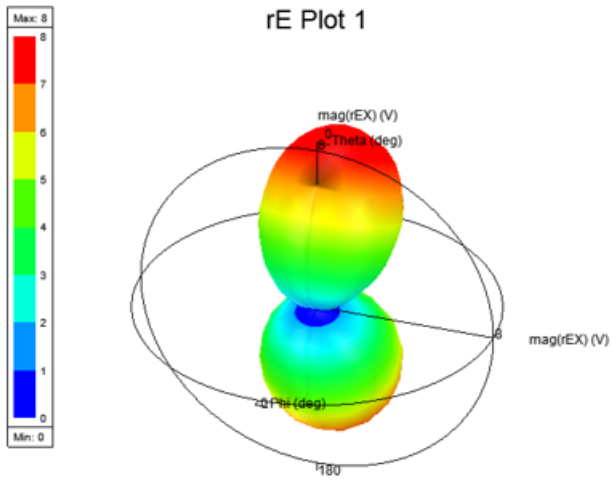
You can also manually set Visual Detail to Custom. In such a case, Custom will inherit property values corresponding to the previous level. This ensures that you can customize settings starting from a baseline provided by the preconfigured Low, Medium or High Visual Detail levels.

3D Polar Plot with Medium Visual Detail



On creation, a 3D Polar Plot has Visual Detail set to Medium and looks and feels as shown above. Specifically, under the Medium Visual Detail level, 3D Polar Plot has 3 ticks per axis (phi, theta and rho axis) which show min, max and middle value. This setting also shows axes labels.

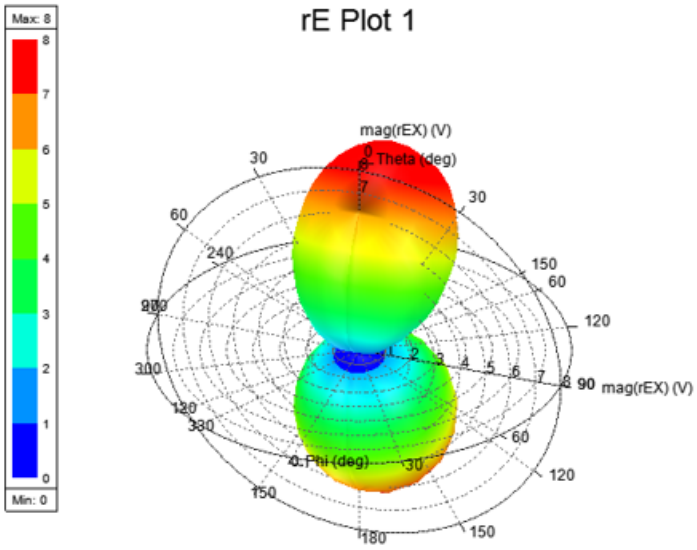
3D Polar Plot with Low Visual Detail



With Visual Detail set to Low, 3D Polar Plot does not show polar grids or grid lines. It only shows axes with 2 ticks corresponding to min and max values. This setting also renders axis labels.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3D Polar Plot with High Visual Detail

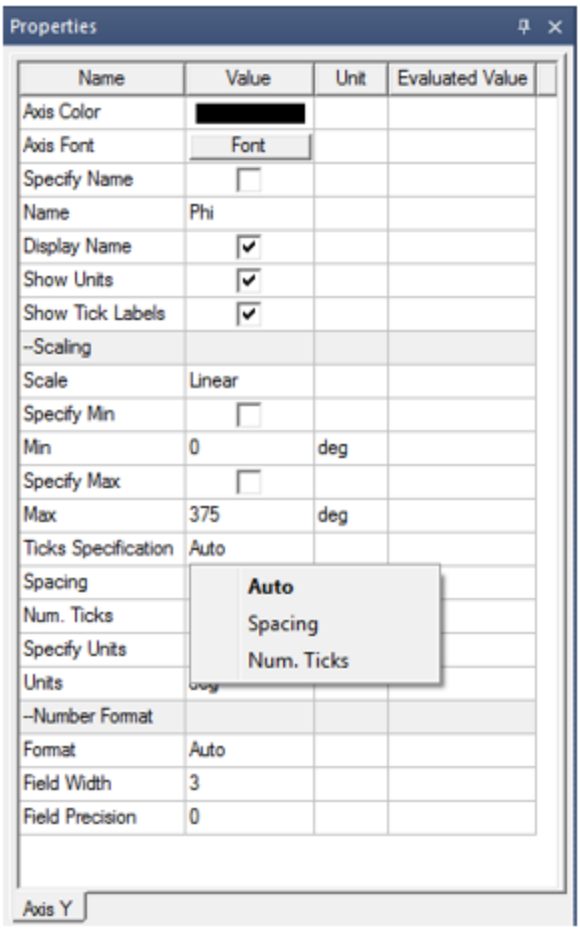


With Visual Detail set to High, a 3D Polar Plot shows all polar axes and grids together with all ticks and axes labels. This is ideal for large plot sizes.

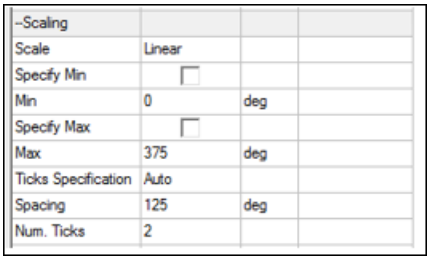
Axis Properties: Ticks Specification and Num. Ticks

Ticks Specification is available on Axis properties, as shown below:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ticks Specification is a menu with possible values as **Auto**, **Spacing**, and **Num**. **Ticks**, with **Auto** being the default value. If Ticks Specification is **Auto**, then a spacing value is automatically calculated and used to calculate and display the tick labels. **Spacing** shows the calculated value, and **Num. Ticks** shows the number of ticks based on this spacing value, as shown below:



You can edit the **Spacing** field when Ticks Specification is set to Spacing; otherwise, it is read only.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can edit the **Num. Ticks** field when Ticks Specification is Num. Ticks; otherwise, it is read only.

Valid Num. Ticks are between 0 and 100, including 0 and 100. If you enter an invalid value, an error message is shown. If you enter a spacing value that results in number of ticks greater than 100, then an appropriate value is shown.

- If Num. Ticks is 0, then no ticks are shown on the axis.
- If Num. Ticks is 1, then only the max value tick is shown on the axis.
- If Num. Ticks is 2, then only the min and max value ticks are shown on the axis.
- If Num. Ticks is greater than 2, then evenly spaced ticks (including min and max) are shown on the axis.

Creating Data Tables in RMxprt

A data table is a grid with rows and columns that displays, in numeric form, selected quantities against a swept variable or other quantities.

1. Click **RMxprt** > **Results** > **Create RMxprt Report** > **Data Table**.

   The **Report** dialog box appears.

2. In the **Context** section make selections from the following field or fields, depending on the design and solution type.
   a. Solution field with a drop down selection list. This lists the available solutions, whether sweeps or adaptive passes.
   b. Domain field with a drop down selection list.
3. Under the **Trace** tab, **Y** component section, select the quantity you are interested in and its associated function:
4. Under the **Y** tab, select the quantity you are interested in and its associated function:
   a. On the **Category** drop down list, click the type of information to plot.
   b. On the **Quantity** list, click the values to plot. Use Ctrl + click to make multiple selections.
   c. In the **Function** list, click the mathematical function to use for the quantity.
   d. The **Value** field displays the currently specified Quantity and Function. You can edit this field directly. **Note**

      > **Note:** Color indicates that an expression is valid.

   e. The **Range Function** button opens the **Set Range Function** dialog box, which applies currently specified Quantity and Function.
5. On the **Trace** tab, **X (Primary Sweep) line,** select the sweep variable from the drop down list. By default **All** of the chosen sweep's values are used. You can also select the browse

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[...] button to display a dialog box that lets you select particular sweep values, specify a range of sweep values (for Time sweeps), or **Use all values** (the default setting).

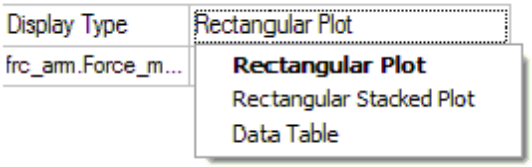
6. On the **Families** tab, confirm or modify the sweep variables that will be plotted.
7. Click **New Report**.

   This creates a new report in Project tree, displays the report with the defined trace, and enables the **Add Trace** button on the **Report** dialog box. The default name is based on the Report Category you selected, for example, Force Plot *n* or Output Variables Plot *n*. You can edit the plot names in the project tree and the plot header text in the report synchronizes.

   The Y quantity will be listed at each variable value or additional quantity value you specified. The data table is listed under **Results** in the project tree. The plot is listed under **Results** in the project tree and the traces are listed under the plot. When you select the traces or plots, their properties are displayed in the Properties window. These properties can be edited directly to modify the plot.

8. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

You can also modify the display type of an existing plot from the Properties dialog for that plot. Select the Report icon in the Project tree to display the Properties dialog box. Selecting the Display Type field displays a menu with selections available for that plot.



Once you make a selection, the plot display updates for the current selection.

If you choose to print a data table:

- Selecting print "All" prints the whole table for current data page (if there is more than one data page).
- Selecting print "Pages" prints user-specified pages.
- If the table is bigger than the screen view (that is, it has a scroll bar), printing first scrolls right, prints until no more scrolling occurs, and then scrolls down.
- The Page number appears at the bottom of the page, aligned at center.
- The table layout of each page follows the screen, but without the scroll bar being printed, and no data page bar as on screen.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Working with Traces in RMxprt

A trace in a 2D or 3D report defines one or more curves on a graph. A trace in a data table defines part of the displayed matrix of text values.

The values used for a plot's axes can be variables in the design or functions and expressions based on the design's solutions. If you have solved one or more variables at several values, you can "sweep" over some or all of those values, resulting in a curve in 2D or 3D space.

A report can include any number of traces and, for rectangular graphs, up to four independent y-axes.

In general, to add a trace to a report:

1. In the **Traces** dialog box, specify the information you want to plot along the appropriate axes.
2. Click **Add Trace**.

A trace is added to the traces list at the top of the **Traces** dialog box. The trace represents the function of the quantity you selected and will be plotted against other quantities or swept variable values. Each column lists an axis on the report and the information that will be plotted on that axis.

You can modify the information to be plotted by typing the name of the quantity or sweep variable to plot along an axis directly in the boxes.

The trace will be visible in the report when you click **Done**.

> **Note:** The **Traces** dialog box can be accessed via the **Create Report** dialog box. You can also Add Trace Characteristics to 2D reports.

Removing Traces in RMxprt

You can traces from the traces list in the following ways:

To remove one trace from the report, select the trace you want to remove from the traces list, and then click **Remove Trace**.

To **remove all traces** from the report, click **Remove All Traces**.

Replacing Traces in RMxprt

To replace a trace in the traces list with a different trace definition:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the trace you want to remove from the traces list.
2. In the **Traces** dialog box, specify the information you want to plot along the appropriate axes.
3. Click **Replace Trace**.

   The trace you selected is removed, and the new trace information you specified replaces it in the traces list.
4. Click **Done**.

Adding Blank Traces in RMxprt

To add a blank trace to the traces list, click **Add Blank Trace**.

You can now type the quantities to plot in the appropriate axes boxes.

Sweeping a Variable in a Report in RMxprt

In RMxprt, a swept variable is an intrinsic, project, or design variable that typically has more than one value. From the **Traces** dialog box, you can plot any calculated or derived quantity against one or more of the swept variable's values.

When you click the Sweeps tab in the **Traces** dialog box, the first sweep variable listed is the "primary sweep". If you are creating a 3D report, the second sweep variable listed is the "secondary sweep". Any additional sweep variables are represented as additional curves on the graph.

To modify which variable is the primary sweep variable, click the **Name** box for the primary sweep variable, and then click the variable name you want to be the primary sweep variable.

To modify the secondary sweep variable or any additional sweep variable, follow the same procedure.

To modify the values that will be plotted for a variable:

1. Click a variable in the table.

   To the right, all of the possible values for the selected variable are listed.
2. Select **All Values**.

   All of the selected variable's values are plotted.

   Alternatively, clear **All Values** and select the specific values to plot against the selected quantity.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting a Function in RMxprt

The value of a quantity being plotted depends upon its mathematical function, which you select from the **Function** list in the **Report** dialog box. The available, valid functions depend on the type of quantity (real or complex) that is being plotted. The function is applied to the quantity which is implicitly defined by all the swept and current variables. For example, "Volume(Stator)" is the volume of the model object named Stator for every swept combination of variables ("height" and so forth).

These functions can also be applied to previously specified Quantities and Functions as **Range Functions** when using the Set Range Function dialog.

Some of these functions can operate along an entire curve. These are: min, max, integ, rms, pk2pk, cang_deg and cang_rad.

You can select from the following functions in the **Trace** tab **Function** list, or type them directly into the Y or X field, if necessary:

abs	Absolute value of the simulation quantity which results in a number that is always positive.
acos	Arc cosine i.e. the inverse function of a cosine.
acosh	Inverse hyperbolic arc cosine
ang	Magnitude of an angle
ang_deg	Angle (phase) of a complex number, cut at +/-180.
ang_rad	Angle in radians
arg	Argument of a complex number. It is the angle the complex number makes with the positive x axis. Same as **ang_deg**.
asin	Arc sine, *i.e.*, inverse function of sine
asinh	Inverse hyperbolic sine
atan	Arc tangent. Takes a tangent value as an argument. Because there are two angles in a circle that can have the same tangent value, and atan can return only one value, it returns a value in the range between -90 degrees and +90 degrees (or between -pi/2 and pi/2 in radians).
atanh	Inverse hyperbolic tan.
atan2	A two-argument version of the **atan** function. Takes the y and x coordinates (including sign information) of a point as arguments and returns the angle from the X-axis. Can return angle values for the full circle (-180 degrees to +180 degrees or -pi to +pi in radians).
avg	Returns the average of the values of the selected quantity. avg = (Area between the curve and the X-axis) / (X length of the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	curve)
avgabs	Returns the mean of the absolute value of the selected quantity
bandwidth	Returns the 3dB bandwidth of the selected simulation quantity. For bandwidth, the calculation is based on 3dB below the maximum peak.
cang_deg	Cumulative angle (phase) of the first parameter (a complex number) in degrees, along the second parameter (typically sweep variable). Returns a double precision value cut at +/-180.
cang_rad	Cumulative angle of the first parameter in radians along a second parameter (typically a sweep variable). Returns a double precision value.
cmplx(*re*, *im*)	A complex number, where *re* is the real part and *im* is the imaginary part.
conjg	Conjugate of the complex number
cos	Cosine
cosh	Hyperbolic cosine
crestfactor	Returns the crest factor (peak/RMS) for the selected quantity
cum_integ	The cumulative integral function returns a set of values that have the same length as the original set of points (the first element will always be zero). Element *I* of the set returned by cum_integ is the integral of elements 1 through *I* of the original data set.
cum_sum	The cumulative sum function returns a data set that has the same length as the original set of points. Element *I* of the set returned by cum_sum is the sum of elements 1 through *I* of the original data set.
dB(x)	20*log10(|x|) to base 10.
dBc	Decibels relative to the carrier. It is the power ratio of the signal to a carrier signal. Gives the relative signal strength.
dBm(x)	10*log10(|x|) +30
dBW(x)	10*log10(|x|)
dB10	10*log(|x|) to base 10
dB10normalize	10*log [normalize(mag(x))]
dB20	20*log(x) to base 10
dB20normalize	20*log [normalize(mag(x))]
deadtime	Obtains the latest time when the qtyl is within a tolerance of zero
delaytime	Obtains the time from zero to 50% of the target point
degel	Conversion from degrees electrical to seconds with respect to Hz.
deriv	Derivative of a given parameter
distortion	Returns the total distortion for the selected simulation quantity and

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	an additional argument frequency, which is the frequency in Hz at which to calculate the fundamental RMS of the simulation quantity.
even	Returns 1 if integer part of the number is even; returns 0 otherwise.
exp	Exponential function (the natural anti-logarithm) of the simulation quantity.
formfactor	Returns the form factor (RMS/Mean Absolute Value) for the selected quantity.
fundamentalmag	Returns the RMS value of the fundamental frequency for the selected quantity, and an additional argument, Frequency, which specifies the fundamental frequency.
gaincrossover	Returns the gain crossover frequency (where the gain is 0 dB) of the selected simulation quantity in Hz.
gainmargin	Returns the gain margin in dB at the phase crossover frequency of the selected simulation quantity. It also requires a reference simulation quantity to which the measured quantity is compared and the AC magnitude and phase angle of the reference quantity. These are entered as the arguments Reference Channel, Base Source Magnitude, and Base Source Angle.
iae	Returns the integral of the absolute deviation of the selected quantity from a target value that is entered via the additional argument. To use this function, you need to open the Add Trace Characteristics dialog and select the Error category.
if	if(cond_exp,true_exp, false_exp).
im	Imaginary part of the complex number
int	Truncated integer function
integ	Integral of the selected quantity. Uses trapezoidal area.
integabs	Absolute value of integral
ise	Returns the integral of the squared deviation of the selected quantity from a target value that is entered via an additional argument. To use this function, you need to open the Add Trace Characteristics dialog and select the Error category.
itae	Returns the time-weighted squared deviation of the selected quantity from a target value that is entered via an additional argument. To use this function, you need to open the Add Trace Characteristics dialog and select the Error category.
itse	Returns the time-weighted squared deviation of the selected quantity from a target value that is entered via an additional argument. To use this function, you need to open the Add Trace Characteristics dialog and select the Error category.
j0	Bessel function of the first kind (0^{th} order).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

j1	Bessel function of the first kind (1st order).
jn	Bessel function of the first kind (nth order).
ln	Natural logarithm
log	Natural logarithm (same as ln)
log10	Logarithm base 10
lowercutoff	Returns the lower 3 dB frequency of the selected simulation channel in Hertz.
lsidelobeX	The 'x' value for the left side lobe: the next highest value to the left of the max value.
lsidelobeY	The 'y' value for the left side lobe: the next highest value to the left of the max value.
mag	Magnitude of the complex number
max	Returns maximum value of the simulation quantity.
max_swp	Returns maximum value of a sweep
max2	Maximum value of the two simulation quantities. For example, **max2(a,b)** will plot maximum of **a** and **b** for a particular instance.
mean	Returns the average in the set of quantities selected. mean = sum(all y-value) / (number of y-values)
min	Returns the minimum value of the simulation quantity
min_swp	Returns the minimum value of a sweep
min2	Minimum value of the two simulation quantities. For example, **min2 (a,b)** will plot minimum of **a** and **b** for a particular instance.
mod	Returns the modulus or absolute value of the simulation quantity
nint	Nearest integer
none	Returns null value
normalize	Divides each value within a trace by the maximum value of the trace. ex. normalize(mag(x)).
odd	Returns 1 if integer part of the number is odd; returns 0 otherwise.
overshoot	Calculates peak overshoot given a threshold value and number of evenly spaced points over entire time range.
peakgain	Returns the peak value of gain of the selected simulation quantity in dB.
peakgainfreq	Returns the frequency in Hz at which the peak gain of the selected simulation quantity occurs.
polar	Coverts the complex number in rectangular coordinates to polar coordinates.
per	Returns the period of a simulation quantity.
phasecrossover	Returns the phase crossover frequency, at which the phase is -180

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	degrees, in Hz for the selected simulation quantity.
phasemargin	Returns the phase angle in degrees at the gain crossover frequency of the selected simulation quantity.
pk2pk	Peak to peak. Difference between max and min of the first parameter over the second parameter. Returns the peak-to-peak value for the selected simulation quantity.
pkavg	Returns the ratio of the peak to peak-to-average for the selected quantity.
pmax	Maximum period of the selected simulation quantity.
pmin	Minimum period of the selected simulation quantity.
pow	Raises x to the power of y; pow(x,y).
prms	Period Root Mean Square
pulsefall9010	Returns the pulse fall time of the selected quantity according to the 90%-10% estimate.
pulsefront1090	Returns the pulse front time of the selected quantity according to the 10%-90% estimate.
pulsefront3090	Returns the pulse front time of the selected quantity according to the 30%-90% estimate.
pulsemax	Returns the pulse maximum from the front and tail estimates for the selected quantity.
pulsemaxtime	Returns the time at which the maximum pulse value of the selected quantity is reached.
pulsemin	Returns the pulse minimum from the front and tail estimates for the selected quantity.
pulsemintime	Returns the time at which the minimum pulse value of the selected quantity is reached.
pulsetail50	Returns the pulse tail time of the selected quantity from the virtual peak to 50%.
pulsewidth5050	Returns the pulse width of the selected quantity as measured from the 50% points on the pulse front and pulse tail.
pwl	Piecewise Linear
pwl_periodic	Piecewise Linear for periodic extrapolation on x
pwlx	Piecewise Linear x with linear extrapolation on x
pw_minus	Pulse width of the first negative pulse
pw_minus_avg	Returns the average of the negative pulse width input stream.
pw_minus_max	Returns the maximum pulse width of the negative pulse of input stream.
pw_minus_min	Returns the minimum pulse width of the negative pulse of input stream.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

pw_minus_rms	RMS of the negative pulse width input stream.
pw_plus	Pulse width of the first positive pulse.
pw_plus_avg	Average of the positive pulse width input stream.
pw_plus_max	Max. Pulse width of the positive pulse of input stream.
pw_plus_min	Min. Pulse width of the positive pulse of input stream.
pw_plus_rms	RMS of the positive pulse width input stream.
re	Real part of the complex number.
rect	Converts the complex number in polar to rectangular co-ordinates.
rem	Fractional part of the selected simulation quantity i.e. remainder.
ripple	Returns the ripple factor (AC RMS/Mean) for the selected quantity.
risetime	Obtains the time taken to go from 10% to 90% of target point.
rms	Returns the root mean square value of the selected quantity.
rmsAC	Returns the AC RMS for the selected quantity.
root	nth root function.
rSidelobeX	Returns the X value of right side-lobe occurrence.
rSidelobeY	Returns the Y value of right side-lobe occurrence.
settlingtime	Returns the latest time at which the value of the selected simulation quantity fell outside its tolerance band. The target value of the quantity and the +/- bandwidth of the tolerance band are the additional args.
sgn	Sign extraction
sin	Sine
sinh	Hyperbolic sine
slidingmean	Returns the moving average value of the selected simulation quantity (specified by the first argument). The average is calculated over a period (specified by the second argument).
slidingrms	Returns the moving RMS value of the selected simulation quantity (specified by the first argument). The RMS value is calculated over a period (specified by the second argument).
sqr	Square of the selected simulation quantity
sqrt	Square root of the selected simulation quantity
stddev	Returns the standard deviation of given values
sum	Returns the sum of the given values
tan	Tangent
tanh	Hyperbolic tangent
undershoot	Calculates peak undershoot given a threshold value and number of evenly spaced points over entire time range.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

uppercutoff	Returns the upper 3 dB frequency of the selected simulation channel in Hz
variance	Calculates the variance of the given values
XAtYMax	Threshold crossing time: report first time (x value) at which an output quantity crosses YMax.
XAtYMin	Threshold crossing time: report first time (x value) at which an output quantity crosses a user definable threshold.
XAtYVal	Returns the X value at the first occurrence of Y value
XWidthAtYVal	Returns the X width between the first two occurrence of Y value
xdb10beamdwidth	Width between left and right occurrences of values 'x' db10 from max. Takes 'x' as argument (3.0 default). To use this function, you need to open the Add Trace Characteristics dialog and select the Radiation category.
xdb20beamwidth	Width between left and right occurrences of values 'x' db20 from max. Takes 'x' as argument (3.0 default). To use this function, you need to open the Add Trace Characteristics dialog and select the Radiation category.
YAtXMax	Returns the X value at maximum value of Y
YAtXMin	Returns the Y value at minimum value of X
YAtXVal	Returns the Y value at the first occurrence of X value
y0	Bessel function of the second kind (0^{th} order)
y1	Bessel function of the second kind (1^{st} order)
yn	Bessel function of the second kind (nth order)

Selecting a Parameter, Variable, or Quantity to Plot in RMxprt

Each trace in a report includes a quantity that is plotted along an axis. The quantity being plotted can be a value that was calculated by RMxprt, such as L_{11}, a value from a calculated expression.

To select a parameter, variable, or quantity to plot:

1. In the **Traces** dialog box, select one of the following categories:

Variables	User-defined project or design variables.
Output Variables	Derived quantities RMxprt project or design variables, parameters or solution curves.

2. Select a quantity to plot from the **Quantity** list. The available quantities depend on the selected category and the setup of the design.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating Quick Reports in RMxprt

Following is the procedure for creating a quick report.

1. On the Project Manager tree, select a **setup** or **sweep icon** of interest.
2. Right-click to display the shortcut menu and select **Quick Report**.

   The **Quick Report** dialog appears.

3. Select the one or more categories for the report from the list and click **OK**.

   A rectangular plot for each selected category displays. The new plot or plots appear in the Project tree under the Results icon. The default Report Name that appears is derived from the report category specified in the **Quick Report** dialog box. You can edit the plot names in the project tree and the plot header text in the report synchronizes.

RMxprt Quick Report Categories

When using the Quick Reports function for Solutions, the following report categories may be available depending upon the solution parameters requested, solution type, *etc*.:

Category	Description
Coil Voltage	Reports voltages in the machine coil
Current	Reports currents for each line or phase of the machine, source current, line current, armature current
Flux Density	Reports flux density in the machine air gap, flux linkages
Induced Voltage	Reports Induced Line and Phase voltages
Misc.	Reports miscellaneous quantities specific to the machine type such as power factor, torque to current ratio
Percentage	Reports machine efficiency
Power	Reports air gap power, output power
Torque	Reports cogging torque, output torque, magnet generated torque, induction torque
Voltage	Reports Line and Phase voltage
Angle	Reports power factor angle
Angular Speed	Reports angular speed
Inductance	Reports air gap permanence

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specifying RMxprt Winding Data

To define the winding data for an RMxprt machine:

1. In the Project Manager tree, under **Machine**, open the folder that requires a winding, and double-click **Winding** to open the winding **Properties** window.
    - For some machine types this would be **Machine** > **Rotor** > **Winding**, for others, **Machine** > **Stator** > **Winding**.
    - You can also enter values in the **Properties** section of the desktop without opening a separate window.
2. Specify the desired settings.
3. Click **OK** to close the **Properties** window.

The specific properties available depend on the specific machine.

The following machine types have winding data available:

- Three-Phase Induction Motors (stator winding) and (rotor winding)
- Three-Phase Synchronous Machines (stator winding) and (rotor winding)
- Brushless PMDC Motors (stator winding)
- Adjust-Speed Synchronous Machines (stator winding)
- PMDC Motors (rotor winding)
- Switched Reluctance Motors (stator winding)
- Line-Start Permanent-Magnet Synchronous Motors (stator winding)
- Universal Motors (stator winding) and (rotor winding)
- General DC Machines (rotor winding)
- Claw-Pole Alternators (stator winding)
- Generic Rotating Machines

Setting the Winding Type

The RMxprt application can automatically arrange almost all commonly used single- or double-layer poly-phase ac windings provided all coils have the same number of turns. Users do not need to define coils one by one. For a double-layer winding, the RMxprt application can also handle the coils with half turns which are arranged in the order of even, odd, even, odd, …, as long as it is physically possible.

The RMxprt application also provides a very flexible tool **Winding Editor** in order for the users to design a variety of special winding types according to their own needs, such as compound single- and double-layer winding, big- and small-phase-spread variable-pole multiple-speed winding, sine-wave three-phase winding, and so forth. The **Winding Editor** is available to the following types of electric machines:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Three-phase induction motors
2. Single-phase induction motors
3. Three-phase synchronous motors and generators
4. Line-start permanent-magnet synchronous motors
5. Claw-pole alternators
6. Adjustable-speed permanent-magnet synchronous motors and generators
7. Brushless permanent-magnet DC motors

When you edit the AC winding of a new design for the first time, the RMxprt application creates a default winding arrangement based on the basic winding specifications: **Number of Phases**, **Number of Poles**, **Number of Slots**, **Winding Layers**, **Conductors per Slot**, and **Coil Pitch**. Then you can edit the winding configuration based on the default arrangement.

Winding Types Available for Machines

Use the **Winding Type** dialog box to set the Winding type.

1. To display the **Winding Type** dialog box, double-click on the **Winding** property button.

   Passing the cursor over the buttons for the Winding types changes the graphic to show the available windings for the motor in the design. Choices differ depending on the motor. A Winding Editor selection does not have a graphic.

   Selections for the Three Phase Induction Motor (stator winding) and (rotor winding), Three-Phase Synchronous Machine (stator winding) and (rotor winding), Brushless Permanent Magnet DC Motor (stator winding), Adjust Speed Synchronous Machine (stator winding), Line Start PM Synchronous Motor (stator winding), and Claw Pole alternator (stator winding) include

   - Editor - enables the Winding Editor
   - Whole Coiled
   - Half Coiled

   Selections for the DC Permanent Magnet Motor (rotor winding) and Universal Motor include (stator winding) and (rotor winding):

   - Lap
   - Wave

   Selection for the General DC Machines (rotor winding) include:

   - Lap
   - Wave
   - Frogleg

   Selections for single-phase induction motor include

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Editor - enables the Winding Editor
- Lap - 90 deg phase belt 2-layer coil for both single and double layer
- Sin_1 - first class sinusoidal coil for double layer only
- Sin_2 - second class sinusoidal coil for double layer only

The Switched Reluctance motor does not involve winding selections.

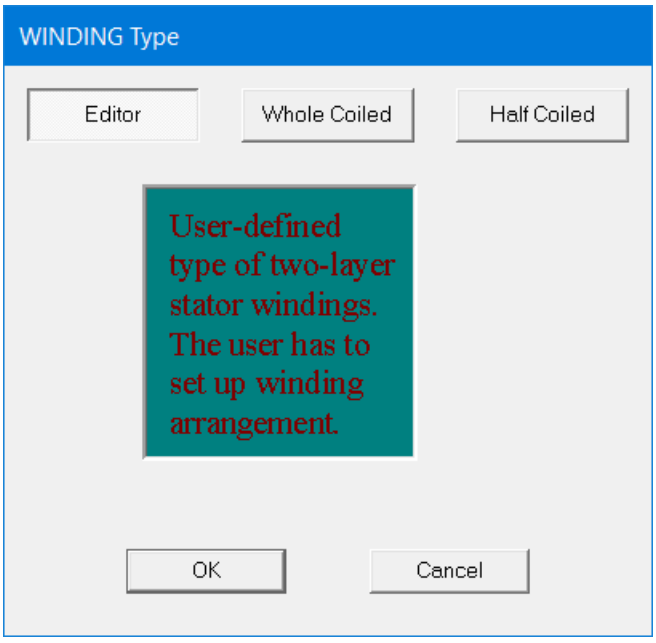
2. Select the **Winding Type** and click **OK**.

   This closes the dialog box and sets the Winding Type property. If you select the Editor type, It also enables the **Machine** > **Edit Layout** command on the menu bar.

Enable the Winding Editor

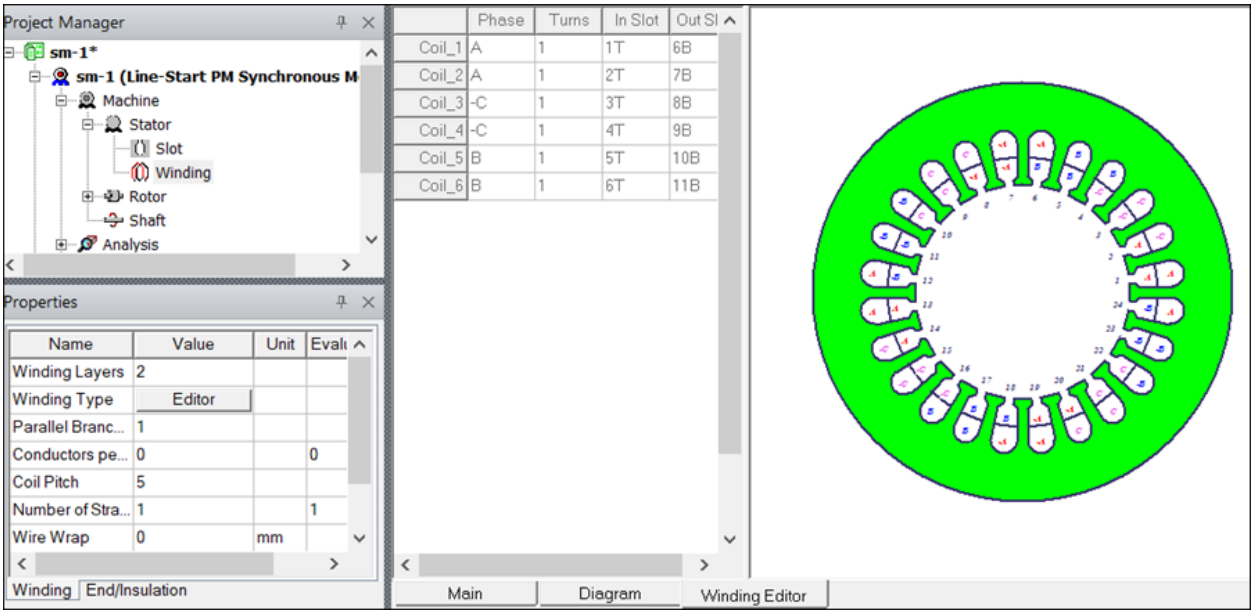
Setting the **Winding Type** property to **Editor** enables the command **Machine** > **Edit Layout** on the menu bar. To display the Winding Editor:

1. Double-click on the button Winding Type value to display the Winding Type selection window.

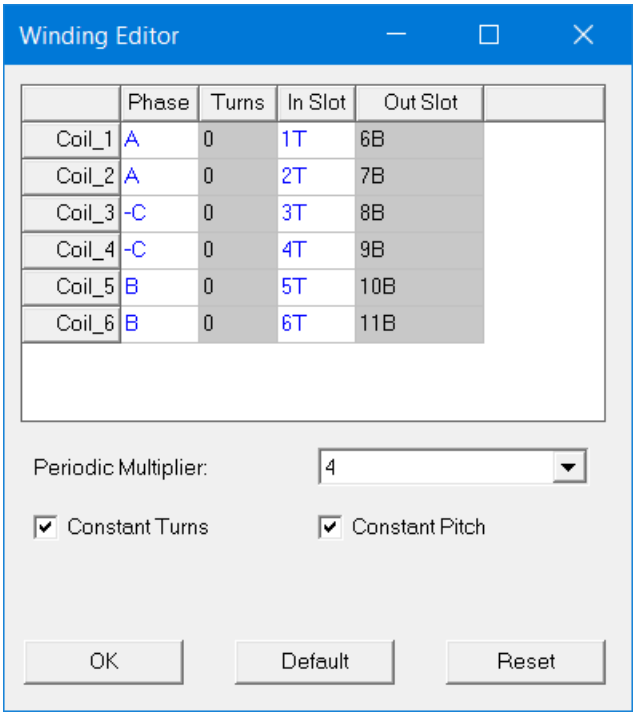


2. Select **Editor** as the Winding Type and click **OK**. This closes the Winding Type selection window and sets the Winding Type Value to Editor. It also enables the command **Machine** > **Winding** > **Edit Layout** on the menu bar. Now the Machine Editor window displays the default winding arrangement.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Click **Machine** > **Winding** > **Edit Layout**. This displays the Winding Editor dialog box as shown. The Winding Editor includes functions that do not appear in the Winding Editor tab sheet in the RMxprt Machine Editor window.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Edit Winding Configuration

Each row of the winding data table in the Winding Editor dialog box is identified with the coil index in the column Coil. This information is displayed in the tab sheet Winding Editor in the RMxprt Machine Editor window as well, but it is editable in the dialog box Winding Editor.

The winding data table contains four columns:

Phase	Defines the phase to which the coil belongs
Turns	Defines the number of turns of the coil
In Slots	Defines the slot number with the coil side current flowing in ('flow-in-side' for short). If two layers are specified in the Winding Properties window, the slot number ends with a "T" to show the top layer.
Out Slots	Defines the slot number with the coil side current flowing out ("flow-out-side" for short). If two layers are specified in the Winding Properties window, the slot number ends with a "B" to show the bottom layer.

Setting the Number of Winding Layers

To set the number of winding layers:

1. Open the **Winding Properties** window by double-clicking on the **Winding** icon in the properties window.
2. Use the drop-down list in the Winding Layers field to set the number as 1 or 2.

   This sets the winding layers used in the winding.

The number of layers selected makes a difference in the display of data in the **Winding Editor**.

Connecting and Disconnecting Windings

When you have specified the winding data, you can execute the following commands to automatically connect or disconnect the windings.

- **Machine** > **Winding** > **Connect All Coils**

  Upon executing, the graphical display in the main window shows the connections.
- **Machine** > **Winding** > **Disconnect All Coils**

Upon executing, the graphical display in the main window updates to remove the connection.

Polyphase Winding Editor

RMxprt provides a **Winding Editor** in order for users to design variety of special winding types according to their own needs, such as compound single- and double-layer winding, big- and

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

small-phase-spread variable-pole multiple-speed winding, sine-wave three-phase winding, and so forth. The **Winding Editor** is available to the following types of electric machines:

- Three-phase Induction Motor
- Three-phase Synchronous Motor
- Three-phase Synchronous Generator
- Permanent-magnet Synchronous Generator
- Line-start Permanent-magnet Synchronous Motor
- Adjustable-speed Permanent-magnet Synchronous Motor
- Brushless Permanent-magnet DC Motor
- Claw-pole Alternator

You input data for **Number of Poles** in **the Machine Properties** window and data for the **Number of Slots** and Slot Type in the **StatorProperties** window. You set the **Number of Slots** in the **Winding Properties** window. **RMxprt** automatically arranges the winding layout and display the relevant information that has been specified in the **Winding Editor** tab of the **RMxprt** main window. As long as the edited winding data have been saved, the **Winding Editor** tab will display the last saved winding data whenever **Winding Editor** dialog is launched. The left top part of the **Winding Editor** tab shows the winding data, as does the Winding Editor dialog box. In this area, the total number of rows equals half the number of slots.

Enabling the Winding Editor

Setting the **Winding Type** property to **Editor** enables the **Machine** > **Edit Layout** command on the menu bar. To display the **Winding Editor**:

1. Open the **Winding Properties** window and set the **Winding** property to **Editor**.

   To do this, double-click on the **Winding** property button to display the **Winding Type** selection window.

2. Select **Editor** as the **Winding Type** and click **OK**.

   This closes the **Winding Type** window and sets the Winding Type property to Editor. It also enables the **Machine** > **Edit Layout** command on the menu bar.

3. Click **Machine** > **Edit Layout**.

   This displays the **Winding Editor** window. The **Winding Editor** window includes functions that do not appear in the **RMxprt** main window **Winding Editor** tab.

   Other ways to invoke the Winding Editor window:

   a. Right-click on the data table section of the Winding Editor tab of the main window. This displays an **Edit Layout** button.
   b. Click the **Edit Layout** button to display the Winding Editor window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Or you can right-click in the **Winding Editor** tab main window display, and select **Edit Layout** from the context menu.

Each row of the winding data is identified with coil index in the **Coil** column. This information is displayed in the **Winding Editor** tab in the RMxprt Main window, and it editable in the **Winding Editor** window.

- Column **Phase** is for the phase to which the coil belongs.
- Column **Turns** is for the number of turns of coil.
- Column **In Slots** is for the slot number with the coil side current flowing in ('flow-in-side' for short). If 2 Layers are specified in the Winding Properties window, the slot number ends with a "T" to show the top layer.
- Column **Out Slots** is for the slot number with the coil side current flowing out ('flow-out-side' for short). If 2 Layers are specified in the Winding Properties window, the slot number ends with a "B" to show the bottom layer.

By changing the belonging phase in column **Phase**, the number of turns in column **Turns**, the flow-in-side slot number in column **In Slot**, the flow-out-side slot number in column **Out Slot** for each coil, it is possible to arrange the distribution of coils of single and double layer winding of any type required.

The **Winding Editor** dialog box also includes three check boxes:

- **Periodic Multiplier**: indicates the possibility to select the number of unit machines for editing winding arrangement. It has a drop-down menu to show the possible numbers for the periodic multiplier.

  When checked, the drop-down menu box to the right displays the numbers of unit machines for selection. Selecting *1* means whole slots are considered as one unit machine, and all coils is listed in the table of the edit window. Selecting *2* lists half of the total coils in the table, and whole slots are divided into two unit machines, etc. When the check box **Periodic Multiplier:** is unchecked, the drop-down menu to the right is disabled; all the coils are listed in the table.

- **Constant Turns**: Checking the check box (multiple choices) indicates that the number of turns keeps constant and the column **Turns** in the table is grayed (disabled). If the check box **Constant Turns** is unchecked, the column **Turns** in the table is brightened allowing for editing and modifying the number of turns.
- **Constant Pitch**: Checking this box grays the Out Slots column to the values cannot be edited. It means that the coil pitch is constant. For two-layer windings, all flow-in-side slots are defined as top layer, and all flow-out-side slots as bottom layer. The flow-out-side slot number is automatically computed based on the input in the edit box Coil Pitch in Stator2 page in RMxprt window, and Out Slot column is disabled.

When the check box Constant Pitch is unchecked, the column Out Slot is enabled to allow arbitrarily changing slot pitch for each coil.

The **Winding Editor** dialog box includes three command buttons:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click the command button **Default** in the window **Winding Editor**, all the data in the table resumes to the situation of data from automatic arrangement by **RMxprt**.
- Click the command button **Reset** in the window **Winding Editor**, all the data in the table resumes to the situation of data when the window **Winding Editor** was first opened, or resumes to the data that you have saved.
- Click **OK** to accept the current values and close the **Winding Editor** window.

Windings Basic Terminology

Conductor

A conductor refers to a half turn of a coil. A conductor may be formed with one insulated wire, or with several strands of insulated wires.

Strands

A conductor may consist of several wires of same or different sizes stranded together. The number of strands is also called **number of wires per conductor**. The conductor current may not uniformly distribute among all wires, but the current density is uniformly distributed.

Coil

A coil is wound with several **turns**, each turn consisting of two conductors. Coils are generally wound with insulation-wrapped electromagnetic wire continuously on a winding mold. However, coils with single-turn for heavy current are often formed with two separate thick conductors. A thick conductor is hammered onto the winding mold to form a half-coil. The linear part of a conductor imbedded into a slot of iron core is termed **effective side**.

Coil Pitch

The number of slots of the armature iron core spanned by the two effective sides of a coil is termed coil pitch, denoted by y. For instance, if the side of a coil in the 1-st slot spans 8 slots and is connected to the side of the coil in the 9-th slot, the coil pitch of the coil is y = 8.

Full coil pitch:	coil pitch = pole pitch
Short coil pitch:	coil pitch < pole pitch
Long coil pitch:	coil pitch > pole pitch, usually used in variable-pole multiple-speed machines
Pole pitch:	distance between two contiguous poles measured in number of slots. $\text{pole pitch} = \frac{\text{total number of slots}}{\text{number of poles}}$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coil Set

The coils belonging to the same phase under one pole are connected in series as a coil set.

Winding

The coils or coil sets of a phase are connected according to certain rules to form a phase winding. A phase may consist of several **branches** connected in parallel. Every branch must produce exactly the same back emf and must have the same resistance. As a result, the phase winding current is uniformly distributed among all branches.

In summary, a **winding** may be connected with several branches in parallel; each **branch** consists of one or more coil sets connected in series; a **coil set** may have several series coils; a **coil** is wound with a number of turns; a **turn** is formed by two conductors; a **conductor** may be stranded by one or more same- or different-size wires.

Polyphase AC Winding

The common armature winding of polyphase AC machines is cataloged and classified as shown in the following table.

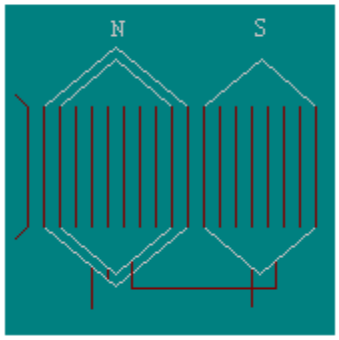
Polyphase AC Winding	
Double layer	Variable-pole multiple speed type
	Fractional slot number type
	Wave-type
	Concentric type
	Lap Type
Single Layer	Crossed Concentric type (whole coiled or half coiled)
	Crossed Chain-type (whole coiled or half coiled)
	Concentric type (whole coiled or half coiled)
	Lap-type (whole coiled or half coiled)
	Chain-type (whole coiled or half coiled)
Compound layer	

Whole-Coiled Windings

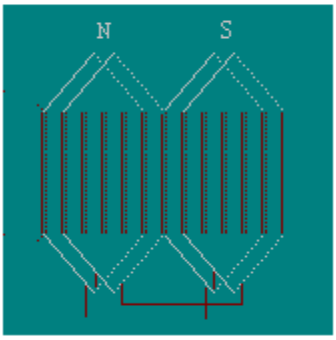
When the coils of an AC winding are connected so that there are as many coil sets per phase as there are poles, the winding is called "whole-coiled."

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Whole Coiled Single Layer



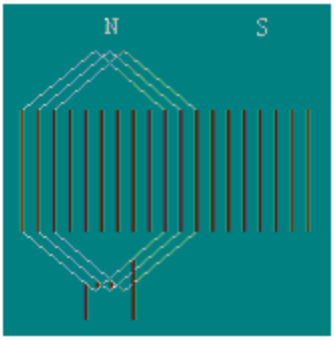
Whole Coiled Double Layer



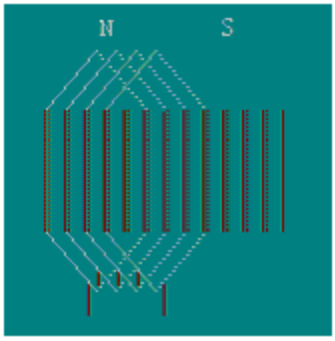
Half-Coiled Windings

When the coils are connected so that there is only one coil set per phase per pair of poles, the winding is called "half-coiled."

Half Coiled Single Layer



Half Coiled Double Layer



Single-Layer Windings

All the conductors in one slot are connected in series with all the conductors in another slot to form a single-layer coil. You set the number of winding layers in the Winding properties window, Winding tab. Comparing to double-layer type, this type is characterized by

- Number of coils halved;
- No need for insulation between layers, therefore higher slot filling factor;
- Coil pitch depends on the connection, and is not adjustable;
- Being widely used in small capacity electric machines.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

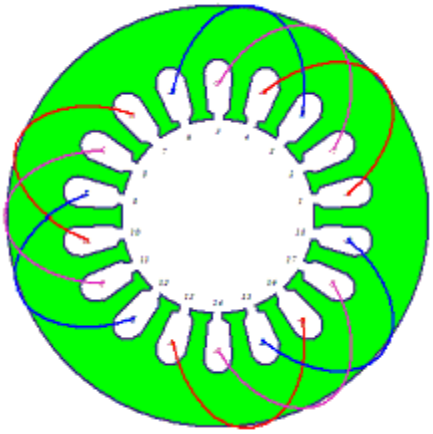
According to different layouts of the end winding, single-layer windings are classified as chain-, lap-, concentric- and crossed-types.

Chain-Type Windings

The name single-layer chain-type is from the linked chain-like developed winding diagram. For a chain-type winding, every coil set has only one coil.

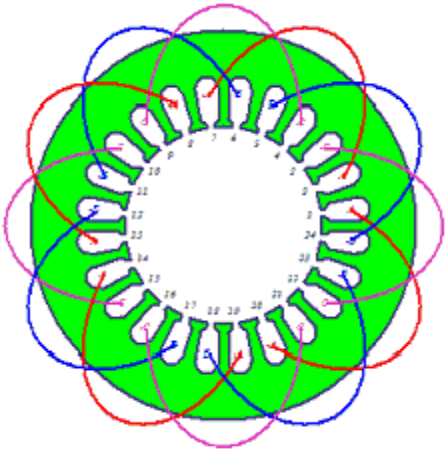
Half-Coiled Chain-Type Winding

An example of three-phase six-pole, 18-slot single-layer half-coiled chain-type winding is shown in the following figure.



Whole-Coiled Chain-Type Winding

An example of three-phase, four-pole 24-slot single-layer whole-coiled chain-type winding is shown in the following figure.



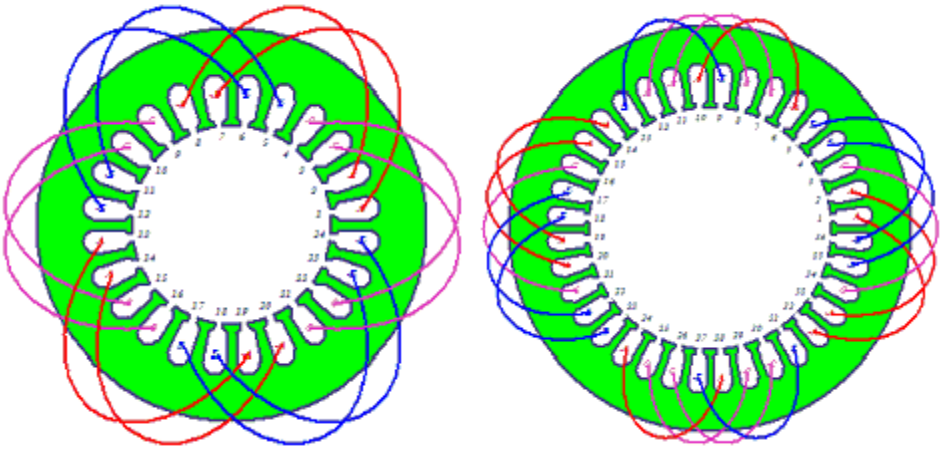
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lap-Type Windings

The name single-layer lap-type is from the lapped layout of end connection. In a lap-type winding, at least one coil set has two or more coils that overlap each other. If some coil sets have only one coil, this winding type is also called "crossed lap-type."

Half-Coiled Lap-Type Winding

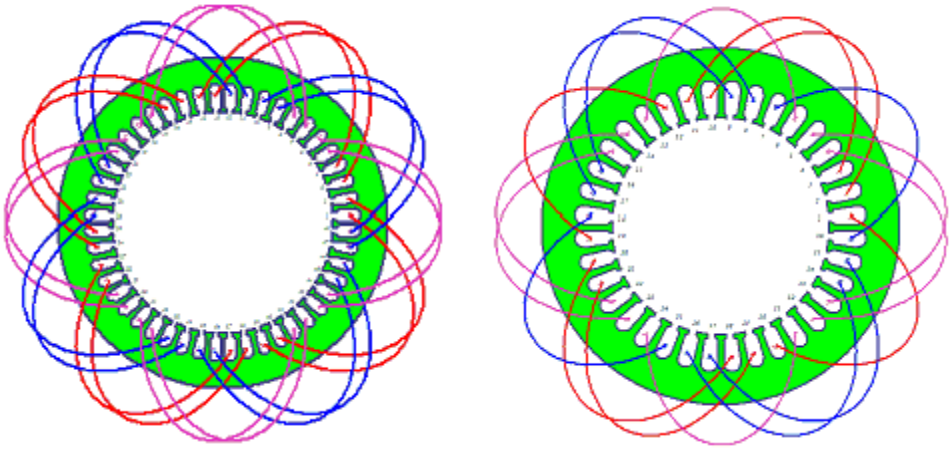
An example of three-phase, four-pole, 24-slot single-layer half-coiled lap-type winding is shown in the figure on the left, and an example of three-phase, eight-pole, 36-slot single-layer half-coiled crossed lap-type windings is shown in the following figure on the right.



Whole-Coiled Lap-Type Winding

An example of three-phase, four-pole 48-slot single-layer whole-coiled lap-type winding is shown on the left, and an example of three-phase, four-pole 36-slot single-layer whole-coiled crossed lap-type winding is shown on the right.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Concentric-Type Windings

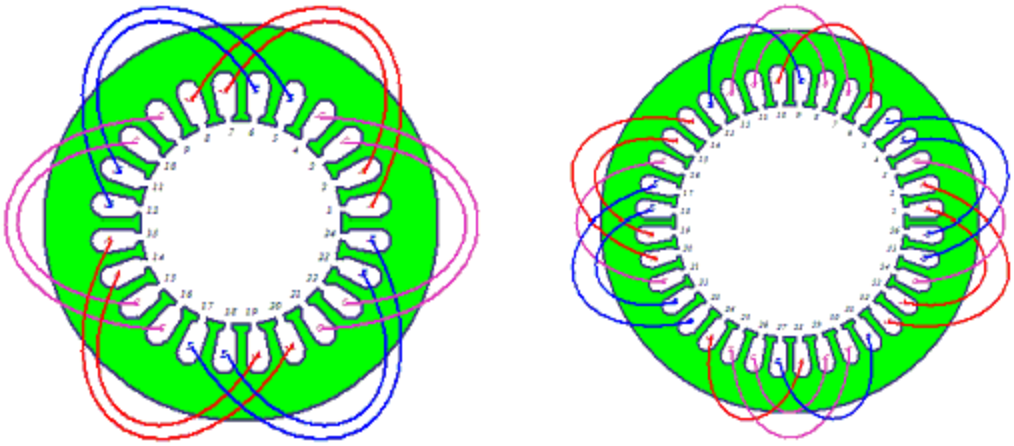
In a concentric-type winding, at least one coil set has two or more coils and the non-coils are overlapping. If some coil sets have only one coil, this winding type is also called "crossed concentric-type".

The single-layer concentric-type is formed of coils with different coil pitch, but with the same central line and of concentric-circle-like, therefore is named concentric-type. Its end connection can be arranged in layers, and therefore is convenient to imbed into slots. Nevertheless, the end magnetic leakage is a bit bigger.

Half-Coiled Concentric-Type Winding

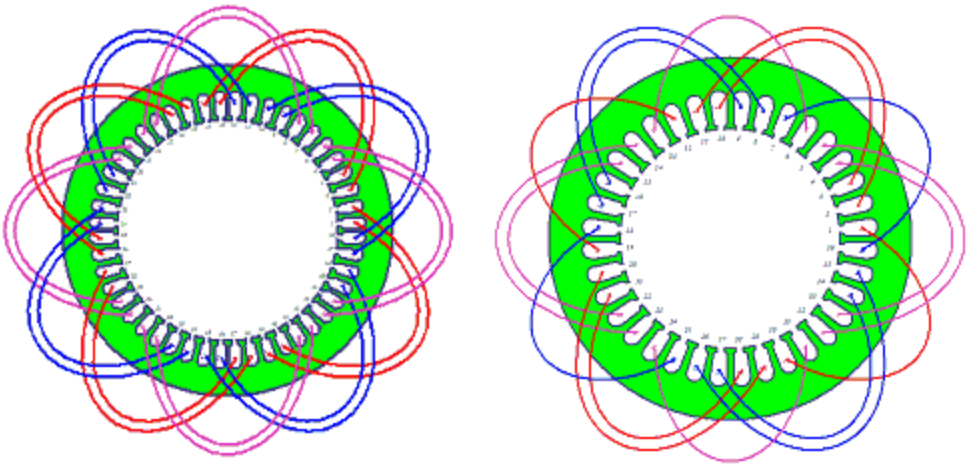
An example of three-phase, four-pole, 24-slot, single-layer, half-coiled, concentric-type winding is shown on the left, and an example of three-phase, eight-pole, 36-slot, single-layer, half-coiled, crossed, concentric-type winding is shown on the right.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Whole-Coiled Concentric-Type Winding

An example of three-phase, four-pole, 48-slot, single-layer, whole-coiled, concentric-type winding is shown on the left, and an example of three-phase, four-pole, 36-slot, single-layer, whole-coiled, crossed, concentric-type winding is shown on the right.



Double-Layer Windings

In this type, the conductors in a slot are arranged in upper and lower layers. One side of each coil is imbedded in the upper layer in one slot and the other side is imbedded in the lower layer in another slot. You set the number of winding layers in the Winding properties window, Winding tab. Comparing to single-layer-type, this type is characterized by

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

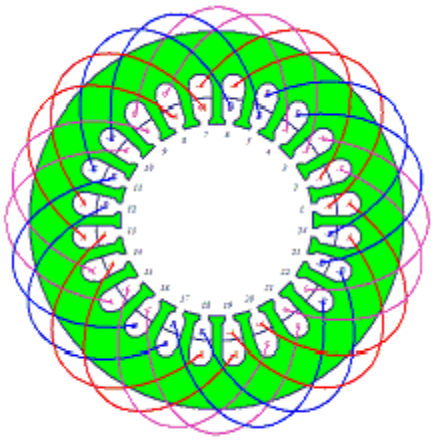
- number of coils doubled;
- need for insulation between layers, therefore lower slot filling factor, and danger in electric breakdown between phases;
- adjustable coil pitch, therefore possible weakening of harmonic emfs with proper short pitch factor to improve electromagnetic properties of electric machines; and
- being widely used in electric machines with capacity over 10 kW.

For the single-speed electric machine, the double-layer winding typically adopts the whole-coiled type. For the double-speed electric machine with doubling number of poles, the double-layer winding is whole-coiled in high speed, half-coiled in low speed.

According to different coil shapes, double-layer windings are classified as lap-, concentric- and wave-types.

Double-Layer Lap-Type Winding

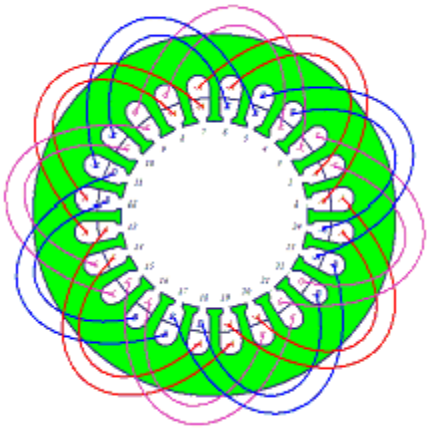
An example of three-phase, four-pole, 24-slot, whole-coiled, double-layer, lap-type windings (short pitch $y = 5$) is shown below.



Double-Layer Concentric-Type Winding

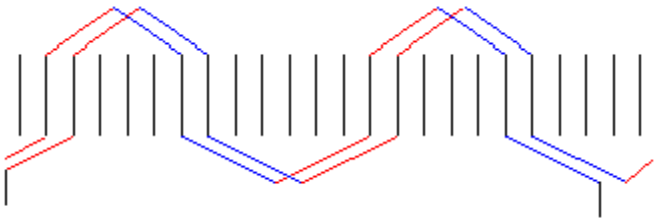
An example of three-phase, four-pole, 24-slot, whole-coiled, double-layer, concentric-type windings (short pitch $y = 5$) is shown below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Double-Layer Wave-Type Winding

The name double-layer wave-type is from the wave-like developed winding diagram as shown below.



Compared to the lap-type:

- The winding of each phase connects the coils under different poles in series in one round, returns to the left to the first coil, then winds the next round, and so on until all the coils belonging to this phase are connected.
- This type is usually used in single-turn preformed hard coil for low-voltage, high-current electric machines.
- This type needs less connection wire between poles.

Fractional-Pitch Winding

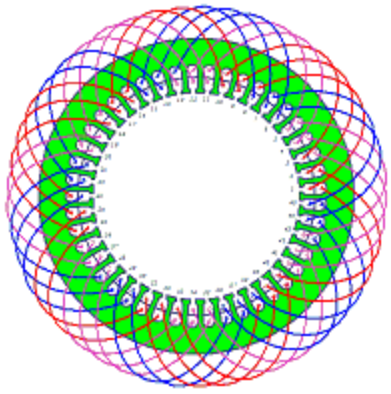
First, introduce a number *q*, called **number of slots per pole per phase**, which is defined as

$$q = \frac{total\ number\ of\ slots}{number\ of\ poles \times number\ of\ phases}$$

A fractional-pitch winding has a fractional number $q = b\frac{c}{d}$.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

An example of three-phase, six-pole, 45-slot, fractional-pitch, double-layer winding ($q = 2\frac{1}{2}$, short pitch *y* = 7, pole pitch $\tau = 7\frac{1}{2}$) is shown here.



Auto-Arrangement of AC Windings

The RMxprt application can arrange these windings automatically if all coils have the same number of turns. This section describes the process to automatically arrange the coil distribution. For winding layout display in **RMxprt**, the lap-type is default if windings are automatically arranged. If a concentric-type layout display is desired, the winding can be defined by winding editor, as described in the next section.

The wave-type winding is effective to a lap-type winding, and is also displayed as a lap-type winding.

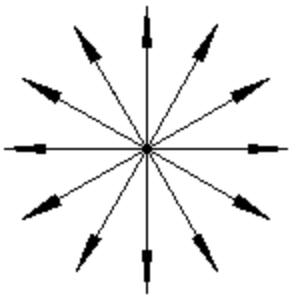
Star Vector Diagram

The conductors (or coils) in slots produce emf (or mmf), which can be expressed with unit vector. When the electric machine has number of pole *p*, and number of slots *Z*, the angular phase difference in electric degrees between two contiguous slots is

$$\alpha = \frac{p \times 180^\circ}{Z}$$

Drawing the vectors of emfs (or mmfs) in all the slots according to their phase angles forms the star vector diagram of the winding. The following figure shows an example of the star vector diagram of four-pole, 24-slot winding.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If there exists the greatest common factor *t* between the number of slots *Z* and the number of pole pairs *pp* (= *p/2*), the star vector diagram repeats *t* times, i.e. the winding has *t* periods. Let

$$Z_0 = \frac{Z}{t}$$

and

$$p_0 = \frac{p}{t}$$

then Z_0 and p_0 construct a complete star vector diagram and form a unit electric machine. For the whole-pitch winding electric machine (*q*, as shown later, is an integer), *t* = *p/2*. For the fractional-pitch winding electric machine,

$$q = \frac{Z}{mp} = b + \frac{c}{d}$$

where *m* is the number of phases. If *t* > 1, the angular phase difference between two contiguous vectors is

$$\alpha = \frac{360^\circ}{Z_0}$$

and the difference between the ordinal numbers of the slots of two contiguous vectors is

$$y_0 = \frac{m(bd+c)G-1}{d}$$

where *G* is a minimum integer to make y_0 equal to an integer (y_0 should take into account the possible reverse connection of coils under the contiguous pole).

Phase Spread

In the star vector diagram of a unit electric machine, the range occupied by the vectors of each phase under one pole is termed phase spread, expressed in electric degrees or number of slots. For a single-layer winding, the phase spread is 180°/*m* (*m* – the number of phases). The phase spread of a double-layer half-coiled winding is 360°/*m*, and the phase spread of a double-layer full-coiled winding is 180°/*m*.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The phase spread of a 2-phase winding is always 90° (= 180°/*m*). Therefore, a 2-phase winding cannot take the double-layer half-coiled winding type. The windings for single-phase induction motor are also considered as 2-phase windings.

When the number of phases is an even number of greater than or equal to 4, the phase spread is always 360°/*m*. Therefore, a winding with even number of phases (4, 6, …) can take only the double-layer half-coiled winding type.

When the number of phases is an odd number of greater than or equal to 3, the phase spread can be either 360°/*m* or 180°/*m*. Therefore, a winding with odd number of phases (3, 5, …) can take any winding types.

Coil Arrangement

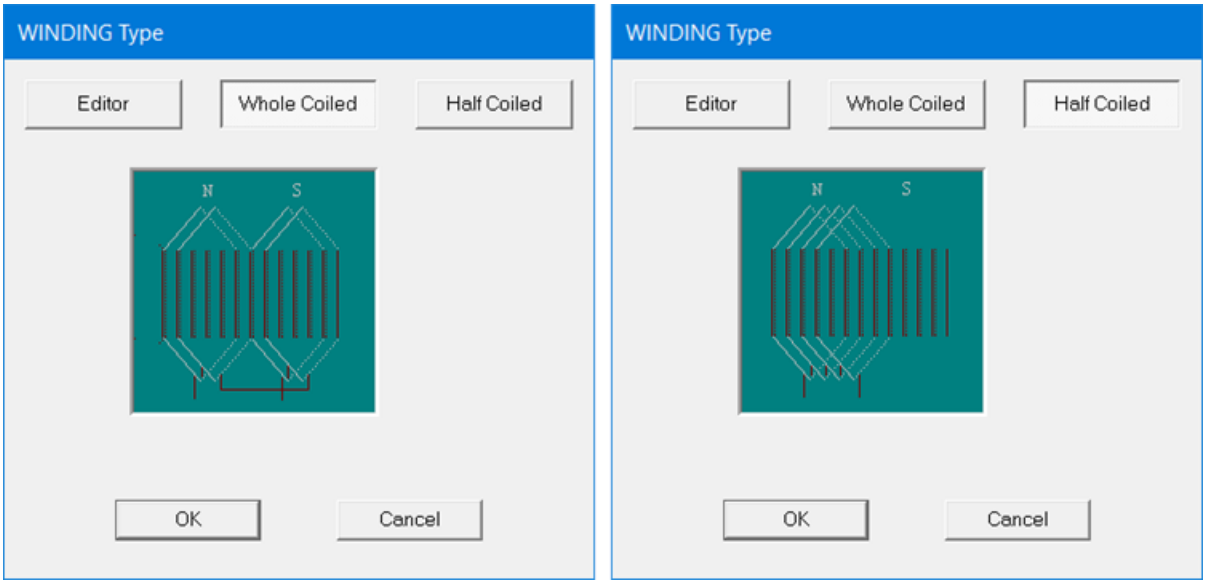
Coil arrangement is completed by the following processes. First, draw the star vector diagram based on number of slots and number of poles. Then divide the whole region (360 electric degrees) to several phase spreads, which is derived from the number of phases and the winding type. Finally, assign all phase spreads to each phase in such a way that the axis of each succeeded phase lags by 360/*m* electric degrees (90 electric degrees for 2 phases).

Double-Layer Windings

Take a three-phase winding as an example. The width of phase spread of half-coiled winding is $360^o / 3 = 120^o$, and the sequence of the phase spread is A, B, C. For whole-coiled winding, the width of phase spread is $180^o / 3 = 60^o$, and the sequence of phase spread is A, –C, B, –A, C, –B, where the phase spread with negative sign is termed negative phase spread.

The winding types can be set in the Winding Type dialog box for a machine that includes these options (in this case, a brushless permanent magnetic DC motor), for double-layer whole-coiled windings as shown in on the left and double-layer half-coiled windings as shown on the right.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

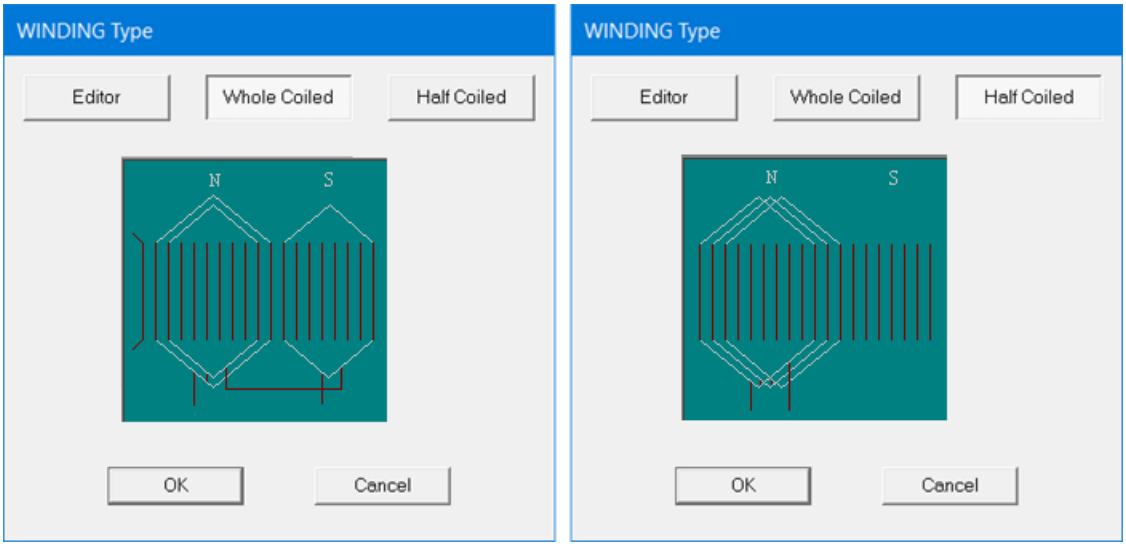


The star vector diagram of a three-phase, whole-coiled (60^o-phase-spread) winding is shown below on the left, and that of a half-coiled (120^o phase spread) winding is shown below on the right.

Single-Layer Windings

The winding layers can be set in the properties window for the winding, for single-layer, whole-coiled windings as shown on the left and single-layer, half-coiled windings as shown on the right.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The phase spread of a three-phase, single-layer, whole-coiled or half-coiled winding is 60°, and the star vector diagram is the same as the double-layer, whole-coiled winding.

Fractional-Pitch Windings

The number of slots per pole per phase of fractional-pitch winding is a mixed number.

$$q = b\frac{c}{d}$$

In the unit electric machine, the numbers of slots occupied by phase spread are not all the same, but repeat with the radix *d*. In each *d* poles, there are *c* poles with the slot number of phase spread equal to *b* + 1 (big phase spread), *d* – *c* poles with the slot number of phase spread equal to *b* (small phase spread).

Take as an example a three-phase 10-pole 36-slot fractional-pitch winding with phase spread of 60°. The number of slots per pole per phase of fractional-pitch winding is

$$q = \frac{36}{3\times 10} = 1\frac{1}{5}$$

the greatest common factor between the number of slots 36 and the number of pole pairs 5 is *t* = 1, the angular phase difference between two contiguous vectors in the star vector diagram is

$$\alpha = \frac{360^\circ}{36} = 10^\circ$$

the difference between the ordinal numbers of the slots of two contiguous vectors is (*G* = 2)

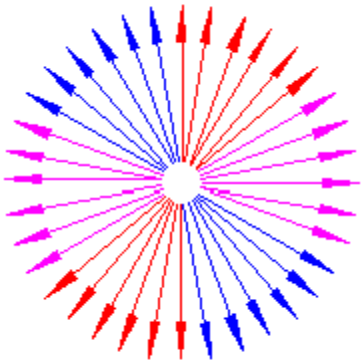
$$y_0 = \frac{3(1\times 5+1)\times 2-1}{5} = 7$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the repetition radix *d* = 5. In each 5-pole region, each phase has big phase spread of 1 + 1 = 2 slots under 1 pole, and small phase spread of 1 slot under 4 poles. The repeating format is 2 1 1 1 1 for phase A. The repetition of phase spread distribution for all phases is shown in the following table.

Slot number	1~2	3	4	5	6	7~8	9	10	11	12	13~14	15	16	17	18
Phase spread	A	–C	B	–A	C	–B	A	–C	B	–A	C	–B	A	–C	B
Slot number	19~20	21	22	23	24	25	27	28	29	30	31	33	34	35	36
Phase spread	–A	C	–B	A	–C	B	–A	C	–B	A	–C	B	–A	C	–B

The star vector diagram of winding is shown below.



Asymmetric Windings

Whole-pitch windings (*q* is integer) are always symmetric. Fractional-pitch windings with

$$q = b\frac{c}{d}$$

becomes asymmetric if the denominator *d* is a multiple of the number of phases *m*. In general, it is avoid using asymmetric windings as possible. Nevertheless, it is sometime possible to design poly-phase windings with little asymmetry in order to use existing punching tools.

If *d* is a multiple of the number of phases *m*, but the total number of slots *Z* can be divided by *m*, it is possible to construct poly-phase winding with little asymmetry. The RMxprt application can

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

perform automatic arrangement for this sort of windings and obtain the phase-spread in electric degrees for each phase.

Take as an example a three-phase, six-pole, 66-slot, fractional-pitch winding electric machine. Because

$$q = \frac{66}{3\times 6} = 3\frac{2}{3}$$

$d = m = 3$, the winding is asymmetric. The output in the window **Design Output** is shown below.

The information for WINDING ARRANGEMENT is displayed as follows:

The distribution of coil slots to phases:

The 3-phase, 2-layer winding can be arranged in 66 slots as below:

AAAAZZZZBBBXXXXCCCCYYYAAAZZZZBBBBXXXCCCCYYYYAAAAZZZBBBBXXXXCCCYYYY

X, Y and Z stands for –A, -B and –C, respectively. For asymmetric windings, additional information is output, as shown below.

The winding factors of each phase are

Phase A	0.954119
Phase B	0.954119
Phase C	0.949042

The angles between two-phase winding axes are

Phase A & B	119.082
Phase B & C	120.459
Phase C & A	120.459

If a sinusoidal rotating field links the winding, the fundamental induced-voltage components will be

Positive-sequence	100%

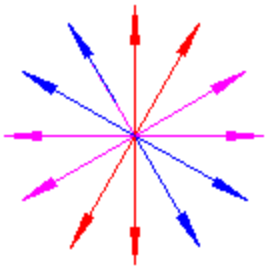
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

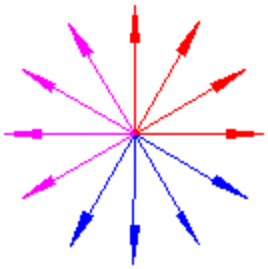
component	
Negative-sequence component	0.286577%
Zero-sequence component	0.639823%

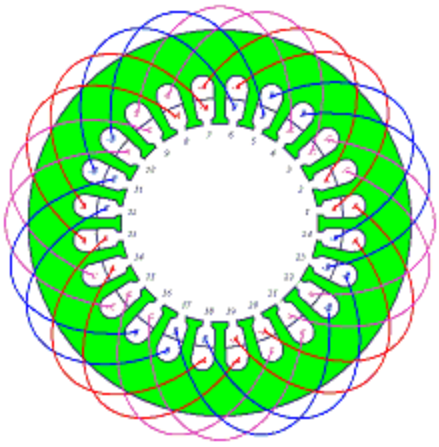
Coil Connections

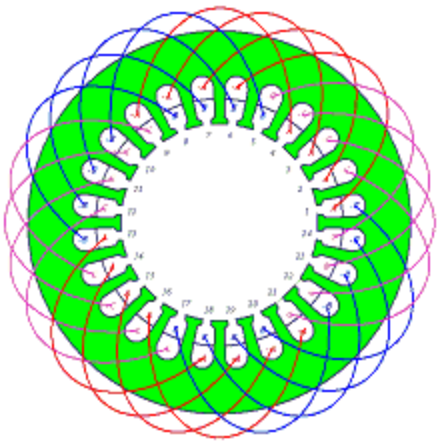
Connection of Double-Layer Lap Windings

Every vector represents the top-layer effective side of a coil. The bottom effective side of the coil is determined based on the coil pitch, and is not displayed in the diagrams. Therefore, every vector in the diagrams can also stand for a coil. Connect all coils in phase spread of A in positive direction, and all coils in phase spread of –A in negative direction to form the phase A winding. In this way, phase B and C windings can also be connected. The winding connection layouts for the vector diagrams are shown below.





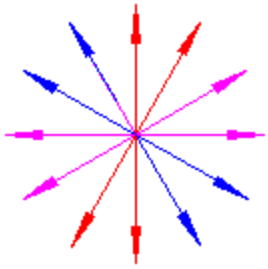


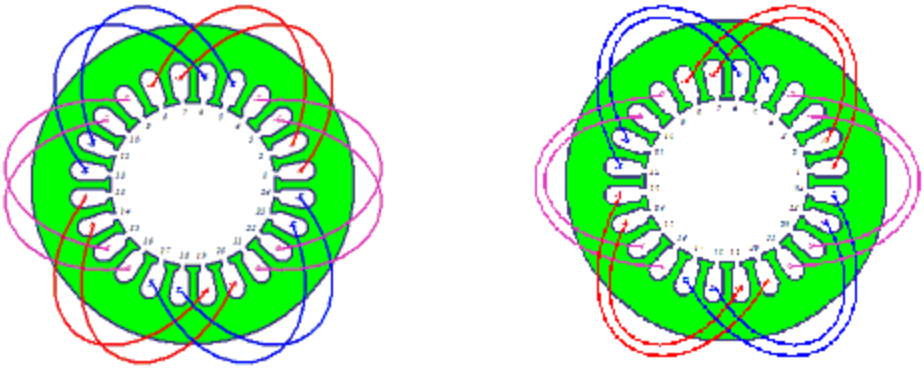


Connection of Single-Layer, Half-Coiled Windings

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Every vector in A, B and C phase spread represents "go" effective side of a coil, the "return" effective side of the coil is located in –A, –B and –C phase spread. For the lap type connection, all coils are with full coil pitch. The connection layouts of the lap type and the concentric type, with respect to the same vector drawing are shown below.



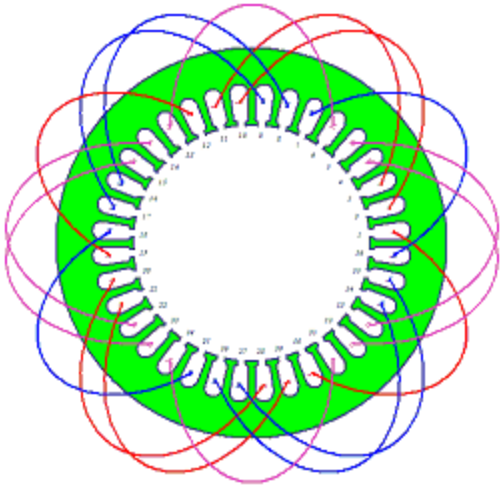


Connection of Single-Layer, Whole-Coiled Windings

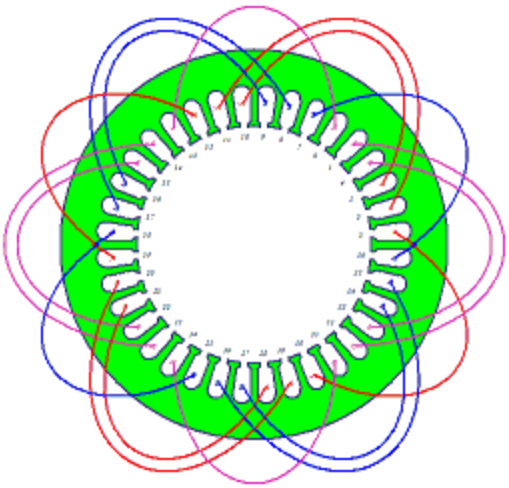
In the previous example, for the concentric type (lower right in the diagram), if coil 1 is not connected from slot 1 to slot 8 (long coil pitch: coil pitch = 7 > pole pitch = 6), but connected from slot 1 to slot 20, and slot 8 to slot 13, all coils of phase A winding have coil pitch of 5. In this way, the winding becomes single-layer whole-coiled type with the same star vector diagram and phase spread, and has much shorter average coil pitch. Therefore, single-layer whole-coiled windings consume less electromagnetic wire than single-layer half-coiled windings. **RMxprt** can optimize connections to minimize the average coil pitch to form a single-layer whole-coiled winding.

An example of three-phase, four pole, 36-slot ,single-layer, whole-coiled, crossed lap-type winding (q = 3, 60^{o} phase-spread) is shown below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

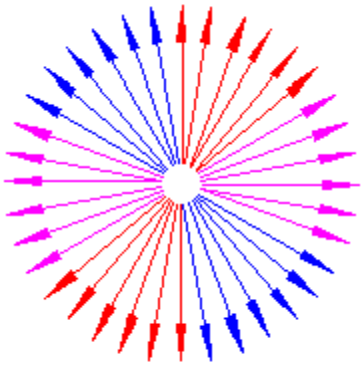


An example of three-phase, four-pole, 36-slot, single-layer, whole-coiled, crossed concentric-type winding (q = 3, 60^o phase-spread) is shown below.

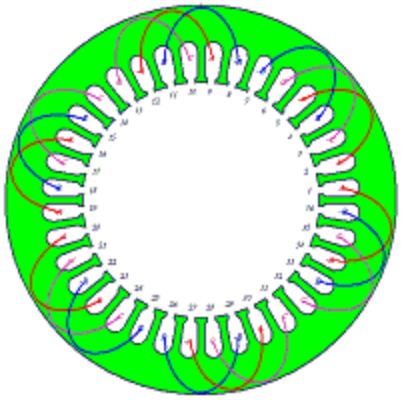


A star vector diagram with fractional coil pitch can also be connected with single-layer whole-coiled type. When the number of slots per pole per phase q <2, as shown in the following vector diagram, the number of coil sets per phase may not equal to the number of poles (6 coils vs 10 poles), but the algorithm to connect coils is the same (minimize the average coil pitch), and therefore, it is still referred as whole coiled windings.

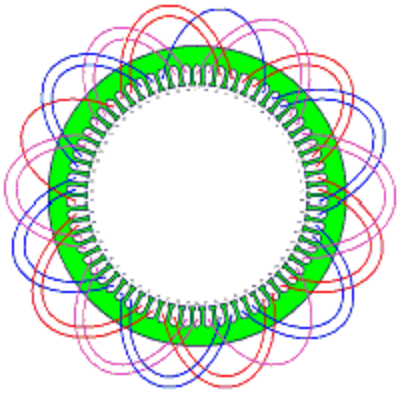
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The winding connection layout for the previous vector diagram is shown below.



Another example is an asymmetric three-phase winding. The connection layout is shown below.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

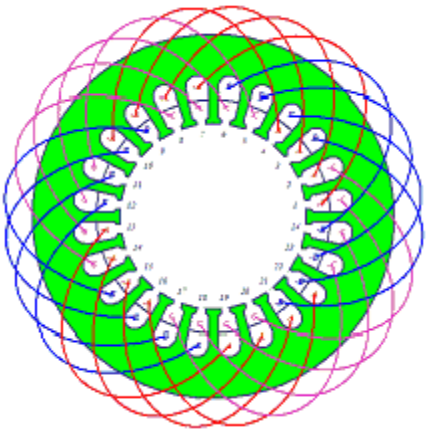
Connection of Double-Pole, Dual-Speed Windings

Commonly used for coils of double-layer windings are lap- and concentric-types. Single-speed double-layer winding electric machine is usually connected as whole-coiled winding. Double-pole, dual-speed electric machine is connected as double-layer, whole-coiled winding at high speed and as double-layer, half-coiled winding at low speed.

Take as an example a three-phase, dual-speed, 2/4-pole, 24-slot, double-layer winding. The arrangement of coils is shown in the Table and in the following figure.

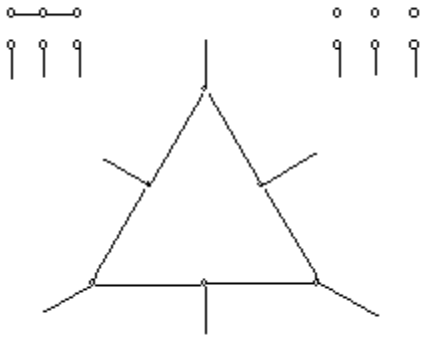
Slot number	1	2	3	4	5	6	7	8	9	10	11	12
Two poles	A	A	A	A	–C	–C	–C	–C	B	B	B	B
Four poles	A	A	A	A	C	C	C	C	B	B	B	B

Slot number	13	14	15	16	17	18	19	20	21	22	23	24
Two poles	–A	–A	–A	–A	C	C	C	C	–B	–B	–B	–B
Four poles	A	A	A	A	C	C	C	C	B	B	B	B



The connection for variation of number of poles is shown in the next figure. On the top-left is the two-pole, 2Y-connection at high speed, on the top-right is the four-pole connection at low speed.

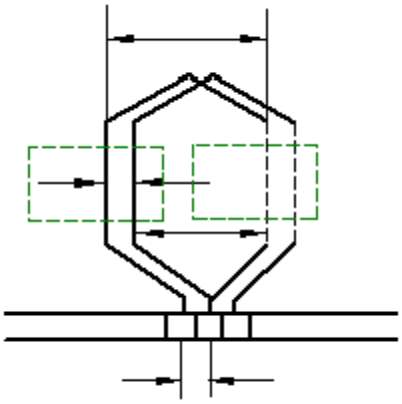
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



DC Windings

Lap Winding

The winding layout of a lap-type winding for DC machines is as shown below.



A lap-type winding has the following relationships:

Coil pitch	$y_1 = \frac{Z}{p} \mp \varepsilon = integer$
Commutator pitch	$y_k = \pm m$
Number of branches in parallel	$a = mp$

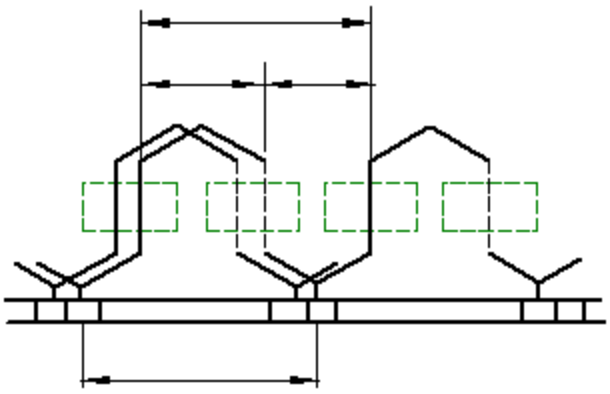
where

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Z	number of slots
p	number of poles
m	number of multiplex

Wave Winding

The winding layout of a wave-type winding for DC machines is as shown below.



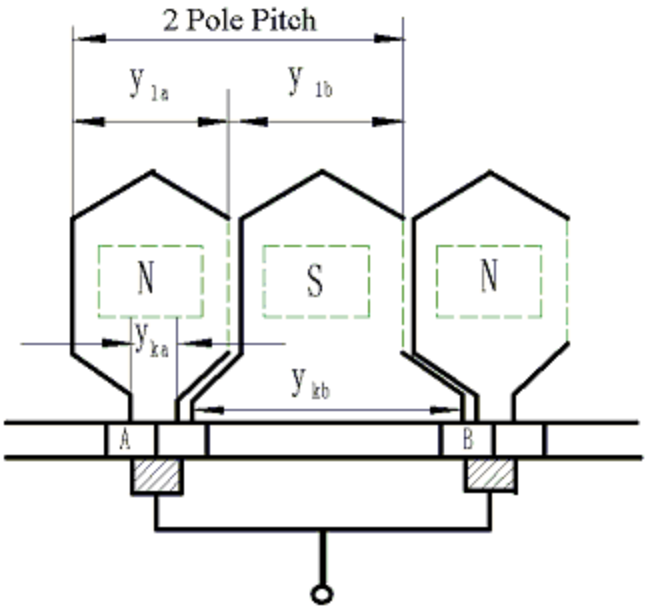
A wave-type winding has the following relationships:

Coil pitch	$y_1 = \frac{Z}{p} \mp \varepsilon = integer$
Commutator pitch	$y_k = \frac{K \mp m}{p/2} = integer$
Number of branches in parallel	$a = m$

Frog-Leg Winding

A frog-leg winding consists of a lap winding and a wave winding. Assume the lap winding has *m* multiplex number. In order to connect the wave winding in parallel with the lap winding, the wave winding should have the same branch bake emf as the lap winding. Therefore, the number of branches in parallel of the wave winding has to be the same as that of the lap winding, or the multiplex number of the wave winding must be *mp/2*. The winding layout of a frog-leg-type winding with *m=1* for dc machines is as shown below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A frog-leg-type winding has the following relationships:

Coil pitch	$y_{1a} + y_{1b} = \frac{2K}{p} = integer$
Commutator pitch	$y_{Ka} + y_{Kb} = \frac{2K}{p} = integer$ $y_{Ka} = \pm m$ $y_{Kb} = \frac{2K}{p} \mp m$
Number of branch in parallel	$a = pm$

where *m* is the number of multiplex (of the lap winding).

Virtual Slots

Windings of the dc machine are usually double-layer type. In many cases, often a number of coil sides are imbedded into one slot for simplification of structure. Therefore, the number of slots *Z* is less than the number of coils *S*. There exists the relationship

$$Z = \frac{S}{\mu}$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where μ is the number of coil sides in each layer in one slot and is termed virtual slot factor. Therefore, the number of conductors per slots is equal to multiple of 2μ.

Equipotential Connectors

The points ideally possessing the same electric potential in armature winding are often wholly or partly connected by short copper wire, which is termed equipotential connector.

Equipotential Connector Class A of Simplex-Lap Winding

Asymmetry in magnetic circuit, such as eccentric air-gap, causes circulating current in lap winding, increases losses and affects commutation in order. The equipotential connector Class A on the commutator of simplex-lap winding can solve this problem. The connection pitch of equipotential connector Class A of simplex-lap winding y_p is equal to the number of commutator segments per pole pair:

$$y_p = \frac{K}{p} = \frac{K}{a} = integer$$

Winding with y_p = integer is termed symmetric winding. Only symmetric winding can have equipotential connector Class A.

There is no need for an equipotential connector for Simplex-Wave Winding because it does not possess electrically equipotential points, therefore, can not have equipotential connector. On the other hand, simplex-wave winding does not need equipotential connector.

Equipotential Connector Class B of Multiplex-Wave Winding

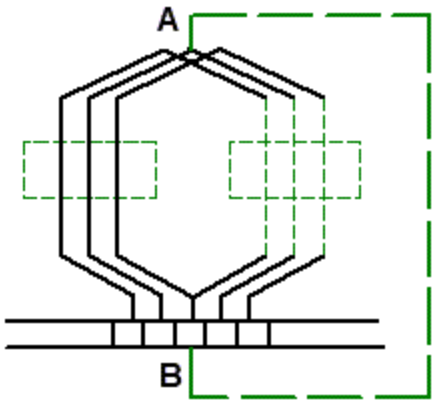
There are electric equipotential points among different sets of simplex-wave windings of multiplex-wave winding. They can be connected to eliminate the nonuniform distribution of electric potential on commutator segments due to inequality of brush resistances. This is termed equipotential connector Class B. The connection pitch of equipotential connector Class B of multiplex-wave winding y_p is equal to the number of commutator segments per branch pair in parallel:

$$y_p = \frac{K}{a} = integer$$

Equipotential Connector of Multiplex-Lap Winding

For multiplex-lap winding, equipotential connector Class A is needed for each set of simplex-lap windings, equipotential connector Class B is needed among different sets of simplex-lap windings. There exist no electrically equipotential points among different sets of simplex-lap windings on the commutator side of armature; however, there exist electrically equipotential points among different sets of simplex-lap windings on different sides of armature as points A and B in the following figure.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Those points can be connected by conductors passing through inside armature.

For frog-leg winding, each pair of electrically equipotential points on commutator is connected by a lap coil and a wave coil in series. The connection acts as equipotential connector Class A for lap winding and as equipotential connector Class B for wave winding. Therefore, there is no need for extra equipotential connectors.

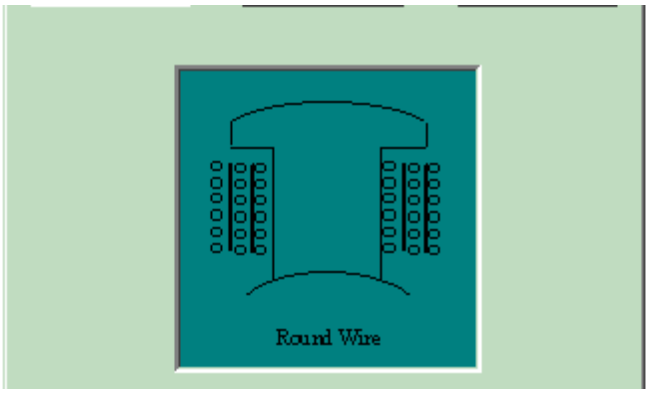
Pole Windings

The following two types of electric machines possess similar pole winding structure:

- DC machine (motor and generator)
- Three-phase synchronous machine (motor and generator)

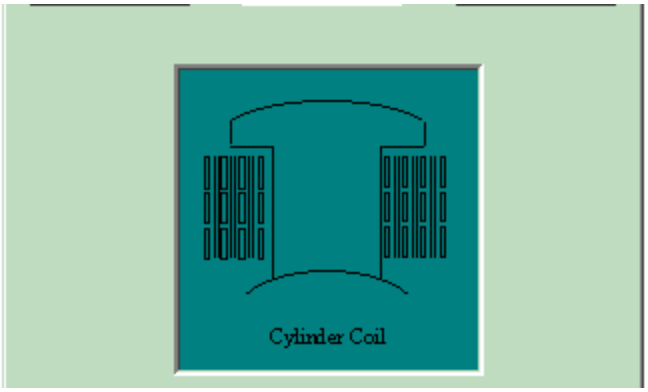
The RMxprt application adopts the same arrangement procedure for pole windings of the two types of electric machines. There are three types of structure as shown in the following figure:

Round Wire Pole winding with round wire:

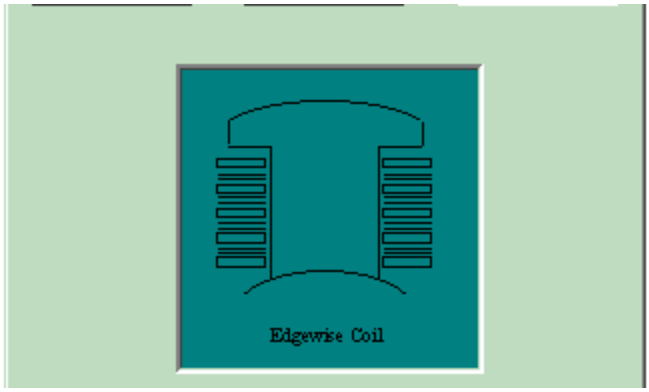


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cylinder Coil	Pole winding with rectangular wire wound in standing way:



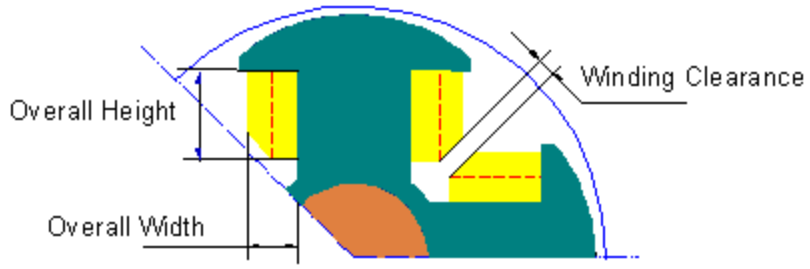
Edgewise Coil	Pole winding with rectangular wire wound in flatting way:



Limited Space for Wire Arrangement

Before completing a winding arrangement, the RMxprt application needs to determine the limited space sizes for the winding. The limited sizes include limited **Overall Height**, limited **Overall Width**, and **Winding Clearance** (the clearance between two adjacent pole windings), as shown below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

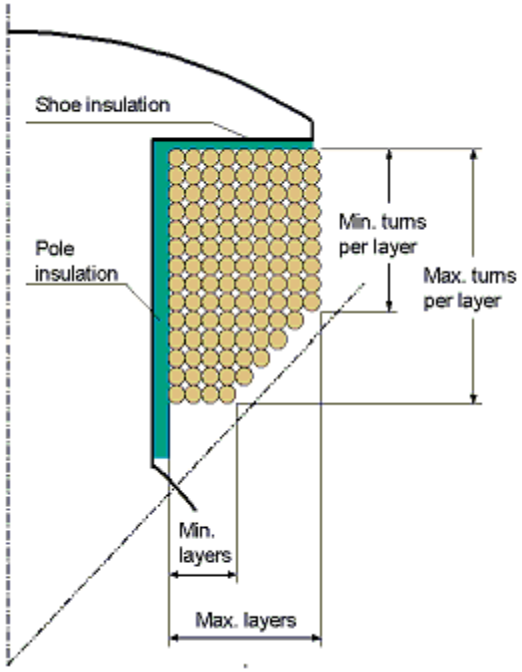


Overall Height, **Overall Width**, and **Winding Clearance** can be input from RMxprt panel. If either **Overall Height** or **Overall Width** is set to *0*, **RMxprt** perform automatic space optimization to obtain the maximum space for pole winding arrangement in the condition to guarantee the clearance between pole windings.

If the space from input or determined by space optimization is not sufficient to arrange for the input number of turns per pole, a message of "The rotor/shunt/series/commutating winding control dimension is not big enough." is displayed in **Design Output** window.

Round Wire Winding

The arrangement of pole winding with round wire is shown below.

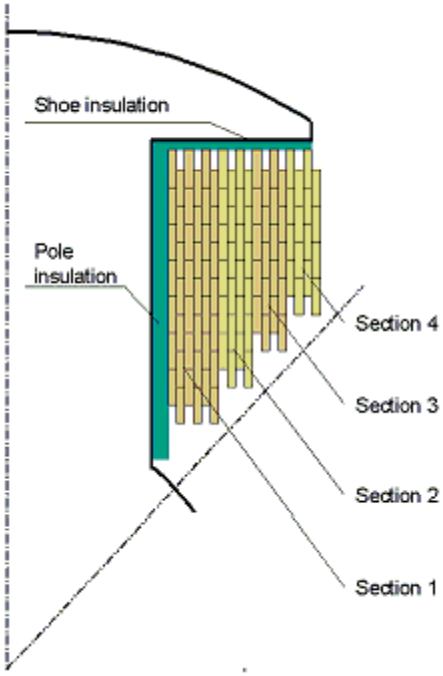


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

With auto-arrangement of pole windings, **RMxprt** calculates maximum number of layers, minimum number of layers, maximum number of turns per layer, minimum number of turns per layer, and the maximum number of turns per pole that is available from the limited space, and so forth.

Cylinder Coil

The arrangement of magnetic-pole winding with rectangular wire wound in standing way by the RMxprt application is shown below.

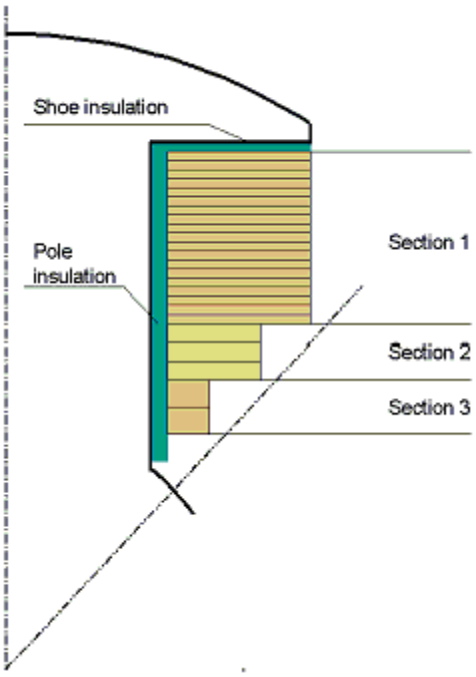


The cylinder coil is wound with half-turn over lapped layer by layer. Layers with the same number of turns constitute a section. The output window **Design Output** displays the number of layers and the number of turns per layer of each section, and the maximum number of turns per pole that is available from the limited space.

Edgewise Coil

The arrangement of a magnetic-pole winding with a rectangular wire wound in a flatting way by the RMxprt application is shown below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

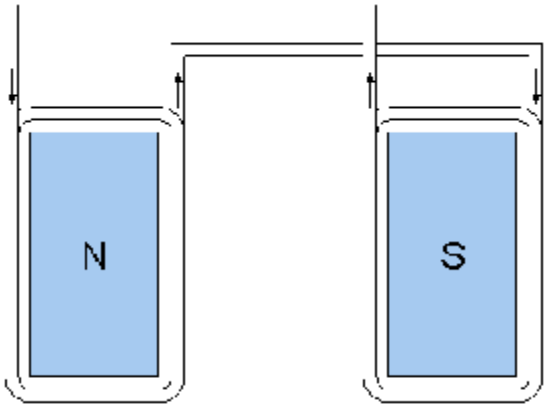


To guarantee the clearance between two adjacent pole windings, the wire width of the lower parts is decreased, while the wire thickness is increased to keep the sectional area of wire invariant as possible. The turns with the same wire gauge constitute a section (maximum three sections are allowed). The output window **Design Output** displays the number of turns of each section and the sizes of wire gauge, and the maximum number of turns per pole that is available from the limited space.

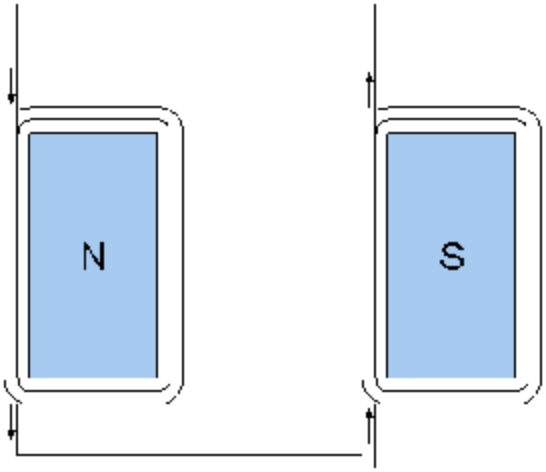
Pole Winding with Half Turns

For some large machines, a half turn may be included due to too few turns per pole. When the number of turns per pole is an integer number, the number of conductors per pole is an even number with equal conductor number at both pole sides. Therefore, the two terminal leads of one pole coil are at the same axial side (or at different pole sides), as shown below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When the number of turns per pole includes a half turn, the number of conductors per pole is an odd number. In this case, one pole side has one more conductor than the other pole side, and the two terminal leads of one pole coil are at different axial sides (or at the same pole side), as shown below.



Exporting Winding Data

Winding data may be exported to a table:

1. Select the **Machine** > **Winding** > **Export Layout** command from the menu. You may also right-click in the **Winding Editor** window and select **Export Layout** from the shortcut menu.
2. Browse to the location to save the file and enter a filename.
3. Click **Save** to export the winding data to a file and dismiss the dialog box.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The winding must be editable for the Export Layout command to be available. If you are using a standard winding, you can switch to the Winding Editor:

1. Click on the **winding** in the Project Manager tree.
2. In the Properties Window, click on the button next to **Winding Type**.

   The **Winding Type** dialog box is displayed.
3. Select **Editor** as the winding type and click **OK**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RMxprt Machine Types

Using Ansys RMxprt, you can simulate and analyze the following standard machine types:

- Adjust-Speed Synchronous Machine
- Brushless Permanent-Magnet DC Motor
- Claw-Pole Alternator
- General DC Machine
- Generic Rotating Machine
- Line-Start Permanent-Magnet Synchronous Motor
- Permanent-Magnet DC Motor
- Single-Phase Induction Motor
- Switched Reluctance Motor
- Three-Phase Induction Motor
- Three-Phase Non-Salient Synchronous Machine
- Three-Phase Synchronous Machine
- Universal Motor

Three-Phase Induction Motors

After you have selected **Three-Phase Induction Motors** as your model type, you must define the following:

- General data, such as the voltage, speed, and materials.
- Stator data, such as the slot types, diameter, and wire dimensions.
- Rotor data, such as the slot dimensions, skew, and ventage holes.
- Solution data, such as rated output voltage and frequency.

By option, you can

- Add a machine housing.
- Add vents to and remove an existing vent from the stator and rotor.

Analysis Approach for Three-Phase Induction Motors

For a three-phase induction motor, the stator winding (with a sinusoidal spatial distribution and *p* pairs of poles) is connected to a three-phase symmetric voltage power supply. The resulting currents in the stator produce a rotating magnetic field. The rotor winding is often a squirrel-cage type with the number of poles dictated by the number of poles in the stator. Currents are induced in the rotor bars and produce, in turn, a second rotating magnetic field. The two rotating fields produce a resultant rotating magnetic field in the air gap of the machine. The interaction of this field in the air gap with the rotor bar currents produces an electromagnetic torque, which acts on

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the rotor in the direction of the rotation of the field in the air gap. A torque of equal value acts upon the stator in the opposite direction.

The stator winding, which is connected to a phase of the supply system, has *p* coils, each with a symmetric spatial distribution and an opening of $\pi D/2p$, where *D* is the diameter of the winding. In this case, the magnetic field in the air gap has *p* periods, and the winding has *p* pairs of poles.

The performance of three-phase induction motors (IndM3) is analyzed based on the equivalent circuit of one phase in the frequency domain as shown in Figure 1.

In the figure, R1 is the stator resistance, X1 is stator leakage reactance, which consists of stator slot leakage reactance, end-winding leakage reactance, and differential leakage reactance. X2 and R2 are rotor leakage reactance and rotor resistance, respectively. X2 includes rotor slot leakage reactance, end-ring leakage reactance, differential leakage reactance, and skewing leakage reactance. Due to the saturation of the leakage field, X1 and X2 are nonlinear. The parameters in the equivalent circuit are dependent on the stator and rotor currents. Due to the skin effects, X2 and R2 are the equivalent values from a distributed-parameter circuit, as shown in Figure 2.

They vary with the rotor slip *s*. All rotor parameters have been referred to the stator side.

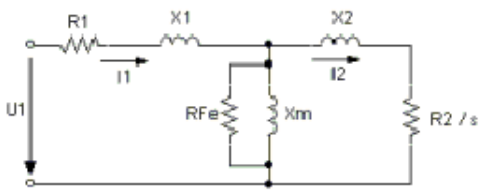


Figure 1

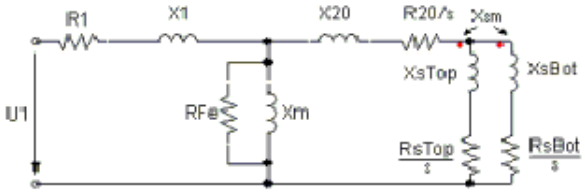


Figure 2

In the exciting branch, Xm is the magnetizing reactance, and RFe is the resistance corresponding to iron-core losses. Xm is a linearized nonlinear parameter that varies with the saturation of the main field.

After a phase voltage U1 is applied to the phase terminals, stator phase current I1 and rotor current I2, which has been referred to the stator, can be easily computed by the circuit analysis. The electromagnetic power Pm, or air-gap power, is computed by the following:

$$Pm = 3 * I2^2 * R2/s$$

The electromagnetic torque Tm is

$$Tm = Pm/\omega$$

where ω is the synchronous speed in rad/s.

The output mechanical shaft torque T2 is

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$T2 = Tm - Tfw$$

where Tfw is the frictional and wind torque.

The output power is

$$P2 = T2 * \omega 2$$

where $\omega 2 = \omega^* (1 - s)$ and is rotor speed in rad/s.

The input power is

$$P1 = P2 + Pfw + Pcu2 + PFe + Pcu1 + Ps$$

where Pfw, Pcu2, PFe, Pcu1, and Ps are frictional and wind loss, rotor copper loss, iron-core loss, stator copper loss, and stray loss, respectively.

The power factor is derived from

$$PF = P1/(m * U1 * I1)$$

The efficiency is computed by

$$\eta = \frac{P_2}{P_1} \times 100\%$$

Defining a Three-Phase Induction Motor

The general procedure for defining a three-phase induction motor is as follows:

1. Insert a three-phase induction motor into an existing or new project.
2. Double-click the **Machine** entry in the Project Manager tree to define the general data, such as the number of poles and machine losses.
3. Double-click the **Machine** > **Stator** entry in the Project Manager tree to define the stator geometry.
4. Double-click the **Machine** > **Stator** > **Slot** entry in the Project Manager tree to define the stator slot dimensions.
5. Double-click the **Machine** > **Stator** > **Winding** entry in the Project Manager tree to define the stator windings and conductors.
6. Double-click the **Machine** > **Rotor** entry in the Project Manager tree to define the rotor geometry.
7. Double-click the **Machine** > **Rotor** > **Slot** entry in the Project Manager tree to define the rotor slot dimensions.
8. Double-click the **Machine** > **Rotor** > **Winding** entry in the Project Manager tree to define the rotor conductor, ventage hole dimensions, and skew.
9. Double-click the **Machine** > **Shaft** entry in the Project Manager tree to define the magnetism of the shaft.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10. Optionally, right-click the **Machine** entry in the Project Manager tree and click **Insert Housing** to add a machine housing.
11. Right-click **Analysis** in the project tree, and click **Add Solution Setup** to define the solution data.
12. Choose **File > Save** to save the project.
13. Choose **RMxprt > Analyze** to analyze the design.

> **Note:** When you place the cursor over an entry field in the data windows, a brief description of that field appears in the status bar at the bottom of the **RMxprt** window.

Once the design is analyzed, the model can be viewed in the Maxwell 2D Modeler, or it can be used to create a new Maxwell 2D project, and a new Maxwell 3D design.

For a specific example of a three-phase induction motor problem, see the *Three-Phase Induction Motor Problem* application note, on the technical support page of the Ansys website.

Defining the General Data for a Three-Phase Induction Motor

Use the **General Data Properties** window to define the basic parameters of the induction motor, such as the number of poles, and frictional loss.

To define the general data:

1. To open the **General Data Properties** window, double-click the **Machine** entry in the Project Manager tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the number of poles for the machine in the **Number of Poles** field. This value is the total number of poles in the stator (or the number of pole pairs multiplied by two).
3. Enter the stray loss factor in the **Stray Loss Factor** field. The stray load loss consists of the losses arising from non-uniform current distribution in the copper and additional core losses produced in the iron by distortion of the magnetic flux by the load current. The IEEE Standard provides different assumed stray load loss values for AC motors rated less than 2500 hp, as follows:
    - 1-125 HP = 1.8% of rated output power
    - 126-500 HP = 1.5%
    - 501-2499 HP = 1.2%
4. Enter the energy loss due to friction at the given speed in the **Frictional Loss** field.
5. Enter the wind loss due to air resistance measured at the reference speed in the **Wind Loss** field.
6. Enter the given speed in the **Reference Speed** field.
7. Click **OK** to close the **Properties** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Data for Three-Phase Induction Motors

To access the general data, double-click the **Machine** entry in the Project Manager tree.

The **General Data Properties** window for a three-phase induction motor contains the following fields:

Machine Type	The machine type you selected when inserting a new RMxprt design (Three Phase Induction Motor).
Number of Poles	The number of poles the machine contains.
Stray Loss Factor	The stray loss factor: the ratio of stray loss to rated output power.
Frictional Loss	The frictional energy loss (due to friction) measured at the reference speed.
Wind Loss	The wind loss (due to air resistance) measured at the reference speed.
Reference Speed	The given speed of reference.

Defining the Stator Data for a Three-Phase Induction Motor

The stator is the outer lamination stack where the three-phase windings reside. In the project tree, double-click **Machine** > **Stator**, **Machine** > **Stator** > **Slot**, and **Machine** > **Stator** > **Winding** to define the physical dimensions, slot data, wires, and conductors for the stator.

To define general stator data:

1. To open the **Stator Data Properties** window, double-click the **Machine** > **Stator** entry in the Project Manager tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the **Outer Diameter** of the stator.
3. Enter the **Inner Diameter** of the stator.
4. Enter the length of the stator core in the **Length** field.
5. Enter the stacking factor for the stator core in the **Stacking Factor** field.
6. Select a **Steel Type** for the stator core:
    a. Click the button for **Steel Type**.
    The **Select Definition** window appears.
    b. Select a steel type from the list, or define a new steel type.
    c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.
7. Enter the **Number of Slots** in the stator.
8. Select the **Slot Type**:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Click the button for the **Slot Type**.

   The **Select Slot Type** window appears.

b. Select a slot type (available types include 1 through 4).

> **Note:** When you place the mouse cursor over the slot type, a schematic of the selected type appears, displaying the slot dimension variables.

   Optionally, check **User Defined Slot** if you wish to define the slot dimensions using the **Slot Editor**.

c. Click **OK** to close the **Select Slot Type** window and return to the **Properties** window.

9. Enter the number of sectors in the **Lamination Sectors** field.
10. Enter the thickness of the magnetic end pressboard in the **Pressboard Thickness** field. Enter **0** for a non-magnetic end pressboard.
11. Enter the skew width, measured in slot number, in the **Skew Width** field.
12. Click **OK** to close the **Properties** window.

Stator Data for Three-Phase Induction Motors

To access the general stator data, double-click the **Machine** > **Stator** entry in the project tree.

The **Stator Data Properties** window contains the following fields:

Outer Diameter	The outer diameter of the stator core
Inner Diameter	The inner diameter of the stator core
Length	The length of the stator core
Stacking Factor	The stacking factor of the stator core
Steel Type	The steel type of the stator core. Click the button to open the **Select Definition** window.
Number of Slots	The number of slots the stator core contains
Slot Type	The type of slots in the stator core. Click the button to open the **Select Slot Type** window.
Lamination Sectors	The number of lamination sectors
Pressboard Thickness	The magnetic press board thickness (0 for a non-magnetic press board)
Skew Width	The skew width measured in slot number

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining the Stator Slots for a Three-Phase Induction Motor

> **Note:** If you chose **User Defined Slot** in the **Select Slot Type** window, you must define the slot dimensions using the **Slot Editor**.

To define the stator slots:

1. To open the **Stator Slot Data Properties** window, double-click the **Machine** > **Stator** > **Slot** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Optionally, to automatically design the dimensions of slots **Hs2**, **Bs1**, and **Bs2**, select the **Auto Design** check box.
3. Optionally, to design dimensions of slots **Bs1** and **Bs2** based on the stator tooth width, select the **Parallel Tooth** check box, and enter a value in the **Tooth Width** field.
4. Enter the available slot dimensions.

   Hs0 Always available

   Hs2 Available only when **Auto Design** is cleared. When **Auto Design** is selected, this slot dimension is determined automatically.

   Bs0 Always available

   Bs1 Available only when **Auto Design** and **Parallel Tooth** are both cleared. When **Auto Design** is selected, this slot dimension is determined automatically. When **Parallel Tooth** is selected, this slot dimension is determined based on the value entered in the **Tooth Width** field.

   Bs2 Available only when **Auto Design** and **Parallel Tooth** are both cleared. When **Auto Design** is selected, this slot dimension is determined automatically. When **Parallel Tooth** is selected, this slot dimension is determined based on the value entered in the **Tooth Width** field.

   Rs Rs is added when the slot type is 3 or 4.
5. Click **OK** to close the **Properties** window.

Stator Slot Data for Three-Phase Induction Motors

To access the stator slot data, double-click the **Machine** > **Stator** > **Slot** entry in the Project Manager tree.

The **Stator Slot Data Properties** window contains the following fields:

Auto Design	Select or clear this to enable or disable auto-design of slots **Hs2**, **Bs1**, and **Bs2**. When this check box is selected, only two other fields appear in the window: **Hs0** and **Bs0**.
Parallel Tooth	Select this to design **Bs1** and **Bs2** based on the tooth width. When this check box is selected, the **Bs1** and **Bs2** fields are removed, and the **Tooth Width** field is

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	added.
Tooth Width	The tooth width for the parallel tooth, on which **Bs1** and **Bs2** are designed.
Hs0	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected).
Hs2	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected).
Bs0	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected).
Bs1	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected).
Bs2	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected).
Rs	A slot dimension. (see the diagram shown in the modeling window when **Machine-Stator-Slot** is selected). Rs is added when the slot type is 3 or 4.

Defining the Stator Windings for a Three-Phase Induction Motor

Define the wires, conductors, insulation, and windings of the stator.

To define the wires and windings:

1. To open the **Stator Slot Winding Properties** window, double-click the **Machine** > **Stator** > **Winding** entry in the project tree on the desktop. You can also enter values in the **Properties** section of the desktop without opening a separate window.
2. Click the **Winding** tab.
3. Enter the number of layers in the stator winding in the **Winding Layers** field.
4. Select a **Winding Type**:
    a. Click the button for **Winding Type**.
    The **Winding Type** window appears.
    b. Select from one of the following three types of winding:
        - **Whole Coiled**
        - **Half Coiled**
        - **Editor**

    When you place the mouse cursor over a winding button, an outline of the selected winding appears. The following table describes the six types of windings that are possible (three for one-layer and three for two-layer):

Type	Description
One layer winding	A user-defined one-layer winding arrangement. You need to set up the winding arrangement for

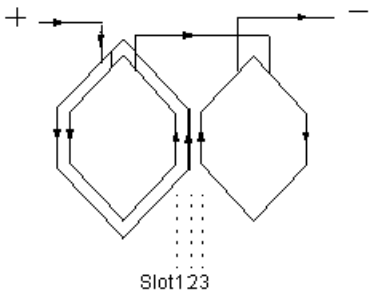
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Editor

each slot.

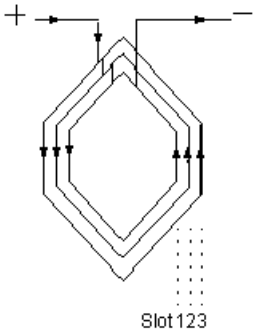
A one-layer whole-coiled winding:

Whole Coiled



A one-layer concentric half-coiled winding:

Half Coiled

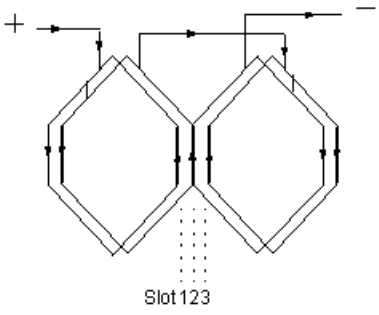


Two Layer Winding Editor

A user-defined two-layer winding arrangement. When you select for winding layers the you can specify a different winding arrangement for each slot in the Winding Editor.

A two-layer whole coiled winding:

Whole Coiled

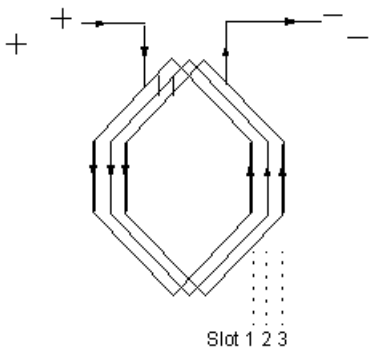


The phase belt for this winding configuration is equal to 360/2*m*, where *m* is the phase number.

Half Coiled

A two-layer half-coiled winding:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

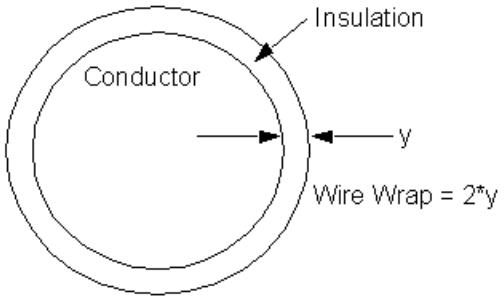


There is only one coil per phase per pair of poles.

> **Note:** For a two-layer winding, if you check **Constant Pitch** in the **Winding Editor**, only the top layer needs to be defined; the bottom layer will be determined according to the coil pitch.

c. Once you have clicked a button to select a winding, click **OK** to close the **Winding Type** window and return to the **Properties** window.

5. Select or enter the number of parallel branches in one phase of the winding in the **Parallel Branches** field.
6. Enter the total number of conductors in each stator slot in the **Conductors per Slot** field. This value is the number of turns per coil multiplied by the number of layers.
7. Enter the coil pitch, measured in number of slots, in the **Coil Pitch** field. The coil pitch is the number of slots separating one winding. For example, if a coil starts in slot 1 and ends in slot 6, it has a coil pitch of 5.
8. Enter the number of wires per conductor in the **Number of Strands** field. Enter 0 to have RMxprt auto-design this value.
9. Enter the thickness of the double-sided wire wrap in the **Wire Wrap** field. Enter 0 to automatically obtain this value from the wire library.



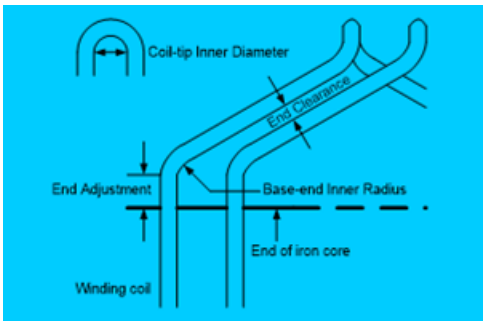
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10. Select the **Wire Size**: Click the button for **Wire Size**. The **Wire Size** window appears.
11. Select a value from the **Wire Diameter** drop-down menu.
12. Select a wire gauge from the **Gauge** drop-down menu. You can select from the following options:

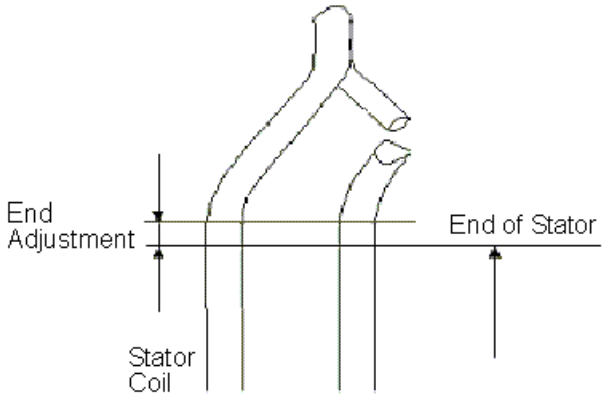
<number>	You can select a specific gauge number. When you select a gauge number, the **Wire Diameter** field is automatically updated.
USER	This option allows you to manually enter the **Wire Diameter**. This is useful when you want to enter a diameter that does not correspond to a particular wire gauge.
AUTO	This option sets the **Wire Diameter** to zero, and RMxprt automatically calculates the optimal value. The diameter information is then written to the output file when you analyze the design.
MIXED	This option allows you to define a conductor that is made of different size wires. For example, a single conductor may consist of 5 wires, 3 wires with a diameter of 0.21 mm and 2 with a diameter of 0.13 mm.

    The gauge number is based on AWG settings. You can create your own wire table using **Machine > Wire**, and then you can select this wire table using the **Tools** > **Options** > **General Options** > **Machines** command.

13. When you are done setting the wire size, click **OK** to close the **Wire Size** window and return to the **Properties** window.
14. Click the **End/Insulation** tab.
15. Select or clear the **Input Half-turn Length** check box.
16. Do one of the following:
    - If you selected **Input Half-turn Length**, then enter the half-turn length of the armature winding in the Half Turn Length field.
    - If you cleared **Input Half-turn Length**, then enter the end length adjustment of the stator coils in the **End Adjustment** field. The end adjustment is the distance one end of the conductor extends vertically beyond the end of the stator.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



17. Enter the inner radius of the base corner in the **Base Inner Radius** field.
18. Enter the inner diameter of the coil tip in the **Tip Inner Diameter** field.
19. Enter the distance between two stator coils in the **End Clearance** field.
20. Enter the thickness of the slot liner insulation in the **Slot Liner** field.
21. Enter the thickness of the wedge insulation in the **Wedge Thickness** field.
22. Enter the thickness of the insulation layer in the **Layer Insulation** field.
23. Enter the limited slot fill factor for the wire design in the **Limited Fill Factor** field. The slot fill factor is the ratio between the cross-sectional area of all conductors in one slot and the entire slot area.
24. Click **OK** to close the **Properties** window.

Stator Winding Data for Three-Phase Induction Motors

To access the stator winding data, double-click the **Machine** > **Stator** > **Winding** entry in the Project Manager tree.

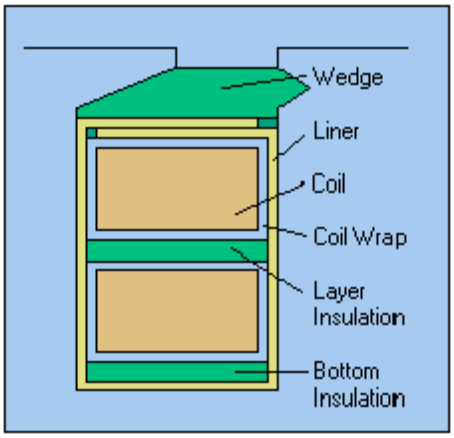
The **Stator Winding Data Properties** window contains the following fields:

Winding tab	**Winding Layers**	The number of winding layers.
	Winding Type	The type of stator winding. Click the button to open the **Winding Type** window and choose from **Whole Coiled**, **Half Coiled**, and **Editor**.
	Parallel Branches	The number of parallel branches in the stator winding.
	Conductors per Slot	The number of conductors per stator slot (0 for auto-design).
	Coil Pitch	The coil pitch measured in number of slots.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Number of Strands	The number of wires per conductor (0 for auto-design).
	Wire Wrap	The thickness of the double-sided wire wrap (0 to automatically obtain this value from the wire library).
	Wire Size	The diameter of the wire (0 for auto-design). Click the button to open the **Wire Size** window where you can specify units, wire type, diameter, and gauge.
End/Insulation tab	**Input Half-turn Length**	Select or clear this check box to specify whether or not you want to enter the half-turn length. When this check box is selected, the **Half Turn Length** field appears the next time you open the **Properties** window. When this check box is selected, the **End Adjustment** field appears instead.
	Half Turn Length	The half-turn length of the armature winding.
	End Adjustment	The end length adjustment of the stator coils, which is the distance one end of the conductor extends vertically beyond the end of the stator.
	Base Inner Radius	The inner radius of the base corner.
	Tip Inner Diameter	The inner diameter of the coil tip.
	End Clearance	The end clearance between two adjacent coils.
	Slot Liner	The thickness of the slot liner insulation.
	Wedge Thickness	The thickness of the wedge insulation.
	Layer Insulation	The thickness of the insulation layer.
	Limited Fill Factor	The limited slot fill factor for the wire design. The slot fill factor is the ratio between the cross-sectional area of all conductors in one slot and the entire slot area.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Winding Editor for a Three-Phase Induction Motor

For a three-phase induction motor, you may want to specify a different number of conductors for each stator slot. The **Winding Editor** makes this possible by enabling you to specify the number of turns for each coil. To enable the **Winding Editor**, you must have set the **Winding Property** for the **Winding Type** to **Editor**.

To specify the number of turns for each coil:

1. Click **Machine** > **Winding** > **Edit Layout**.

   The **Winding Editor** window appears.

2. In the table in the upper left, set which phase you want for each coil and which slot is the “in” and “out” slot for the current in each coil.
3. If you are working on a quarter or half model, you may want to specify a multiplier by clicking the **Periodic Multiplier** check box and specifying a value.
4. Select or deselect the **Constant Turns** or **Constant Pitch** check boxes, depending on whether you want to be able to change these setting in the table above. When these options are selected, you cannot change the turns or pitch.
5. When you are satisfied with the coil settings, click **OK** to close the **Winding Editor** window.

Defining Different Size Wires for a Three-Phase Induction Motor

Use the **Gauge** option if you have a conductor that is made up different size wires.

To define different size wires:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the **Wire Size** window, select **MIXED** from the **Gauge** drop-down menu.
2. Select either **Round** or **Rectangular** as the **Wire Type**.
3. Enter the appropriate wire data in the table:
    - For a round wire:
        - Enter the **Diameter** in the table.
        - Enter a **Number** in the table to specify how many of the conductor's wires have this diameter.
    - For a rectangular wire:
        - Enter the **Width** of the wire in the table.
        - Enter the **Thickness** of the wire in the table.
        - Enter the **Fillet** value in the table.
        - Enter a **Number** in the table to specify how many of the conductor's wires have this data.
4. Choose **Add** to add the new wire data.
5. Repeat steps 3 and 4 for each size wire you want to add.
6. When you are finished defining the wires, click **OK** to close the **Wire Size** window.

> **Note:** For example, if one conductor is made up of five wires, and three of those wires have a diameter of 0.21 mm, and the other 2 have a diameter of 0.13 mm, then the mixed wire size table will have two lines. The first line will list Diameter = 0.21 and Number = 3. The second line will list Diameter = 0.13 and Number = 2. An equivalent wire diameter is displayed as Wire Size value in the Winding tab in the Properties window.

Stator Vent Data for Three-Phase Induction Motors

By option, you can add a vent to a three-phase induction motor.

To add a vent to stator in a three-phase induction motor:

1. Select the stator icon in the Project Manager tree.
2. Right-click to display the drop-down list and select **Insert Vent**.

The vent icon appears in the Project Manager tree under the stator.

To remove a vent to stator in a three-phase induction motor:

1. Select the stator icon in the Project Manager tree.
2. Right-click to display the drop-down list and select **Remove Vent**.

The vent icon disappears in the Project Manager tree under the stator.

The Vent data for the stator includes the following fields.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Vent Ducts	Number of radial vent ducts
Duct Width	Width of radial vent ducts
Magnetic Spacer Width	Width of magnetic spacer which holds vent ducts; 0 for non magnetic spacer.
Duct Pitch	Center-to-Center distance between two adjacent Vent ducts

Defining the Rotor Data for a Three-Phase Induction Motor

The rotor consists of copper bars in which current is induced by the magnetic fields produced by the stator windings. In the project tree, double-click **Machine** > **Rotor**, **Machine** > **Rotor** > **Slot**, and **Machine** > **Rotor** > **Winding** to define the rotor slots and vents.

To define general stator data:

1. To open the **Rotor Data Properties** window, double-click the **Machine** > **Rotor** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the stacking factor for the rotor core in the **Stacking Factor** field.
3. Enter the **Number of Slots** in the rotor.
4. Select the **Slot Type**:
    a. Click the button for the **Slot Type**.
    The **Select Slot Type** window appears.
    b. Select a slot type (available types include 1 through 4).

    > **Note:** When you place the mouse cursor over the slot type, a schematic of the selected type appears, displaying the slot dimension variables.

    c. Click **OK** to close the **Select Slot Type** window and return to the **Properties** window.
5. Enter the outer diameter of the rotor in the **Outer Diameter** field.
6. Enter the inner diameter of the rotor in the **Inner Diameter** field.
7. Enter the length of the rotor core in the **Length** field.
8. Select a **Steel Type** for the rotor core:
    a. Click the button for **Steel Type**.
    The **Select Definition** window appears.
    b. Select a steel type from the list, or define a new steel type.
    c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.
9. Enter the **Skew Width**, measured in rotor slot pitch. This value defines by how much the rotor bars are skewed.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10. Optionally, select **Cast Rotor** to allow the conductor to fill all the space available in the slot. Otherwise, RMxprt assumes the slot wedge that fixes the bars is filled with insulator material in a 2D/3D geometry model.
11. Optionally, select **Half Slot** to draw only half of the rotor slots.
12. Optionally, select **Double Cage** to specify the winding as a double-squirrel-cage winding.

    If you select Double Cage, another line appears in the properties to let you specify the Bottom Slot type.

    a. Click on the Custom button on the Double Cage row. This displays the Select Slot Type window.
    The **Select Slot Type** window appears.
    b. Select a slot type (available types include 1 through 4).

> **Note:** When you place the mouse cursor over the slot type, a schematic of the selected type appears, displaying the slot dimension variables.

13. Click **OK** to close the **Properties** window.

Rotor Data for Three-Phase Induction Motors

To access the general rotor data, double-click the **Machine** > **Rotor** entry in the Project Manager tree.

The **Rotor Data Properties** window contains the following fields:

Stacking Factor	The stacking factor of the rotor core
Number of Slots	The number of slots the rotor core contains
Slot Type	The type of slots in the rotor core. Click the button to open the **Select Slot Type** window.
Outer Diameter	The outer diameter of the rotor core
Inner Diameter	The inner diameter of the rotor core
Length	The length of the rotor core
Steel Type	The steel type of the rotor core. Click the button to open the **Select Definition** window.
Skew Width	The skew width measured in slot number.
Cast Rotor	Select or clear this to specify whether the rotor squirrel-cage winding is cast or not

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Half Slot	Select this to specify a half-shaped unsymmetrical slot
Double Cage	Select this to specify the winding as double-squirrel-cage

Defining the Rotor Slots for a Three-Phase Induction Motor

To define the type and dimensions of the rotor's slots:

1. To open the **Rotor Data Slot Properties** window, in the Project Manager tree, double-click **Machine** > **Rotor** > **Slot**. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the slot dimensions in the following fields: **Hs0**, **Hs01**, **Hs2**, **Bs0**, **Bs1**, **Bs2**, **Rs**.
3. Click **OK** to close the **Properties** window.

Rotor Slot Data for Three-Phase Induction Motors

To access the rotor slot data, double-click **Machine** > **Rotor** > **Slot** entry in the Project Manager tree.

The **Rotor Slot Data Properties** window contains the following fields:

Hs0	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected)
Hs01	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Motor** > **Slot** is selected)
Hs2	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Motor** > **Slot** is selected)
Bs0	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Motor** > **Slot** is selected)
Bs1	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Motor** > **Slot** is selected)
Bs2	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Motor** > **Slot** is selected)
Rs	A slot dimension. (see the diagram shown in the modeling window when **Machine** > **Motor** > **Slot** is selected). Rs is added when the slot type is 3 or 4.

Defining the Rotor Winding for a Three-Phase Induction Motor

To define the rotor winding data:

1. To open the **Rotor Data Slot Properties** window, double-click **Machine** > **Rotor** > **Winding** entry in the Project Manager tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select a **Bar Conductor Type** for the rotor winding bar:
    a. Click the button for **Bar Conductor Type**.
    The **Select Definition** window appears.
    b. Select a conductor type from the list, or define a new conductor type.
    c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.
3. Enter the length of the gap between the end ring and the iron core in the **End Length** field. This field specifies the value for only one end of the gap, not both.
4. Enter the end ring dimension in the axial direction in the **End-Ring Width** field. The end ring connects the bars of the rotor to one another.
5. Enter the end ring dimension in the radius direction in the **End-Ring Height** field. The end ring's height covers at least the cross-section of the rotor conductor.
6. Select an **End Ring Conductor Type** for the rotor winding end ring:
    a. Click the button for **End Ring Conductor Type**.
    The **Select Definition** window appears.
    b. Select a conductor type from the list, or define a new conductor type.
    c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.
7. Click **OK** to close the **Properties** window.

Rotor Winding for Three-Phase Induction Motors

To access the rotor winding data, double-click **Machine** > **Rotor** > **Winding** entry in the project tree.

The **Rotor Winding Data Properties** window contains the following fields:

Bar Conductor Type	The type of bar conductor used in the winding. Click the button to open the **Select Definition** window.
End Length	The length of the single-side end of the extended bar
End Ring Width	The width of one side of the end rings in the axial direction. The end ring connects the bars of the rotor to one another.
End Ring Height	The height of the end rings in the radian direction. The end ring connects the bars of the rotor to one another.
End Ring Conductor Type	The type of end ring conductor used in the winding. Click the button to open the **Select Definition** window.

Rotor Vent Data for Three-Phase Induction Motors

By option, you can add a vent to a rotor in a three-phase induction motor.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To add a vent to rotor:

1. Select the rotor icon in the project tree.
2. Right-click to display the pop-up menu and select **Insert Vent**.

The vent icon appears in the project tree under the rotor.

To remove a vent to stator in a three-phase induction motor.

1. Select the rotor icon in the project tree.
2. Right-click to display the pop-up menu and select **Remove Vent**.

The vent icon disappears in the project tree under the stator.

The Vent data for the rotor includes the following fields.

Vent Ducts	Number of radial vent ducts
Duct Width	Width of radial vent ducts
Magnetic Spacer Width	Width of magnetic spacer which holds vent ducts. 0 for non-magnetic spacer
Duct Pitch	Center to center distance between vent ducts
Holes per row	Number of axial vent holes per row
Inner hole diameter	Diameter of vent holes in inner row
Outer hole diameter	Diameter of vent holes in outer row
Inner hole location	Center to center diameter of inner hole vents
Outer hole location	Center to center diameter of outer hole vents

Defining the Shaft Data for a Three-Phase Induction Motor

To define the shaft:

1. To open the **Shaft Data Properties** window, double-click the **Machine** > **Shaft** entry in the Project Manager tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Select or clear the **Magnetic Shaft** check box to specify whether or not the shaft is to be made of magnetic material.
3. Click **OK** to close the **Properties** window.

Access Shaft Data for Three-Phase Induction Motors

To access the shaft data, double-click the **Machine** > **Shaft** entry in the Project Manager tree.

The **Shaft Data Properties** window contains the following fields:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Magnetic Shaft	Select or clear this check box to indicate whether or not the shaft is made of magnetic material.

Setting Up Analysis Parameters for a Three-Phase Induction Motor

To define the solution data:

1. To open the Solution Setup window, right-click **Analysis** in the project tree, and click **Add Solution Setup**.
2. Click the **General** tab. The **Operation Type** is automatically set to **Motor** for this machine type.
3. Select the **Load Type** used in the motor from the following options:

Const Speed	The speed remains constant in the motor.
Const Power	The output power remains constant in the motor.
Const Torque	The torque remains constant regardless of the speed. In this case, $T_{load} = T_{rated}$, given by the output power divided by the given rated speed.
Linear Torque	The torque increases linearly with speed. In this case, $T_{load} = T_{rated} * (n/n_{rated})$ where T_{rated} is given by the output power divided by the given rated speed.
Fan Load	The load varies nonlinearly with speed. In this case, $T_{load} = T_{rated} * (n/n_{rated})^2$ where T_{rated} is given by the output power divided by the given rated speed.

4. Enter the output power developed at the shaft of the motor in the **Rated Output Power** field.
5. Enter the RMS line-to-line voltage in the **Rated Voltage** field.
6. Enter the desired rotor speed in the **Rated Speed** field. When input Rated Speed is less than the Synchronous Speed, the machine operates as a motor. When input Rated Speed is greater than the Synchronous Speed, the machine operates as a generator. For an induction generator, the rated performance will be calculated at the input Rated Speed with the three-phase windings connecting to infinite bus.
7. Enter the temperature at which the system functions in the **Operating Temperature** field.
8. Click the **Three-Phase Induction Motor** tab.
9. Enter the electrical line frequency in the **Frequency** field, and select the units.
10. Select the **Winding Connection** from the following options:
    - **Wye** (Y)
    - **Delta**
11. Click **OK** to close the **Solution Setup** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution Data for Three-Phase Induction Motors

To access the solution data, right-click **Analysis** in the project tree, and click **Add Solution Setup**.

The **Solution Setup** window contains the following fields:

Operation Type	On the **General** tab. The operation type is automatically set to **Motor** for this machine type.
Load Type	On the **General** tab. Select from **Const Speed**, **Const Power**, **Const Torque**, **Linear Torque**, and **Fan Load**. The default is **Const Power**.
Rated Output Power	On the **General** tab. Type a value for the rated output voltage, and select the units.
Rated Voltage	On the **General** tab. Type a value for the rated voltage, and select the units.
Rated Speed	On the **General** tab. Type a value for the rated speed, and select the units.
Operating Temperature	On the **General** tab. Type a value for the operating temperature, and select the units.
Frequency	On the **Three-Phase Induction Motor** tab. Type a value for the frequency, and select the units.
Winding Connection	On the **Three-Phase Induction Motor** tab. Select from **Wye** or **Delta**.

Single-Phase Induction Motors

After you have selected **Single-Phase Induction Motors** as your model type, you must define the following:

- General data, such as the voltage, speed, and materials used in the motor.
- Stator data, such as the slot types, diameter, and wire dimensions.
- Rotor data, such as the slot dimensions, skew width, and ventage holes.
- Solution data, such as rated output voltage and frequency.

By option, you can:

- Add a machine housing.
- Add a vent or remove a vent from the rotor.

Analysis Approach for Single-Phase Induction Motors

The construction of a single-phase induction motor is structurally similar to the polyphase squirrel-cage induction motors. The primary difference is that the stator windings, which consist of a main winding and an auxiliary winding, have axes of these that are displaced 90 electrical

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

degrees in space. To produce a starting torque, the currents in the two windings must be out of phase. Usually a capacitor is connected in series with the auxiliary winding so that the auxiliary winding current is forced to lead the main winding current by about 90 electrical degrees. Two parallel capacitors can also be used: one for starting, and one for running, so that both a starting and running performance are obtained.

An algorithm called symmetric component method is applied to analyze single-phase induction motors (IndM1). Both voltages and currents of the main-phase and auxiliary-phase windings are decomposed to positive- and negative-sequence components. The equivalent circuits for main-phase positive-sequence components, auxiliary-phase positive-sequence components, main-phase negative-sequence components, and auxiliary-phase negative-sequence components are shown in (a), (b), (c), and (d) of Figure 3, respectively.

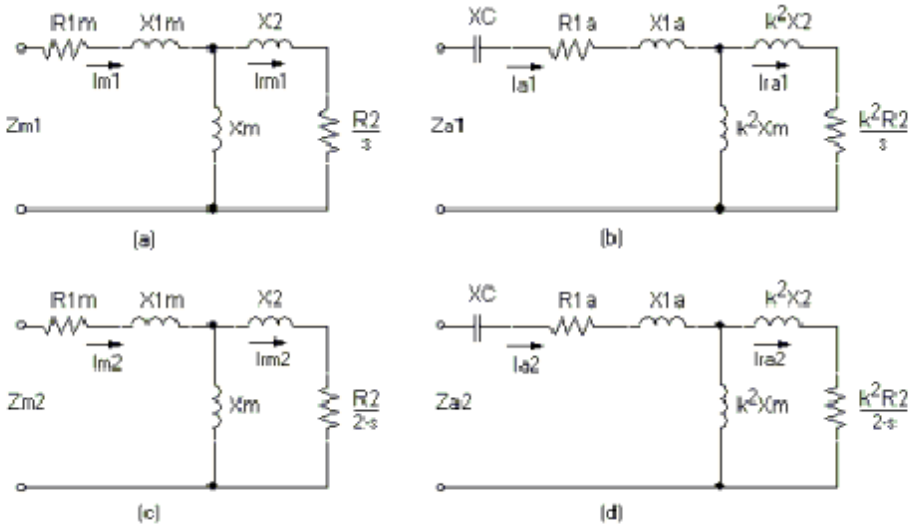


Figure 3

In the figures, R1m, X1m, R1a, X1a, R2, X2, and Xm are main-phase stator resistance, main-phase stator leakage reactance, auxiliary-phase stator resistance, auxiliary-phase stator leakage reactance, rotor resistance, rotor leakage reactance, and magnetizing reactance, respectively. XC is the reactance of the capacitor connected in series with the auxiliary winding, and the coefficient k is the ratio of effective turns of the auxiliary winding to that of the main winding. R2, X2, and Xm have been referred to the main winding. The equivalent impedance of the four circuits is Z_{m1}, Z_{a1}, Z_{m2}, and Z_{a2}, as shown in the figures.

According to the symmetric component method, the positive and negative components of auxiliary-phase currents can be expressed in the form of a phasor as the following:

$$\boldsymbol{I}_{a1} = (j / k)\boldsymbol{I}_{m1}$$
$$\boldsymbol{I}_{a2} = (j / k)\boldsymbol{I}_{m2}$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Because the main winding and the auxiliary winding have the same applied terminal voltage U_1, the voltage equations for both windings become the following:

$$\boldsymbol{U}_1 = \boldsymbol{U}_{m1} + \boldsymbol{U}_{m2} = \boldsymbol{I}_{m1}\boldsymbol{Z}_{m1} + \boldsymbol{I}_{m2}\boldsymbol{Z}_{m2}$$
$$\boldsymbol{U}_1 = \boldsymbol{U}_{a1} + \boldsymbol{U}_{a2} = \boldsymbol{I}_{a1}Z_{a1} + \boldsymbol{I}_{a2}Z_{a2} = (j / k)(\boldsymbol{I}_{m1}Z_{a1} - \boldsymbol{I}_{m2}Z_{a2})$$

The positive and negative components of main-phase current are calculated by the following:

$$\boldsymbol{I}_{m1} = \boldsymbol{U}_1(Z_{a2} - jkZ_{m2}) / (Z_{m1}Z_{a2} + Z_{m2}Z_{a1})$$
$$\boldsymbol{I}_{m2} = \boldsymbol{U}_1(Z_{a1} + jkZ_{m1}) / (Z_{m1}Z_{a2} + Z_{m2}Z_{a1})$$

The total input current is

$$\boldsymbol{I}_1 = \boldsymbol{I}_m + \boldsymbol{I}_a = (\boldsymbol{I}_{m1} + \boldsymbol{I}_{m2}) + (\boldsymbol{I}_{a1} + \boldsymbol{I}_{a2})$$

Based on these two components of main-phase current, all current components shown in Figure 3 can be obtained by simple computation.

Then the total input current is

$$\boldsymbol{I}_1 = \boldsymbol{I}_m + \boldsymbol{I}_a = (\boldsymbol{I}_{m1} + \boldsymbol{I}_{m2}) + (\boldsymbol{I}_{a1} + \boldsymbol{I}_{a2})$$

The positive- and negative-sequence air-gap power can be computed in the following way:

$$P_{m1} = 2 * Ir^2_{m1} * R2 / s$$
$$P_{m2} = 2 * Ir^2_{m2} * R2 / (2 - s)$$

The total air-gap power is

$$P_m = P_{m1} - P_{m2}$$

T_m, T_2, P_2, P_1, and eff are computed in the same way as for three-phase induction motors.

The power factor is derived from:

$$P_F = P_1 / (U_1 * I_1)$$

Defining a Single-Phase Induction Motor

The general procedure for structurally defining a single-phase induction motor is as follows:

1. Insert a single-phase induction motor design in an existing or newly created project.
2. Double-click the **Machine** entry in the Project Manager tree to define the general data.
3. Double-click the **Machine** > **Stator** entry in the Project Manager tree to define the stator geometry.
4. Double-click the **Machine** > **Stator** > **Slot** entry in the Project Manager tree to define the stator slot dimensions.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Double-click the **Machine** > **Stator** > **Winding** entry in the Project Manager tree to define the stator windings and conductors.
6. Double-click the **Machine** > **Rotor** entry in the project tree to define the rotor geometry.
7. Double-click the **Machine** > **Rotor** > **Slot** entry in the project tree to define the rotor slot dimensions.
8. Double-click the **Machine** > **Rotor** > **Winding** entry in the Project Manager tree to define the rotor conductor, ventage hole dimensions, and skew.
9. Double-click the **Machine-Shaft** entry in the Project Manager tree to define the magnetism of the shaft.
10. Optionally, for Inner Rotor machines only, right-click the **Machine** entry in the Project Manager tree and click **Insert Housing** to add a machine housing.
11. Right-click **Analysis** in the Project Manager tree, and click **Add Solution Setup** to define the solution data.
12. Choose **File** > **Save** to save the project.
13. Choose **RMxprt** > **Analyze** to analyze the design.

> **Note:** When you place the cursor over an entry field in the data windows, a brief description of that field appears in the status bar at the bottom of the **RMxprt** window.

Once the design is analyzed, the model can be imported into the Maxwell 2D Modeler, or can be used to create a new Maxwell 2D project, and a new Maxwell 3D design.

For a specific example of a single-phase induction motor problem, see the *A Capacitor-Run Single-Phase Induction Motor Problem* application note, on the technical support page of the Ansys web site.

Defining the General Data for a Single-Phase Induction Motor

Use the **General Data Properties** window to define the basic parameters of the induction motor, such as the number of poles, frictional loss, and operation mode.

To define the general data:

1. To open the **General Data Properties** window, double-click the **Machine** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the number of poles for the machine in the **Number of Poles** field. This value is the total number of poles in the stator (or the number of pole pairs multiplied by two).
3. Select one of the following for the **Rotor Position**:
   - **Inner Rotor**
   - **Outer Rotor**
4. Enter the energy loss due to friction at the given speed in the **Frictional Loss** field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Enter the wind loss due to air resistance measured at the reference speed in the **Wind Loss** field.
6. Enter the given speed in the **Reference Speed** field.
7. Select the **Operation Mode**:
   a. Click the button.

      The **Select Operation Mode** window appears.

   b. Select from one of the following:

C-Run	Capacitance-run mode. The capacitor is in series with the auxiliary winding. In the Capacitor run mode, the capacitor will be designed (if the auto-design mode is selected) to minimize the backward magnetomotive force.
C-Start	Capacitance-start mode. The auxiliary winding is in series with the capacitor and is disconnected when the rotor reaches the switching speed.
C-R&S	Capacitance-run and start mode. Two capacitors are in series with the auxiliary winding; one for starting, one for running.
R-Start	Resistor-start mode. The auxiliary winding is disconnected when the rotor reaches the switching speed.

   c. Click **OK** to close the **Select Operation Mode** window and return to the **Properties** window.
8. Enter values in the following capacitance, resistance, and switching speed fields:

Run Capacitance	Available for C-Run, C-R&S
Run Resistance	Available for C-Run, C-R&S
Start Capacitance	Available for C-Start, C-R&S
Start Resistance	Available for C-Start, C-R&S
Switching Speed	Available for C-Start, C-R&S, R-Start

9. If the start winding needs to be optimized, select the **Objective Type** from the following three options:
   - **(Tst/Ist)max**. Accept the defaults. This is the ratio of the maximum starting torque to the starting current ratio.
   - **(Tst)max**. Enter the given start current ratio. This is the maximum starting torque
   - **(Ist)min (minimum starting current). Enter the given start torque ratio.**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** The start-winding optimization goal is disabled for the **C-Run** operation mode. In capacitor-run mode, the capacitor is designed to minimize the backward magnetomotive force. For other modes, if the auto-design function is active, the capacitor and the resistance are designed according to the start goal, selected from the following:
>
> - The maximum value of (Starting Torque/Starting Current).
> - The maximum starting torque.
> - The minimum starting current.

10. Click **OK** to close the **Properties** window.

General Data for Single-Phase Induction Motors

To access the general data, double-click the **Machine** entry in the project tree.

The **General Data Properties** window for a three-phase induction motor contains the following fields:

Machine Type	The machine type you selected when inserting a new RMxprt design (Single Phase Induction Motor).
Number of Poles	The number of poles the machine contains.
Rotor Position	Select whether the rotor is an **Inner Rotor** or **Outer Rotor**.
Frictional Loss	The frictional energy loss (due to friction) measured at the reference speed.
Wind Loss	The wind loss (due to air resistance) measured at the reference speed.
Reference Speed	The given speed of reference.
Operation Mode	Click the button to select from the following four modes: **C-Run**, **C-Start**, **C-R&S**, and **R-Start**.
Run Capacitance	The capacitance of the run capacitor. Available for C-Run and C-R&S operation modes.
Run Resistance	The resistance of the run capacitor. Available for C-Run and C-R&S operation modes. See **Note** below.
Start Capacitance	The resistance of the start capacitor. Available for C-Start and C-R&S operation modes.
Start Resistance	The resistance of the start capacitor. Available for C-Start and C-R&S operation modes.
Switching	The switching speed of the capacitor or resistor. Available for C-Start, C-

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Speed	R&S, and R-Start operation modes.
Objective Type	If the start winding needs to be optimized, select from the following three objective types: **(Tst/Ist)max**, **(Tst)max**, or **(Ist)min**.

• For **(Tst/Ist) max**, accept the defaults. This is the ratio of the maximum starting torque to the starting current ratio.

• For **(Tst) max**, enter the **Given Start Current Ratio**. This is the maximum starting torque.

• For **(Ist) min** (minimum starting current), enter the **Given Start Torque Ratio**.

The start-winding optimization goal is disabled for the **C-Run** operation mode.

> **Note:** When exporting the RMxprt model to Maxwell, if the value of the **Run Resistance** is **zero** in RMxprt, the value of the **Run Resistance** will be autocomputed in Maxwell to a value of 1% of the capacitor reactance.
> To neglect the **Run Resistance** in Maxwell, set the value to a small non-zero number in RMxprt.

Defining the Stator Data for a Single-Phase Induction Motor

The stator is the outer lamination stack where the three-phase windings reside. In the project tree, double-click **Machine** > **Stator**, **Machine** > **Stator** > **Slot**, and **Machine** > **Stator** > **Winding** to define the physical dimensions, slot data, wires, and conductors for the stator.

To define general stator data:

1. To open the **Stator Data Properties** window, double-click the **Machine** > **Stator** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the **Outer Diameter** of the stator.
3. Enter the **Inner Diameter** of the stator.
4. Enter the length of the stator core in the **Length** field.
5. Enter the stacking factor for the stator core in the **Stacking Factor** field.
6. Select a **Steel Type** for the stator core:
    a. Click the button for **Steel Type**.

       The **Select Definition** window appears.

    b. Select a steel type from the list, or define a new steel type.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.

7. Enter the **Number of Slots** in the stator.
8. Select the **Slot Type**:
   a. Click the button for the **Slot Type**.

      The **Select Slot Type** window appears.

   b. Select a slot type (available types include 1 through 4).

      > **Note:** When you place the mouse cursor over the slot type, a schematic of the selected type appears, displaying the slot dimension variables.

   c. Click **OK** to close the **Select Slot Type** window and return to the **Properties** window.
9. Enter the overall width of the stator outer profile in the **Overall Width** field.
10. Click **OK** to close the **Properties** window.

Stator Data for Single-Phase Induction Motors

To access the general stator data, double-click the **Machine** > **Stator** entry in the project tree.

The **Stator Data Properties** window contains the following fields:

Outer Diameter	The outer diameter of the stator core
Inner Diameter	The inner diameter of the stator core
Length	The length of the stator core
Stacking Factor	The stacking factor of the stator core
Steel Type	The steel type of the stator core. Click the button to open the **Select Definition** window
Number of Slots	The number of slots the stator core contains
Slot Type	The type of slots in the stator core. Click the button to open the **Select Slot Type** window
Overall Width	The overall width of the stator outer profile

Defining the Stator Slots for a Single-Phase Induction Motor

Use the **Stator1** window to define the physical dimensions of the stator slots.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To define the stator slots:

1. To open the **Stator Slot Data Properties** window, double-click the **Machine** > **Stator** > **Slot** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Optionally, to automatically design the dimensions of slots **Hs2**, **Bs1**, and **Bs2**, select the **Auto Design** check box.
3. Optionally, to design dimensions of slots **Bs1** and **Bs2** based on the stator tooth width, select the **Parallel Tooth** check box, and enter a value in the **Tooth Width** field.
4. Enter the available slot dimensions.

   Hs0 Always available

   Hs1 Always available

   Hs2 Available only when **Auto Design** is cleared. When **Auto Design** is selected, this slot dimension is determined automatically.

   Bs0 Always available

   Bs1 Available only when **Auto Design** and **Parallel Tooth** are both cleared. When **Auto Design** is selected, this slot dimension is determined automatically. When **Parallel Tooth** is selected, this slot dimension is determined based on the value entered in the **Tooth Width** field.

   Bs2 Available only when **Auto Design** and **Parallel Tooth** are both cleared. When **Auto Design** is selected, this slot dimension is determined automatically. When **Parallel Tooth** is selected, this slot dimension is determined based on the value entered in the **Tooth Width** field.

   Rs Rs is added when the slot type is 3 or 4.
5. Click **OK** to close the **Properties** window.

Stator Slot Data for Single-Phase Induction Motors

To access the stator slot data, double-click the **Machine** > **Stator** > **Slot** entry in the project tree.

The **Stator Slot Data Properties** window contains the following fields:

Auto Design	Select or clear this to enable or disable auto-design of slots **Hs2**, **Bs1**, and **Bs2**. When this check box is selected, only two other fields appear in the window: **Hs0** and **Bs0**.
Parallel Tooth	Select this to design **Bs1** and **Bs2** based on the tooth width. When this check box is selected, the **Bs1** and **Bs2** fields are removed, and the **Tooth Width** field is added.
Tooth Width	The tooth width for the parallel tooth, on which **Bs1** and **Bs2** are designed.
Hs0	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected).
Hs1	A slot dimension (see the diagram shown in the modeling window when **Machine**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Stator** > **Slot** is selected).

Hs2 A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected).

Bs0 A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected).

Bs1 A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected).

Bs2 A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected).

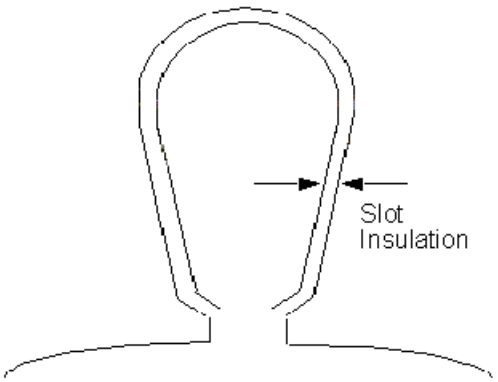
Rs A slot dimension. (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected). Rs is added when the slot type is 3 or 4.

Defining the Stator Windings for a Single-Phase Induction Motor

Define the wires, conductors, insulation, and windings of the stator.

To define the wires and windings:

1. To open the **Stator Winding Properties** window, double-click the **Machine** > **Stator** > **Winding** entry in the Project Manager tree on the desktop. You can also enter values in the **Properties** section of the desktop without opening a separate window.
2. Click the **Winding** tab.
3. Enter the thickness of the slot liner in the **Slot Liner** field.

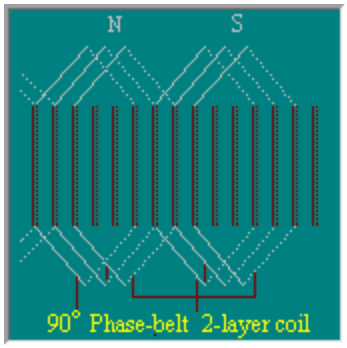


4. Enter the thickness of the wedge insulation in the **Wedge Thickness** field.
5. Enter the limited slot fill factor for the wire design in the **Limited Fill Factor** field. The slot fill factor is the ratio between the cross-sectional area of all conductors in one slot and the entire slot area.

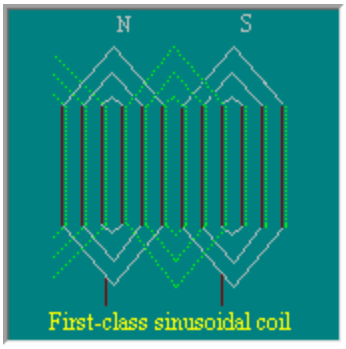
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Select or clear the **Include Series Winding** check box. This option sets whether or not to include the series winding in the speed adjustment. When this option is selected, a third tab, **Series (C)**, appears in the **Properties** window.
7. Enter the number of layers in the **Winding Layers** field.
8. Enter the number of slots in the **Coil Pitch** field.
9. Select a **Winding Type**:
    a. Click the button for **Winding Type**.
    The **Winding Type** window appears.
    b. Select from one of the following types of winding:
        - **Lap**
        - **Sin_1**
        - **Sin_2**
        - **Editor**

    When you place the mouse cursor over a winding button, an outline of the selected winding appears. The following table describes the four types of windings that are possible:

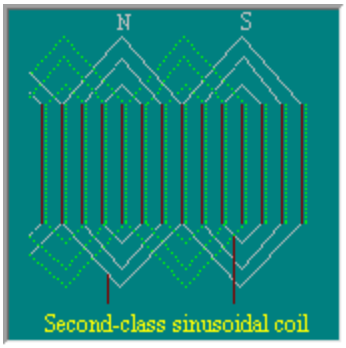
Editor	A user-defined one or two-layer winding arrangement, determined by your **Winding Layers** selection. Enables the Winding Editor, where you can specify a different winding arrangement for each slot.
Lap	90 deg phase belt 2-layer coil for both single and double layer 
Sin_1	A first-class sinusoidal coil for double layer only. The **Conductors per Layer** field defines the maximum number of conductors in the slot. The software will determine the winding distribution in the slots to get the sinusoidal current distribution.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A second-class sinusoidal coil for double layer only. The **Conductors per Layer** field defines the maximum number of conductors in the slot. The software will determine the winding distribution in the slots to get the sinusoidal current distribution.

Sin_2

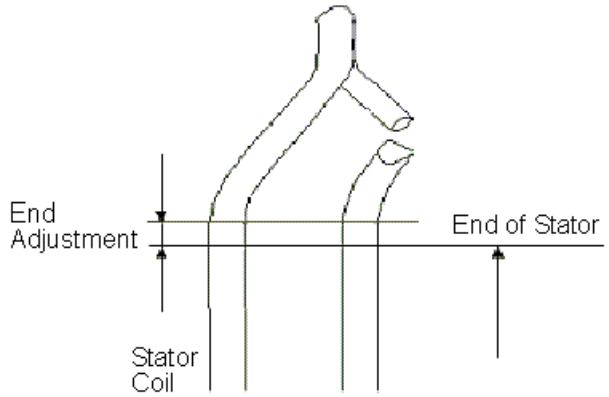


c. Once you have clicked a button to select a winding, click **OK** to close the **Winding Type** window and return to the **Properties** window.

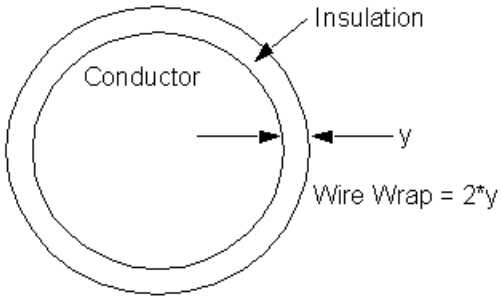
10. Click the **Main (A)** tab.

    a. Enter the end length adjustment of the main stator coil in the **End Extension** field. The end adjustment is the distance one end of the conductor extends vertically beyond the end of the stator.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



b. Enter the number of conductors per layer of main winding in the **Conductors per Layer** field.
c. Enter the number of parallel branches in the main stator winding in the **Parallel Branches** field.
d. Enter the number of wires per conductor in the **Number of Strands** field. Enter 0 to have RMxprt auto-design the value.
e. Enter the thickness of the double-sided wire wrap in the **Wire Wrap** field. Enter 0 to automatically obtain this value from the wire library.



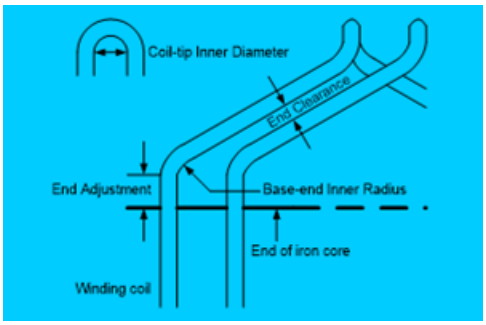
f. To select the **Wire Size**: Click the button for **Wire Size**.
The **Wire Size** window appears.
g. Select a value from the **Wire Diameter** drop-down menu.
h. Select a wire gauge from the **Gauge** drop-down menu. You can select from the following options:

<number>	You can select a specific gauge number. When you select a gauge number, the **Wire Diameter** field is automatically updated.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

USER	This option allows you to manually enter the **Wire Diameter**. This is useful when you want to enter a diameter that does not correspond to a particular wire gauge.
AUTO	This option sets the **Wire Diameter** to zero, and RMxprt automatically calculates the optimal value. The diameter information is then written to the output file when you analyze the design.
MIXED	This option allows you to define a conductor that is made of different size wires. For example, a single conductor may consist of five wires: three wires with a diameter of 0.21 mm and two wires with a diameter of 0.13 mm.

The gauge number is based on AWG settings. You can create your own wire table using **Machine** > **Wire**, and then you can select this wire table using **Tools** > **Options** > **General Options**> **Machines**.



i. Enter the inner radius of the base corner in the **Base Inner Radius** field.

j. Enter the inner diameter of the coil tip in the **Tip Inner Diameter** field.

k. Enter the distance between two stator coils in the **End Clearance** field.

l. Enter the thickness of the slot liner insulation in the **Slot Liner** field.

m. Enter the thickness of the wedge insulation in the **Wedge Thickness** field.

n. Enter the thickness of the insulation layer in the **Layer Insulation** field.

o. Enter the limited slot fill factor for the wire design in the **Limited Fill Factor** field. The slot fill factor is the ratio between the cross-sectional area of all conductors in one slot and the entire slot area.

p. When you are done setting the wire size, click **OK** to close the **Wire Size** window and return to the **Properties** window.

11. Click the **Aux (B)** tab.

a. Enter the end length adjustment of the auxiliary stator coil in the **End Extension** field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Enter the number of conductors per layer of auxiliary winding in the **Conductors per Layer** field.
c. Enter the number of parallel branches in the auxiliary stator winding in the **Parallel Branches** field.
d. Enter the number of wires per conductor in the **Number of Strands** field. Enter 0 to have RMxprt auto-design the value.
e. Enter the thickness of the double-sided wire wrap in the **Wire Wrap** field. Enter 0 to automatically obtain this value from the wire library.
f. To select the **Wire Size**: Click the button for **Wire Size**.
The **Wire Size** window appears.
g. Select a value from the **Wire Diameter** drop-down menu.
h. Select a wire gauge from the **Gauge** drop-down menu. You can select from the following options:

<*number*>	You can select a specific gauge number. When you select a gauge number, the **Wire Diameter** field is automatically updated.
USER	This option allows you to manually enter the **Wire Diameter**. This is useful when you want to enter a diameter that does not correspond to a particular wire gauge.
AUTO	This option sets the **Wire Diameter** to zero, and RMxprt automatically calculates the optimal value. The diameter information is then written to the output file when you analyze the design.
MIXED	This option allows you to define a conductor that is made of different size wires. For example, a single conductor may consist of five wires: three wires with a diameter of 0.21 mm and two wires with a diameter of 0.13 mm.

The gauge number is based on AWG settings. You can create your own wire table using **Machine** > **Wire**, and then you can select this wire table using **Tools** > **Options** > **General Options** > **Machines**.

i. When you are done setting the wire size, click **OK** to close the **Wire Size** window and return to the **Properties** window.

12. Click **OK** to close the **Properties** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stator Winding Data for Single-Phase Induction Motors

To access the stator winding data, double-click the **Machine** > **Stator** > **Winding** entry in the project tree.

The **Stator Winding Data Properties** window contains the following fields:

Winding tab	**Slot Liner**	The thickness of the slot liner
	Wedge Thickness	The thickness of the wedge insulation
	Limited Fill Factor	The limited slot fill factor for the wire design. The slot fill factor is the ratio between the cross-sectional area of all conductors in one slot and the entire slot area.
	Winding Layers	The number of winding layers. Not visible if the **Winding Type** is either **Sin_1** or **Sin_2**.
	Coil Pitch	The coil pitch measured in number of slots. Not visible if the **Winding Type** is either **Sin_1** or **Sin_2**.
	Winding Type	The type of stator winding for the main phase. Click the button to open the **Winding Type** window and choose from **Lap**, **Sin_1**, **Sin_2**,and **Editor**.
Main (A)	**End Extension**	The end length adjustment of the stator coils
	Conductors per Layer	The number of conductors per layer in the main winding
	Parallel Branches	The number of parallel branches in the main stator winding
	Number of Strands	The number of wires per conductor (0 for auto-design)
	Wire Wrap	The thickness of the double-sided wire wrap (0 for auto-pickup from the wire library).
	Wire Size	The wire diameter (0 for auto-design).
Aux (B)	**End Extension**	The end length adjustment of the auxiliary winding
	Conductors per Layer	The number of conductors per layer in the auxiliary winding
	Parallel Branches	The number of parallel branches in the auxiliary stator winding
	Number of Strands	The number of wires per conductor in the auxiliary winding (0 for auto-design)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Wire Wrap	The thickness of the double-sided wire wrap (0 for auto-pickup from the wire library)
Wire Size	The wire diameter (0 for auto-design)

Winding Editor for a Single-Phase Induction Motor

For a single-phase induction motor, you may want to specify a different number of conductors for each stator slot. The Winding Editor makes this possible by enabling you to specify the number of turns for each coil.

To specify the number of turns for each coil:

1. Click **Machine** > **Winding** > **Edit Layout**.

   The **Winding Editor** window appears.

2. In the table in the upper left, set which phase you want for each coil and which slot is the “in” and “out” slot for the current in each coil.
3. If you are working on a quarter or half model, you may want to specify a multiplier by clicking the **Periodic Multiplier** check box and specifying a value.
4. Select or deselect the **Constant Turns** or **Constant Pitch** check boxes, depending on whether you want to be able to change these setting in the table above. When these options are selected, you cannot change the turns or pitch.
5. When you are satisfied with the coil settings, click **OK** to close the **Winding Editor** window.

Defining Different Size Wires for a Single-Phase Induction Motor

Use the **Gauge** option if you have a conductor that is made up of different size wires.

To define different size wires:

1. In the **Wire Size** window, select **MIXED** from the **Gauge** drop-down menu.
2. Select either **Round** or **Rectangular** as the **Wire Type**.
3. Enter the appropriate wire data in the table:
   - For a round wire:
     - Enter the **Diameter** in the table.
     - Enter a **Number** in the table to specify how many of the conductor’s wires have this diameter.
   - For a rectangular wire:
     - Enter the **Width** of the wire in the table.
     - Enter the **Thickness** of the wire in the table.
     - Enter the **Fillet** value in the table.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Enter a **Number** in the table to specify how many of the conductor's wires have this data.

4. Choose **Add** to add the new wire data.
5. Repeat steps 3 and 4 for each size wire you want to add.
6. When you are finished defining the wires, click **OK** to close the **Wire Size** window.

> **Note:** For example, if one conductor is made up of five wires, and three of those wires have a diameter of 0.21 mm, and the other 2 have a diameter of 0.13 mm, then the mixed wire size table will have two lines. The first line will list Diameter = 0.21 and Number = 3. The second line will list Diameter = 0.13 and Number = 2. An equivalent wire diameter is displayed as Wire Size value in the Winding tab in the Properties window.

Defining the Rotor Data for a Single-Phase Induction Motor

The rotor consists of copper bars in which current is induced from the stator windings. The rotor rotates at a slightly slower speed than the stator electromagnetic field. In the project tree, double-click **Machine** > **Rotor**, **Machine** > **Rotor** > **Slot**, and **Machine** > **Rotor** > **Winding** to define the physical dimensions, slot data, wires, and conductors for the rotor.

To define the general rotor data:

1. To open the **Rotor Properties** window, double-click the **Machine** > **Rotor** entry in the Project Manager tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the stacking factor for the rotor core in the **Stacking Factor** field.
3. Enter the **Number of Slots** in the rotor.
4. Select the **Slot Type**:
    a. Click the button for the **Slot Type**.
    b. In the **Select Slot Type** window that appears, select a slot type (available types include 1 through 4).

    > **Note:** When you place the mouse cursor over the slot type, a schematic of the selected type appears, displaying the slot dimension variables.

    c. Click **OK** to close the **Select Slot Type** window and return to the **Properties** window.
5. Enter the outer diameter of the rotor in the **Outer Diameter** field.
6. Enter the inner diameter of the rotor in the **Inner Diameter** field.
7. Enter the length of the rotor core in the **Length** field.
8. Select a **Steel Type** for the rotor core:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Click the button for **Steel Type**.
b. In the **Select Definition** window that appears, select a steel type from the list, or define a new steel type.
c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.

9. Enter the **Skew Width**, measured in rotor slot pitch. This value defines by how much the rotor bars are skewed.
10. Optionally, select **Cast Rotor** to allow the conductor to fill all the space available in the slot. Otherwise, RMxprt assumes the slot wedge that fixes the bars is filled with insulator material in a 2D/3D geometry model.
11. Click **OK** to close the **Properties** window.

Rotor Data for Single-Phase Induction Motors

To access the general rotor data, double-click the **Machine** > **Rotor** entry in the Project Manager tree.

The **Rotor Data Properties** window contains the following fields:

Stacking Factor	The stacking factor of the rotor core
Number of Slots	The number of slots the rotor core contains
Slot Type	The type of slots in the rotor core. Click the button to open the **Select Slot Type** window.
Outer Diameter	The outer diameter of the rotor core
Inner Diameter	The inner diameter of the rotor core
Length	The length of the rotor core
Steel Type	The steel type of the rotor core. Click the button to open the **Select Definition** window.
Skew Width	The skew width measured in slot number
Cast Rotor	Select or clear this to specify whether the rotor squirrel-cage winding is cast or not

Defining the Rotor Slots for Single-Phase Induction Motors

To define the rotor's slots:

1. To open the **Rotor Slot Properties** window, double-click the **Machine** > **Rotor** > **Slot** entry in the Project Manager tree on the desktop. (You can also enter values in the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Properties section of the desktop without opening a separate window.)

2. Enter the slot dimensions in the following fields: **Hs0**, **Hs01**, **Hs2**, **Bs0**, **Bs1**, **Bs2, Rs**.
3. Click **OK** to close the **Properties** window.

Rotor Slot Data for Single-Phase Induction Motors

To access the rotor slot data, double-click the **Machine** > **Rotor** > **Slot** entry in the Project Manager tree.

The **Rotor Slot Data Properties** window contains the following fields:

Hs0 A slot dimension (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected)

Hs01 A slot dimension (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected)

Hs1 A slot dimension (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected)

Hs2 A slot dimension (see the diagram shown in the modeling window when **Machine-Rotor-Slot** is selected)

Bs0 A slot dimension (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected)

Bs1 A slot dimension (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected)

Bs2 A slot dimension (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected)

Rs A slot dimension. (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected). Rs is added when the slot type is 3 or 4.

Defining the Rotor Windings for Single-Phase Induction Motors

To define the rotor windings:

1. To open the **Rotor Winding Properties** window, double-click the **Machine** > **Rotor** > **Winding** entry in the Project Manager tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Select a **Bar Conductor Type** for the rotor winding bar:
    a. Click the button for **Bar Conductor Type**.
    b. In the **Select Definition** window that appears, select a conductor type from the list, or define a new conductor type.
    c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Enter the length of the gap between the end ring and the iron core in the **End Length** field. This field specifies the value for only one end of the gap, not both.
4. Enter the end ring dimension in the axial direction in the **End-Ring Width** field. The end ring connects the bars of the rotor to one another.
5. Enter the end ring dimension in the radius direction in the **End-Ring Height** field. The end ring's height covers at least the cross-section of the rotor conductor.
6. Select an **End Ring Conductor Type** for the rotor winding end ring:
    a. Click the button for **End Ring Conductor Type**.
    b. In the **Select Definition** window that appears, select a conductor type from the list, or define a new conductor type.
    c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.
7. Click **OK** to close the **Properties** window.

Rotor Winding Data for Single-Phase Induction Motors

To access the rotor winding data, double-click the **Machine** > **Rotor** > **Winding** entry in the project tree.

The **Rotor Winding Data Properties** window contains the following fields:

Bar Conductor Type	The type of bar conductor used in the winding. Click the button to open the **Select Definition** window.
End Length	The length of the single-side end of the extended bar
End Ring Width	The width of one side of the end rings in the axial direction
End Ring Height	The height of the end rings in the radian direction
End Ring Conductor Type	The type of end ring conductor used in the winding. Click the button to open the **Select Definition** window.

Defining the Shaft Data for a Single-Phase Induction Motor

To define the shaft:

1. To open the **Shaft Data Properties** window, double-click the **Machine** > **Shaft** entry in the Project Manager tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Select or clear the **Magnetic Shaft** check box to specify whether or not the shaft is to be made of magnetic material.
3. Click **OK** to close the **Properties** window.

Shaft Data for Single-Phase Induction Motors

To access the shaft data, double-click the **Machine** > **Shaft** entry in the Project Manager tree.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Shaft Data Properties** window contains the following fields:

Magnetic Shaft	Select or clear this check box to indicate whether or not the shaft is made of magnetic material.

Adding or Removing a Vent from a Single-Phase Induction Motor

By option, you can add a vent to a single-phase induction motor. To add a vent:

1. Select the rotor icon in the Project Manager tree.
2. Right-click to select **Insert Vent**.

The vent icon appears under the rotor.

To remove a vent from a rotor in a three-phase induction motor.

1. Select the rotor icon in the Project Manager tree.
2. Right-click to select **Remove Vent**.

The vent icon disappears under the rotor.

The Vent data for the stator includes the following fields.

Holes per row	Number of axial vent holes per row.
Inner hole diameter	Diameter of vent holes in inner row.
Outer hole diameter	Diameter of vent holes in outer row.
Inner hole location	Center to center diameter of inner hole vents
Outer hole location	Center to center diameter of outer hole vents.

Setting Up Analysis Parameters for a Single-Phase Induction Motor

To define the solution data:

1. To open the **Solution Setup** window, right-click **Analysis** in the project tree, and click **Add Solution Setup**.
2. Click the **General** tab. The **Operation Type** is automatically set to **Motor** for this machine type.
3. Select the **Load Type** used in the motor from the following options:

Const Speed	The speed remains constant in the motor.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Const Power	The output power remains constant in the motor.
Const Torque	The torque remains constant regardless of the speed. In this case, $T_{load} = T_{rated}$, given by the output power divided by the given rated speed.
Linear Torque	The torque increases linearly with speed. In this case, $T_{load} = T_{rated} * (n/n_{rated})$ where T_{rated} is given by the output power divided by the given rated speed.
Fan Load	The load varies nonlinearly with speed. In this case, $T_{load} = T_{rated} * (n/n_{rated})^2$ where T_{rated} is given by the output power divided by the given rated speed.

4. Enter the output power developed at the shaft of the motor in the **Rated Output Power** field.
5. Enter the RMS line-to-line voltage in the **Rated Voltage** field.
6. Enter the desired output speed of the motor at the load point in the **Rated Speed** field.
7. Enter the temperature at which the system functions in the **Operating Temperature** field.
8. Click the **Single-Phase Induction Motor** tab.
9. Enter the electrical line frequency in the **Frequency** field, and select the units.
10. Click **OK** to close the **Solution Setup** window.

Solution Data for Single-Phase Induction Motors

To access the solution data, right-click **Analysis** in the project tree, and click **Add Solution Setup**.

The **Solution Setup** window contains the following fields:

Operation Type	On the **General** tab. The operation type is automatically set to **Motor** for this machine type.
Load Type	On the **General** tab. Select from **Const Speed**, **Const Power**, **Const Torque**, **Linear Torque**, and **Fan Load**. The default is **Const Power**.
Rated Output Power	On the **General** tab. Type a value for the rated output voltage, and select the units.
Rated Voltage	On the **General** tab. Type a value for the rated voltage, and select the units.
Rated Speed	On the **General** tab. Type a value for the rated speed, and select the units.
Operating Temperature	On the **General** tab. Type a value for the operating temperature, and select the units.
Frequency	On the **Single-Phase Induction Motor** tab. Type a value for the frequency, and select the units.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adjust-Speed Synchronous Machines

After you have selected **Adjust-Speed Synchronous Machines** as your model type, you need to define the following:

- General data, such as the voltage, speed, and circuit type of the model.
- Circuit data, such as trigger pulse width, transistor drop, and control circuit information.
- Stator data, such as the diameter, slot dimensions, and skew width of the stator.
- Stator Winding
- Rotor pole data, such as the associated permanent-magnet dimensions, air gap, and stacking factor.
- Solution data, such as rated output voltage and frequency.

By option, you can add a machine housing.

Analysis Approach for Adjust-Speed Synchronous Machines

In adjustable-speed permanent-magnet synchronous machines, the rotor speed is controlled by adjusting the frequency of the input voltage. Unlike standard brushless permanent-magnet DC motors, this type of machine does not utilize position sensors.

Permanent magnets are mounted on the rotor of a permanent-magnet synchronous machine, which could be either inner or outer rotor type. The polyphase armature winding is embedded in the stator, whose number of poles is the same as that of the rotor. The machine can operate as a generator or as a motor. When the machine operates as a motor, the stator poly-phase winding can be fed either by a sinusoidal AC source or by a DC source via a DC to AC inverter. When the machine operates as a generator, the stator polyphase winding supplies an AC source for electric loads.

Stator Winding Connected to a Sinusoidal AC Source

In this case, the performance of the machine can be analyzed in the frequency domain based on the phasor diagrams, as shown in Figure 6.1 for the generators and Figure 6.2 for the motors.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

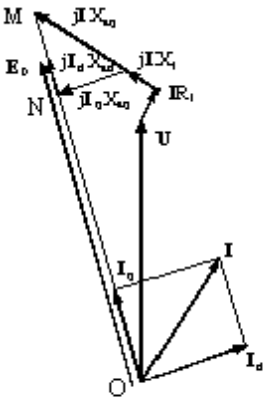


Figure 6.1 The phasor diagram for generators

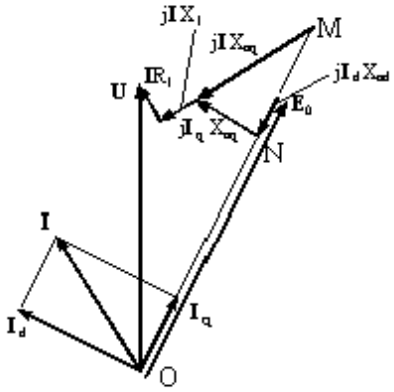


Figure 6.2 The phasor diagram for motors

In the figures, R_1 and X_1 are the resistance and the leakage reactance of the armature winding, X_{ad} and X_{aq} are the d-axis armature reactance and the q-axis armature reactance, respectively. In the phasor diagram, X_{ad} is a linearized nonlinear parameter, and X_{aq} is a linear parameter. The d-axis synchronous reactance X_d and q-axis synchronous reactance X_{aq} are calculated directly from

Let θ denote the power angle for a generator (the angle that U lags E_0), or the torque angle for a motor (the angle that E_0 lags U), then we have

$$X_d = X_1 + X_{ad}$$

$$X_q = X_1 + X_{aq}$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$I_d X_d + I_q R_1 = \pm(U cos\theta - E_0)$$

$$-I_d R_1 + I_q X_q = U sin\theta$$

where the plus sign is for the motor and the minus sign is for the generator.

Solving for I_d and I_q yields

$$I_d = \frac{\pm X_q(U cos\theta - E_0) - R_1 U sin\theta}{R_1^2 + X_d X_q}$$

$$I_q = \frac{\pm R_1(U cos\theta - E_0) - X_d U sin\theta}{R_1^2 + X_d X_q}$$

where the plus sign is for the motor and the minus sign is for the generator.

Let the angle that *I* lags E_0 be ψ, we have

$$\psi = atan\frac{I_d}{I_q}$$

The power factor angle φ (the angle that *I* lags *U*) is

$$\varphi = \psi \pm \theta$$

where the plus sign is for the motor and the minus sign is for the generator.

For the motor operation, the input electric power is

$$P_1 = 3UIcos\varphi$$

The output mechanical power is

$$P_2 = P_1 - (P_{fw} + P_{Cua} + P_{Fe})$$

where P_{fw}, P_{Cua}, and P_{Fe} denote the frictional and wind, the armature copper and the iron-core losses, respectively.

The output mechanical torque is

$$T_2 = \frac{P_2}{\omega}$$

where ω denotes the synchronous speed in rad/s.

The efficiency of the motor is

$$\eta = \frac{P_2}{P_1} \times 100\%$$

For the generator operation, the output electric power is

$$P_2 = 3UIcos\varphi$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The input mechanical power is

$$P_1 = P_2 + P_{fw} + P_{Cua} + P_{Fe}$$

where P_{fw}, P_{Cua}, and P_{Fe} denote the frictional and wind, the armature copper and the iron-core losses, respectively.

The input mechanical torque is

$$T_1 = \frac{P_1}{\omega}$$

where ω denotes the synchronous speed in mechanical rad/s.

The efficiency of the generator is

$$\eta = \frac{P_2}{P_1} \times 100\%$$

Stator Winding Fed by a DC to AC Inverter

In this case, this adjustable-speed synchronous machine (ASSM) operates as a motor, and the analysis approach is similar to that of a brushless DC (BLDC) motor. The stator poly-phase armature winding is connected to a DC power supply through a DC to AC inverter to produce the rotational magnetic field in the air-gap. The main difference between ASSM and BLDC motor is: in BLDC motor, trigger time exactly **depends on** the rotor position; but in ASSM, the trigger time is **independent of** the rotor position. If the mechanical load of a BLDC motor increases, the rotor speed and the induced voltage decreases, causing the armature current and torque increase to balance the increased mechanical load. However, for an ASSM, if the mechanical load increases, the rotor speed decreases temporarily, which causes the torque angle (the same as lead angle of trigger for a BLDC motor) increase and then torque increase to retain the synchronous speed.

Therefore, the speed of a BLDC motor varies with input voltage and mechanical load, while the speed of an ASSM does not. The speed of an ASSM can be changed by adjusting the frequency of the controlling signal, which explains why it is called *Adjustable-Speed Permanent-Magnet Synchronous Machine*.

Using the time-domain mathematical model to analyze the characteristics of the electric machine, Park's voltage equation in the matrix form is as follows

$$\begin{bmatrix} v_d \\ v_q \\ v_0 \end{bmatrix} - \begin{bmatrix} e_d \\ e_q \\ e_0 \end{bmatrix} = \begin{bmatrix} R_1 + L_d p & -L_q \omega_e & 0 \\ -L_d \omega_e & R_1 + L_q p & 0 \\ 0 & 0 & R_1 + L_0 p \end{bmatrix} \cdot \begin{bmatrix} i_d \\ i_q \\ i_0 \end{bmatrix}$$

where R_1 is the armature winding resistance, L_d, L_q and L_0 are the d-, the q- and the 0-axis inductances respectively, ω_e is the revolution speed in electric radians per second, the differential operator is

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$p = \frac{d}{dt}$$

The coordinate transformation equations for the terminal voltage, the induced emf and the armature winding current are

$$\begin{bmatrix} v_d \\ v_q \\ v_0 \end{bmatrix} = C^T \begin{bmatrix} v_a \\ v_b \\ ? \end{bmatrix} \qquad \begin{bmatrix} e_d \\ e_q \\ e_0 \end{bmatrix} = C^T \begin{bmatrix} e_a \\ e_b \\ ? \end{bmatrix} \qquad \begin{bmatrix} i_a \\ i_b \\ ? \end{bmatrix} = C \begin{bmatrix} i_d \\ i_q \\ i_0 \end{bmatrix}$$

The transformation matrices for the two-, the three- and the four-phase systems are C_2, C_3 and C_4, respectively, as follows:

$$C_2 = \begin{bmatrix} \cos\theta & \sin\theta & 0 \\ \sin\theta & \cos\theta & 0 \end{bmatrix}$$

$$C_3 = \sqrt{\frac{2}{3}} \begin{bmatrix} \cos\theta & \sin\theta & \frac{1}{\sqrt{2}} \\ \cos(\theta - \alpha) & \sin(\theta - \alpha) & \frac{1}{\sqrt{2}} \\ \cos(\theta - 2\alpha) & \sin(\theta - 2\alpha) & \frac{1}{\sqrt{2}} \end{bmatrix}$$

$$C_4 = \begin{bmatrix} \cos\theta & \sin\theta & 0 \\ \sin\theta & -\cos\theta & 0 \\ -\cos\theta & -\sin\theta & 0 \\ -\sin\theta & \cos\theta & 0 \end{bmatrix}$$

where

$$\alpha = \frac{2}{3}\pi$$

The input electric power is obtained from the voltage and the current as

$$P_1 = \frac{1}{T}\int_0^T (v_d i_d + v_q i_q + v_0 i_0) dt$$

The output mechanical power is

$$P_2 = P_1 - (P_{fw} + P_{Cua} + P_{Fe})$$

where P_{fw}, P_{Cua}, P_t and P_{Fe} denote the frictional and wind, the armature copper, the switching and the iron-core losses, respectively.

The output mechanical torque is

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$T_2 = \frac{P_2}{\omega}$$

where ω denotes the revolution speed in mechanical radians per second.

The efficiency of the electric machine is

$$\eta = \frac{P_2}{P_1} \times 100\%$$

Defining an Adjustable-Speed Synchronous Machine

The general procedure for defining a adjust-speed synchronous machine is as follows:

1. Insert the adjust-speed synchronous machine into a new or existing project.
2. Double-click the **Machine** entry in the Project Manager tree to define the general data.
3. Double-click the **Machine** > **Circuit** entry in the Project Manager tree to define the control circuit.
4. Double-click the **Machine** > **Stator** entry in the Project Manager tree to define the stator geometry.
5. Double-click the **Machine** > **Stator** > **Slot** entry in the Project Manager tree to define the stator slot dimensions.
6. Double-click the **Machine** > **Stator** > **Winding** entry in the Project Manager tree to define the stator windings and conductors.
7. Double-click the **Machine** > **Rotor** entry in the Project Manager tree to define the rotor geometry.
8. Double-click the **Machine** > **Rotor** > **Pole** entry in the Project Manager tree to define the pole, embrace, offset, and air gap data for the rotor pole.
9. Double-click the **Machine** > **Shaft** entry in the Project Manager tree to define the magnetism of the shaft.
10. Optionally, right-click the **Machine** entry in the Project Manager tree and select **Insert Housing** to add a machine housing.
11. Right-click **Analysis** in the project tree, and click **Add Solution Setup** to define the solution data.
12. Choose **File** > **Save** to save the project.
13. Choose **RMxprt** > **Analyze** to analyze the design.

> **Note:** When you place the cursor over an entry field in the data windows, a brief description of that field appears in the status bar at the bottom of the **RMxprt** window.

Once analyzed, the model can be viewed in the Maxwell 2D Modeler, or it can be used to create a new Maxwell 2D project, and a new Maxwell 3D design.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining the General Data for an Adjust-Speed Synchronous Machine

Use the **General** window to define the basic parameters of the motor, such as the motor's rated output power, rated voltage, losses, and circuit types.

To define the general data:

1. To open the **General Data Properties** window, double-click the **Machine** entry in the Project Manager tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the number of poles for the machine in the **Number of Poles** field. This value is the total number of poles in the stator (or the number of pole pairs multiplied by two).
3. Enter the energy loss due to friction at the given speed in the **Frictional Loss** field.
4. Enter the wind loss due to air resistance measured at the reference speed in the **Windage Loss** field.
5. Enter the given speed in the **Reference Speed** field.
6. Select one of the following from the **Control Type** drop-down menu:
   - **DC**: Switched DC voltage at the given input frequency.
   - **PWM**: Pulse width modulation. When you select this source type, you must enter the following values in the **Circuit Data Properties** window: **Modulation Index** (the ratio of the sine wave amplitude to the triangular amplitude) and **Carrier Frequency Times** (the ratio of the triangular frequency to the sine wave frequency).
   - **AC**: An AC excitation.
7. Select a **Circuit Type** from the following types:

Y3	Y-connected, three-phase
L3	Loop-type, three-phase
S3	Star-type, three-phase
C2	Cross-type, two-phase
L4	Loop-type, four-phase
S4	Star-type, four-phase

   The circuit types are based on industry standards. By default, type **Y3**, a three-phase, six-status circuit, is selected as the circuit type.

   > **Note:** When you place the mouse cursor over a circuit type, an outline schematic of the circuit appears.

8. Click **OK** to close the **Properties** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Data for Adjust-Speed Synchronous Machines

To access the general data, double-click the **Machine** entry in the Project Manager tree.

The **General Data Properties** window for a three-phase induction motor contains the following fields:

Machine Type	The machine type you selected when inserting a new RMxprt design (Adjust-Speed Synchronous Machine)
Number of Poles	The number of poles the machine contains
Rotor Position	Select whether the rotor is an **Inner Rotor** or **Outer Rotor**
Frictional Loss	The frictional energy loss (due to friction) measured at the reference speed
Windage Loss	The windage loss (due to air resistance) measured at the reference speed
Reference Speed	The given speed of reference
Control Type	The way the circuit is controlled. Select from **DC**, **PWM** (pulse-width modulation), or **AC**.
Circuit Type	The drive circuit type. Click the button to open the **Circuit Type** window and select from the following six types: • **Y3**: Y-Type, 3-Phase • **L3**: Loop-Type, 3-Phase • **S3**: Star-Type, 3-Phase • **C2**: Cross-Type, 2-Phase • **L4**: Loop-Type, 4-Phase • **S4**: Star-Type, 4-Phase

Defining the Circuit Data for an Adjust-Speed Synchronous Machine

Use the **Circuit Data Properties** window to define the circuit properties for an adjustable-speed synchronous machine.

> **Note:** No circuit data properties exist when **AC** is selected as the **Control Type**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. To open the **Circuit Data Properties** window, double-click the **Machine** > **Circuit** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. If you selected DC as the **Control Type**, enter the period from on-status to off-status of a transistor, in electrical degrees, in the **Trigger Pulse Width** field.
3. Enter the voltage drop across one transistor when the transistor is turned on in the **Transistor Drop** field. Refer to the figures of the different circuit types in step 2.
4. Enter the voltage drop of one diode in the discharge loop in the **Diode Drop** field. If you selected a star-type circuit (**S3** or **S4**) as the **Circuit Type**, enter the total discharge voltage in this field.
5. If you selected PWM as the **Control Type**, then enter values in the following two fields:
   - **Modulation Index**: The ratio of the sine-wave amplitude to the triangular amplitude. (PWM circuits only.)
   - **Carrier Frequency Times**: The ratio of the triangular frequency to the sine-wave frequency. (PWM circuits only.)
6. Click **OK** to close the **Properties** window.

Circuit Data for Adjust-Speed Synchronous Machines

To access the **Circuit Data Properties** window, double-click the **Machine** > **Circuit** entry in the project tree. No circuit data properties exist when **AC** is selected as the **Control Type**.

Trigger Pulse Width	The period from on-status to off-status for a transistor, in electrical degrees. (DC circuits only.)
Transistor Drop	The voltage drop across one transistor when the transistor is turned on.
Diode Drop	The voltage drop across one diode in the discharge loop.
Modulation Index	The ratio of the sine-wave amplitude to the triangular amplitude. (PWM circuits only.)
Carrier Frequency Times	The ratio of the triangular frequency to the sine-wave frequency. (PWM circuits only.)

Defining the Stator Data for an Adjust-Speed Synchronous Machine

Use the **Stator Properties** windows to define the stator dimensions, slots, windings, and conductors.

The stator is the outer lamination stack where the polyphase voltage windings reside.

To define the general stator data:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. To open the **Stator Data Properties** window, double-click the **Machine** > **Stator** entry in the Project Manager tree. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the **Outer Diameter** of the stator.
3. Enter the **Inner Diameter** of the stator.
4. Enter the length of the stator core in the **Length** field.
5. Enter the stacking factor for the stator core in the **Stacking Factor** field.
6. Select a **Steel Type** for the stator core:
    a. Click the button for **Steel Type**.
    The **Select Definition** window appears.
    b. Select a steel type from the list, or define a new steel type.
    c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.
7. Enter the **Number of Slots** in the stator.
8. Select the **Slot Type**:
    a. Click the button for the **Slot Type**.
    The **Select Slot Type** window appears.
    b. Select a slot type (available types include 1 through 4).

    > **Note:** When you place the mouse cursor over the slot type, a schematic of the selected type appears, displaying the slot dimension variables.

    c. Click **OK** to close the **Select Slot Type** window and return to the **Properties** window.
9. Enter the skew width, measured in slot number, in the **Skew Width** field.
10. Click **OK** to close the **Properties** window.

Defining the Stator Dimensions and Slots

To define the stator slots:

1. To open the **Stator Slot Data Properties** window, double-click the **Machine** > **Stator** > **Slot** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Optionally, to automatically design the dimensions of slots **Hs2**, **Bs1**, and **Bs2**, select the **Auto Design** check box.
3. Optionally, to design dimensions of slots **Bs1** and **Bs2** based on the stator tooth width, select the **Parallel Tooth** check box, and enter a value in the **Tooth Width** field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Enter the available slot dimensions.

   Hs0 Always available

   Hs2 Available only when **Auto Design** is cleared. When **Auto Design** is selected, this slot dimension is determined automatically.

   Bs0 Always available

   Bs1 Available only when **Auto Design** and **Parallel Tooth** are both cleared. When **Auto Design** is selected, this slot dimension is determined automatically. When **Parallel Tooth** is selected, this slot dimension is determined based on the value entered in the **Tooth Width** field.

   Bs2 Available only when **Auto Design** and **Parallel Tooth** are both cleared. When **Auto Design** is selected, this slot dimension is determined automatically. When **Parallel Tooth** is selected, this slot dimension is determined based on the value entered in the **Tooth Width** field.

   Rs Rs is added when the slot type is 3 or 4.

5. Click **OK** to close the **Properties** window.

Stator Data for Adjust-Speed Synchronous Machines

To access the general stator data, double-click the **Machine** > **Stator** entry in the Project Manager tree.

The **Stator Data Properties** window contains the following fields:

Outer Diameter	The outer diameter of the stator core
Inner Diameter	The inner diameter of the stator core
Length	The length of the stator core
Stacking Factor	The stacking factor of the stator core
Steel Type	The steel type of the stator core. Click the button to open the **Select Definition** window.
Number of Slots	The number of slots the stator core contains
Slot Type	The type of slots in the stator core. Click the button to open the **Select Slot Type** window.
Skew Width	The skew width measured in slot number

Stator Slot Data for Adjust-Speed Synchronous Machines

To access the stator slot data, double-click the **Machine** > **Stator** > **Slot** entry in the Project Manager tree.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Stator Slot Data Properties** window contains the following fields:

Auto Design	Select or clear this to enable or disable auto-design of slots **Hs2**, **Bs1**, and **Bs2**. When this check box is selected, only two other fields appear in the window: **Hs0** and **Bs0**.
Parallel Tooth	Select this to design **Bs1** and **Bs2** based on the tooth width. When this check box is selected, the **Bs1** and **Bs2** fields are removed, and the **Tooth Width** field is added.
Tooth Width	The tooth width for the parallel tooth, on which **Bs1** and **Bs2** are designed.
Hs0	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected)
Hs1	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected)
Hs2	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected)
Bs0	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected)
Bs1	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected)
Bs2	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected)
Rs	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected); Rs is added when the slot type is 3 or 4.

Defining the Stator Windings and Conductors for an Adjust-Speed Synchronous Machine

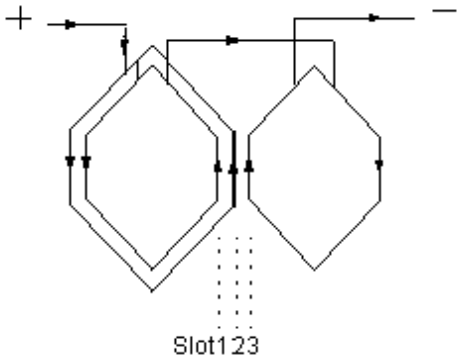
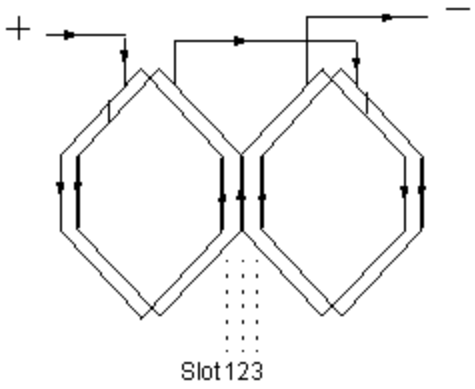
To define the stator windings and conductors:

1. To open the **Stator Slot Winding Properties** window, double-click the **Machine-Stator-Winding** entry in the project tree on the desktop. You can also enter values in the **Properties** section of the desktop without opening a separate window.
2. Click the **Winding** tab.
3. Enter the number of layers in the stator winding in the **Winding Layers** field.
4. Select a **Winding Type**:
    a. Click the button for **Winding Type**.
    b. In the **Winding Type** window that appears, select from one of the following three types of winding:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Whole Coiled**
- **Half Coiled**
- **Editor**

When you place the mouse cursor over a winding button, an outline of the selected winding appears. The following table describes the six types of windings that are possible (three for one-layer and three for two-layer):

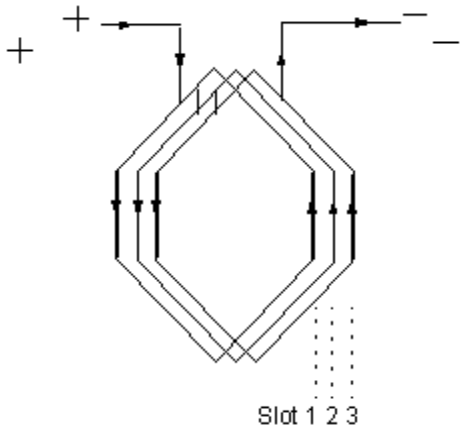
Type	Description
One Layer Winding Editor	A user-defined one-layer winding arrangement. You need to set up the winding arrangement for each slot.
Whole Coiled	A one-layer whole-coiled winding: 
Whole Coiled	A two-layer wave winding: 

The phase belt for this winding configuration is equal to 360/2*m*, where *m* is the phase number.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A two-layer half-coiled winding:

Half Coiled



There is only one coil per phase per pair of poles.

> **Note: For a two-layer winding**, if you click **Constant Pitch** in the **Winding Editor**, only the top layer needs to be defined; the bottom layer will be determined according to the coil pitch.

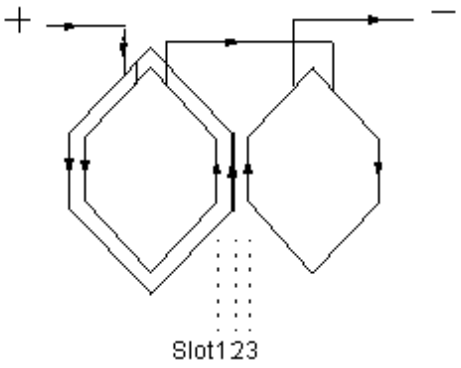
c. Once you have clicked a button to select a winding, click **OK** to close the **Winding Type** window and return to the **Properties** window. Select the **Winding Type** for the stator.

> **Note:** When you place the mouse cursor over the winding type, a schematic of the selected winding appears.

Winding types **10** and **20** are user-defined. If you select either of these, a window appears, asking you to define the name of the winding arrangement. The window closes when the user-defined winding is entered.
Select from the following winding types:

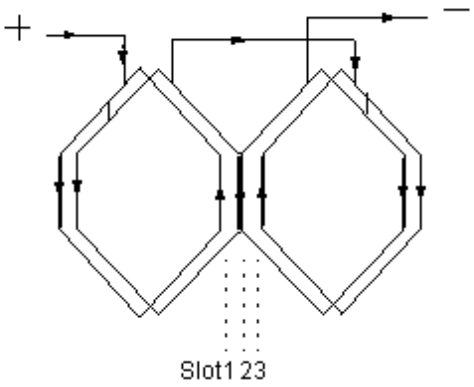
One-Layer Winding Editor	A user-defined single-layer winding arrangement. When you select this type, enter the winding arrangement, and choose **OK**.
11	A one-layer whole-coiled winding:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



20 A user-defined winding arrangement. When you select this type, enter the winding arrangement, and choose **OK**.

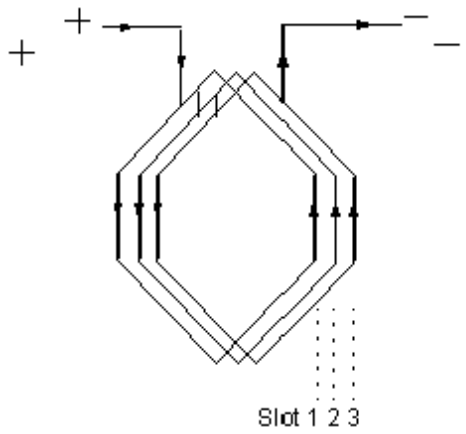
21 A two-layer wave winding:



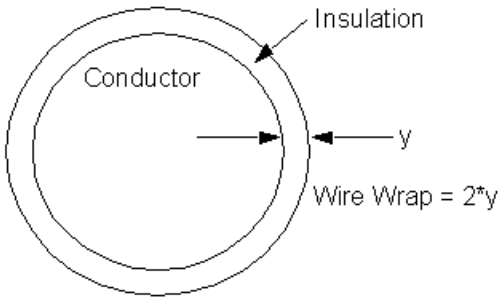
The phase belt for this winding configuration is equal to 360/2*m*, where *m* is the phase number.

22 A two-layer winding:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Select or enter the number of parallel branches in one phase of the winding in the **Parallel Branches** field.
6. Enter the total number of conductors in each stator slot in the **Conductors per Slot** field. This value is the number of turns per coil multiplied by the number of layers.
7. Enter the coil pitch, measured in number of slots, in the **Coil Pitch** field. The coil pitch is the number of slots separating one winding. For example, if a coil starts in slot 1 and ends in slot 6, it has a coil pitch of 5.
8. Enter the number of wires per conductor in the **Number of Strands** field. Enter 0 to have RMxprt auto-design this value.
9. Enter the thickness of the double-sided wire wrap in the **Wire Wrap** field. Enter 0 to automatically obtain this value from the wire library.



10. Select the **Wire Size**:
    a. Click the button for **Wire Size**.
    b. In the **Wire Size** window that appears, select a value from the **Wire Diameter** drop-down menu.
    c. Select a wire gauge from the **Gauge** drop-down menu. You can select from the following options:

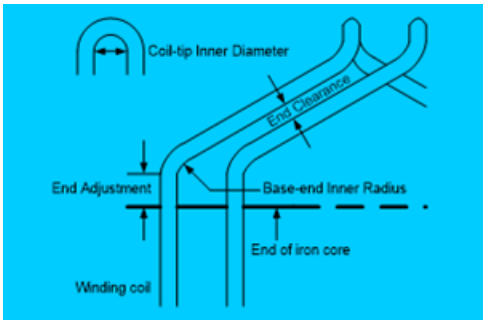
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

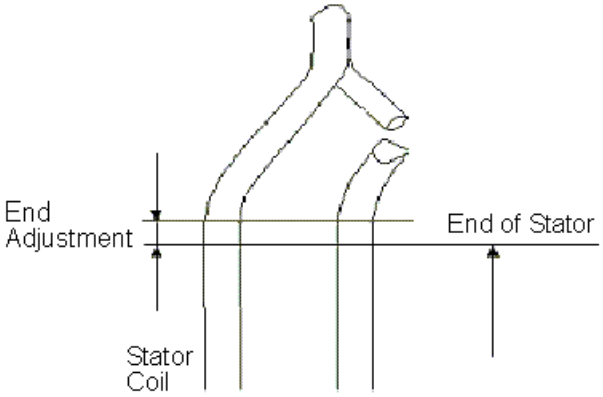
< ***number*** **>**	You can select a specific gauge number. When you select a gauge number, the **Wire Diameter** field is automatically updated.
USER	This option allows you to manually enter the **Wire Diameter**. This is useful when you want to enter a diameter that does not correspond to a particular wire gauge.
AUTO	This option sets the **Wire Diameter** to zero, and RMxprt automatically calculates the optimal value. The diameter information is then written to the output file when you analyze the design.
MIXED	This option allows you to define a conductor that is made of different size wires. For example, a single conductor may consist of five wires: 3 wires with a diameter of 0.21 mm and two wires with a diameter of 0.13 mm.

The gauge number is based on AWG settings. You can create your own wire table using **Machine** > **Wire**, and then you can select this wire table using the **Tools** > **Options** > **General Options** > **Machines** function. When you are done setting the wire size, click **OK** to close the **Wire Size** window and return to the **Properties** window.

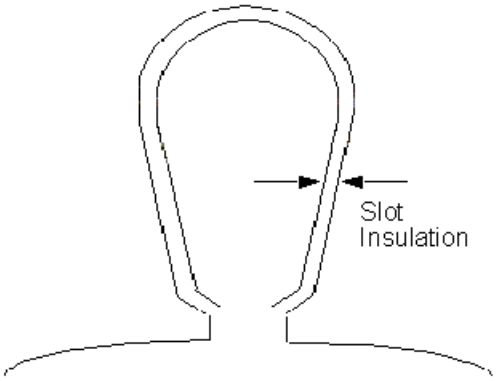
11. Click the **End/Insulation** tab.
12. Select or clear the **Input Half-turn Length** check box.
13. Do one of the following:
    - If you selected **Input Half-turn Length**, then enter the half-turn length of the armature winding in the Half Turn Length field.
    - If you cleared **Input Half-turn Length**, then enter the end length adjustment of the stator coils in the **End Adjustment** field. The end adjustment is the distance one end of the conductor extends vertically beyond the end of the stator.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





14. Enter the inner radius of the base corner in the **Base Inner Radius** field.
15. Enter the inner diameter of the coil tip in the **Tip Inner Diameter** field.
16. Enter the distance between two stator coils in the **End Clearance** field.
17. Enter the thickness of the slot liner insulation in the **Slot Liner** field.



18. Enter the thickness of the wedge insulation in the **Wedge Thickness** field.
19. Enter the thickness of the insulation layer in the **Layer Insulation** field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20. Enter the limited slot fill factor for the wire design in the **Limited Fill Factor** field. The slot fill factor is the ratio between the cross-sectional area of all conductors in one slot and the entire slot area.
21. Click **OK** to close the **Properties** window.

Winding Editor for an Adjustable-Speed Synchronous Machine

For a adjustable-speed synchronous machine, you may want to specify a different number of conductors for each stator slot. The Winding Editor makes this possible by enabling you to specify the number of turns for each coil.

To specify the number of turns for each coil:

1. Click **Machine** > **Winding** > **Edit Layout**.
   The **Winding Editor** window appears.
2. In the table in the upper left, set which phase you want for each coil and which slot is the “in” and “out” slot for the current in each coil.
3. If you are working on a quarter or half model, you may want to specify a multiplier by clicking the **Periodic Multiplier** check box and specifying a value.
4. Select or deselect the **Constant Turns** or **Constant Pitch** check boxes, depending on whether you want to be able to change these setting in the table above. When these options are selected, you cannot change the turns or pitch.
5. When you are satisfied with the coil settings, click **OK** to close the **Winding Editor** window.

Defining Different Size Wires for an Adjustable Speed Synchronous Machine

Use the **Gauge** option in the **Wire Size** dialog if you have a conductor that is made up different size wires.

To define different size wires:

1. In the **Wire Size** window, select **MIXED** from the **Gauge** drop-down menu.
2. Select either **Round** or **Rectangular** as the **Wire Type**.
3. Enter the appropriate wire data in the table:
   - For a round wire:
   - Enter the **Diameter** in the table.
   - Enter a **Number** in the table to specify how many of the conductor’s wires have this diameter.
   - For a rectangular wire:
   - Enter the **Width** of the wire in the table.
   - Enter the **Thickness** of the wire in the table.
   - Enter the **Fillet** value in the table.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Enter a **Number** in the table to specify how many of the conductor's wires have this data.

4. Click **Add** to add the new wire data.
5. Repeat steps 3 and 4 for each size wire you want to add.
6. When you are finished defining the wires, click **OK** to close the **Wire Size** window and return to the RMxprt **Properties** window.

> **Note:** For example, if one conductor is made up of five wires, three of those wires have a diameter of 0.21 mm, and the other two have a diameter of 0.13 mm, then the mixed wire size table will have two lines. The first line will list Diameter = 0.21 and Number = 3. The second line will list Diameter = 0.13 and Number = 2. An equivalent wire diameter is displayed as Wire Size value in the Winding tab in the Properties window.

Stator Winding Data for Adjust-Speed Synchronous Machines

To access the stator winding data, double-click the **Machine** > **Stator** > **Winding** entry in the Project Manager tree.

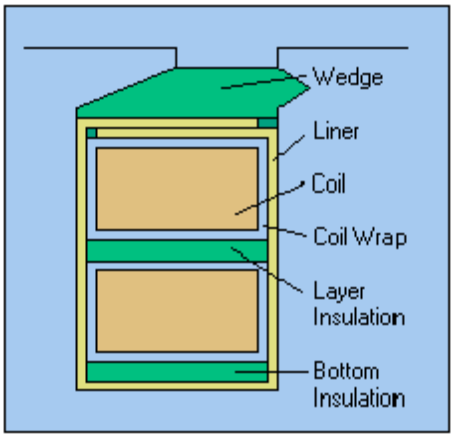
The **Stator Winding Data Properties** window contains the following fields:

Winding tab	**Winding Layers**	The number of winding layers.
	Winding Type	The type of stator winding. Click the button to open the **Winding Type** window and choose from **Whole Coiled**, **Half Coiled**, and **Editor**.
	Parallel Branches	The number of parallel branches in the stator winding.
	Conductors per Slot	The number of conductors per stator slot (0 for auto-design).
	Coil Pitch	The coil pitch measured in number of slots.
	Number of Strands	The number of wires per conductor (0 for auto-design).
	Wire Wrap	The thickness of the double-sided wire wrap (0 to automatically obtain this value from the wire library).
	Wire Size	The diameter of the wire (0 for auto-design). Click the button to open the **Wire Size** window where you can specify units, wire type, diameter, and gauge.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

End/Insulation tab	**Input Half-turn Length**	Select or clear this check box to specify whether or not you want to enter the half-turn length. When this check box is selected, the **Half Turn Length** field appears the next time you open the **Properties** window. When this check box is selected, the **End Adjustment** field appears instead.
	Half Turn Length	The half-turn length of the armature winding.
	End Adjustment	The end length adjustment of the stator coils, which is the distance one end of the conductor extends vertically beyond the end of the stator.
	Base Inner Radius	The inner radius of the base corner.
	Tip Inner Diameter	The inner diameter of the coil tip.
	End Clearance	The end clearance between two adjacent coils.
	Slot Liner	The thickness of the slot liner insulation.
	Wedge Thickness	The thickness of the wedge insulation.
	Layer Insulation	The thickness of the insulation layer.
	Limited Fill Factor	The limited slot fill factor for the wire design. The slot fill factor is the ratio between the cross-sectional area of all conductors in one slot and the entire slot area.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Defining the Rotor Data for an Adjust-Speed Synchronous Machine

The rotor consists of copper bars in which current is induced by the magnetic fields produced by the stator windings. In the project tree, double-click **Machine** > **Rotor** and **Machine** > **Rotor** > **Pole** to define the rotor and the pole.

To define general stator data:

1. To open the **Rotor Data Properties** window, double-click the **Machine** > **Rotor** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the outer diameter of the rotor in the **Outer Diameter** field.
3. Enter the inner diameter of the rotor in the **Inner Diameter** field.
4. Enter the length of the rotor core in the **Length** field.
5. Select a **Steel Type** for the rotor core:
    a. Click the button for **Steel Type**.
    The **Select Definition** window appears.
    b. Select a steel type from the list, or define a new steel type.
    c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.
6. Enter the stacking factor for the rotor core in the **Stacking Factor** field.
7. Select a **Pole Type**:
    a. Click the button.
    The **Select Pole Type** window appears.
    b. Click a button to select the desired pole type (1, 2, 3, 4, or 5). TIP: When you run the mouse over each option, the diagram changes to show that pole type.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** When you place the mouse cursor over a rotor type, an outline of the selected circuit type appears.

c. Click **OK** to close the **Select Pole Type** window and return to the **Properties** window.

8. Click **OK** to close the **Properties** window.

Rotor Data for Adjust-Speed Synchronous Machines

To access the general rotor data, double-click the **Machine** > **Rotor** entry in the Project Manager tree.

The **Rotor Data Properties** window contains the following fields:

Outer Diameter	The outer diameter of the rotor core
Inner Diameter	The inner diameter of the rotor core
Length	The length of the rotor core
Steel Type	The steel type of the rotor core; click the button to open the **Select Definition** window.
Stacking Factor	The stacking factor of the rotor core
Pole Type	The pole type for the rotor; click this button to open the **Select Pole Type** window and select from the following types: **1**, **2**, **3**, **4**, **5**.

Defining the Rotor Pole for an Adjust-Speed Synchronous Machine

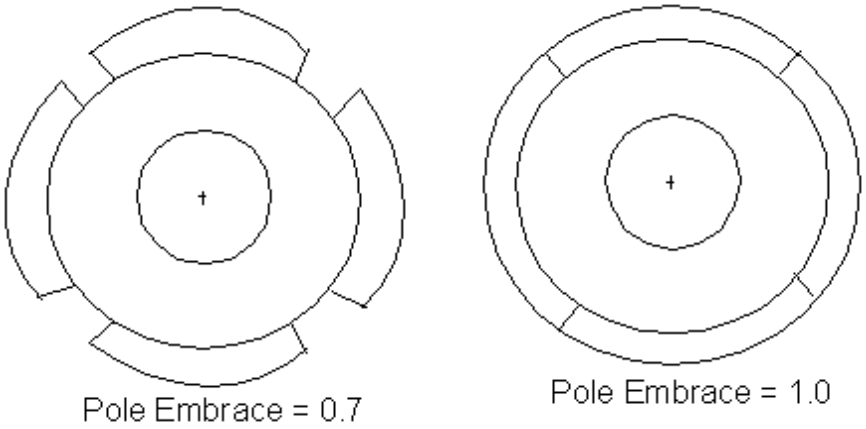
The rotor pole drives the electromagnetic field which is coupled with the stator windings. Use the **Rotor Pole Data Properties** window to define the rotor pole.

> **Note:** Some of the fields in the **Rotor Pole** window change or are inactive depending on the **Rotor Type** you select.

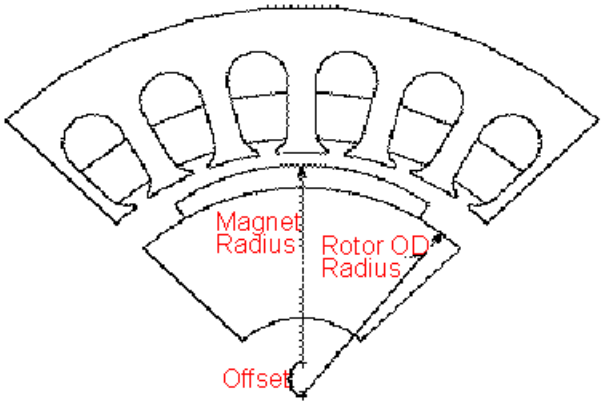
To define the rotor pole:

1. To open the **Rotor Pole Data Properties** window, double-click the **Machine** > **Rotor** > **Pole** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. For all pole types except type **4**, enter the ratio of the actual arc distance in relation to the maximum possible arc distance in the **Embrace** field. This value is between 0 and 1.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. For pole type **4**, enter the shaft diameter of the rotor in the **Shaft Diameter** field.
4. For pole types **1**, **2**, and **3**, enter the distance from the center of the rotor to the polar arc center in the **Offset** field. Enter 0 for a uniform air gap.



5. For pole type **5**, enter the thickness of the bridge across the two poles in the **Bridge** field.
6. For pole type **5**, enter the width of the rib supporting the bridge in the **Rib** field.
7. Select the type of magnet to use in the rotor pole from the **Magnet Type** drop-down menu.
8. For pole types **4** and **5**, enter the width of the magnet in the **Magnet Width** field.
9. Enter the maximum radial thickness of the magnet in the **Magnet Thickness** field.
10. Click **OK** to close the **Properties** window.

Rotor Pole Data for Adjust-Speed Synchronous Machines

To access the pole rotor data, double-click the **Machine** > **Rotor** > **Pole** entry in the Project Manager tree.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Rotor Pole Data Properties** window contains the following fields:

Embrace	The pole embrace. For pole types **1**, **2**, **3**, and **5**.
Shaft Diameter	The shaft diameter of the rotor. For pole type **4**.
Offset	The pole-arc center offset from the rotor center (0 for a uniform air gap). For pole types **1**, **2**, and **3**.
Bridge	The thickness of the bridge across two adjacent poles. For pole type **5**.
Rib	The width of the rib at the center of two adjacent poles that support the bridge. For pole type **5**.
Magnet Type	The type of magnet. Click the button to open the **Select Definition** window. For all pole types.
Magnet Width	The maximum width of the magnet. For pole types **4** and **5**.
Magnet Thickness	The maximum thickness of the magnet. For all pole types.

Defining the Shaft Data for an Adjust-Speed Synchronous Machine

To define the shaft:

1. To open the **Shaft Data Properties** window, double-click the **Machine** > **Shaft** entry in the Project Manager tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Select or clear the **Magnetic Shaft** check box to specify whether or not the shaft is to be made of magnetic material.
3. Click **OK** to close the **Properties** window.

Shaft Data for Adjust-Speed Synchronous Machines

To access the shaft data, double-click the **Machine** > **Shaft** entry in the Project Manager tree.

The **Shaft Data Properties** window contains the following fields:

Magnetic Shaft	Select or clear this check box to indicate whether or not the shaft is made of magnetic material.

Setting Up Analysis Parameters for an Adjust-Speed Synchronous Machine

To define the solution data:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Right-click **Analysis** in the project tree, and click **Add Solution Setup** to open the Solution Setup window.
2. Click the **General** tab. The **Operation Type** is automatically set to **Motor** for this machine type.

> **Note:** To enable selection of **Generator** for Adjust-Speed Synchronous Machines, the machine **Control Type** must be set to **AC** in its Properties window.

   a. If Motor was selected for the Operation Type, select the **Load Type** used in the motor from the following options:

Const Speed	The speed remains constant in the motor.
Const Power	The output power remains constant in the motor.
Const Torque	The torque remains constant regardless of the speed. In this case, $T_{load} = T_{rated}$, given by the output power divided by the given rated speed.
Linear Torque	The torque increases linearly with speed. In this case, $T_{load} = T_{rated} * (n/n_{rated})$ where T_{rated} is given by the output power divided by the given rated speed.
Fan Load	The load varies nonlinearly with speed. In this case, $T_{load} = T_{rated} * (n/n_{rated})^2$ where T_{rated} is given by the output power divided by the given rated speed.

   b. If Generator was selected for the Operation Type, select the **Load Type** used in the generator from the following options: Infinite Bus or Independent Generator.

Infinite Bus	When the generator is connected to an infinite bus of the power system, the terminal voltages remain constant and the output power is determined by the output currents.
Independent Generator	The generator supplies electrical power to a load inductor. For an AC generator, the resistance and inductance of the load inductor are determined by the rated output power, the rated voltage, and the rated power factor. For a DC generator, the resistance of the load resistor is determined by the rated output power and the rated voltage.

3. Enter the output power developed at the shaft of the motor in the **Rated Output Power** field.
4. Enter either the RMS line-to-line voltage (for AC control type), or the DC voltage (for DC and PWM control types) in the **Rated Voltage** field.
5. Enter the desired output speed of the motor at the load point in the **Rated Speed** field.
6. Enter the temperature at which the system functions in the **Operating Temperature** field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Click the **Adjust-Speed Synchronous Machine** tab, and select either **Time** or **Frequency** as the **Domain** for the solution.
8. Click **OK** to close the **Solution Setup** window.

Solution Data for Adjust-Speed Synchronous Machines

To access the solution data, right-click **Analysis** in the Project Manager tree, and click **Add Solution Setup**. For this machine type, there is only one tab, the **General** tab.

The **Solution Setup** window contains the following fields:

Operation Type	The operation type is automatically set to **Motor** for this machine type.
Load Type	Select from **Const Speed**, **Const Power**, **Const Torque**, **Linear Torque**, and **Fan Load**. The default is **Const Power**.
Rated Output Power	Type a value for the rated output voltage, and select the units.
Rated Voltage	Type a value for the rated voltage, and select the units.
Rated Speed	Type a value for the rated speed, and select the units.
Operating Temperature	Type a value for the operating temperature, and select the units.

Permanent-Magnet DC Motors

After you have selected **Permanent-Magnet DC Motors** as your model type, you need to define the following:

- General data, such as the voltage, speed, and circuit type of the model.
- Stator data, such as the diameter, slot dimensions, and skew width of the stator.
- Stator pole data, such as its associated pole dimensions, type of steel, and pole magnet specifications.
- Rotor data, such as the slot types and dimensions, slot data, and windings.
- Commutator and brush data, such as the commutator dimensions and brush length.
- Shaft data
- Solution data, such as rated output voltage and frequency.

By option, you can add a machine housing.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Analysis Approach for PMDC Motors

For a permanent-magnet DC motor, the stator is equipped with P pairs of permanent magnets, creating P pairs of alternating north and south poles. The distribution of the magnetic field produced by the permanent magnet's field flux is fixed with respect to the stator. The rotor is equipped with a distributed winding connected to a commutator that revolves together with the rotor.

A system of brushes is kept in permanent electrical contact with the commutator. When DC current is applied to the rotor winding (via the brushes and commutator), a torque is produced by the interaction of the rotor (armature) currents and the field produced by the permanent magnets.

The commutator causes the armature to create a magnetic flux distribution that is fixed in space and whose axis is perpendicular to the axis of the field flux produced by the permanent magnets. For these motors, the commutator acts as a mechanical rectifier.

The performance of a permanent-magnet DC (PMDC) motor is computed by DC analysis only. The voltage equation of a PMDC motor is

$$U = U_b + R_1 * I + E$$

where U_b is the voltage drop of one-pair brushes, R_1 is the armature resistance, $E = K_e * \omega$ is the back emf with K_e the back-emf constant in Vs/rad, and ω is the speed in rad/s. For a given speed ω, armature current can be computed based on the applied voltage U, as shown below:

$$I = (U - U_b - K_e * \omega)/R_1$$

The shaft torque T_2 is computed by

$$T_2 = K_t * I - T_{fw}$$

where K_t is the torque constant in Nm/A, which is numerically the same as K_e, and T_{fw} is the frictional torque.

The output power (mechanical power) is

$$P_2 = T_2 * \omega$$

The input power (electrical power) is

$$P_1 = P_2 + P_{fw} + P_{Cua} + P_b + P_{Fe}$$

where *Pfw*, *Pcua*, *Pb*, and *PFe* are frictional and wind loss, armature copper loss, brush drop-loss, and iron core-loss, respectively.

The efficiency is

$$\eta = \frac{P_2}{P_1} \times 100\%$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining a Permanent-Magnet DC Motor

The general procedure for defining a permanent-magnet DC motor is as follows:

1. Insert the permanent-magnet DC motor into a new or existing project.
2. Double-click the **Machine** entry in the Project Manager tree to define the general data.
3. Double-click the **Machine** > **Stator** entry in the Project Manager tree to define the stator geometry.
4. Double-click the **Machine** > **Stator** > **Pole** entry in the Project Manager tree to define the stator pole dimensions.
5. Double-click the **Machine** > **Rotor** entry in the Project Manager tree to define the rotor geometry.
6. Double-click the **Machine** > **Rotor** > **Slot** entry in the Project Manager tree to define the rotor slot dimensions.
7. Double-click the **Machine** > **Rotor** > **Winding** entry in the project tree to define the rotor windings and conductors.
8. Double-click the **Machine** > **Commutator** entry in the Project Manager tree to define the commutator and brush data.
9. Double-click the **Machine** > **Shaft** entry in the Project Manager tree to define the magnetism of the shaft.
10. Optionally, for Inner Rotor machines only, right-click the **Machine** entry in the Project Manager tree and click **Insert Housing** to add a machine housing.
11. Right-click **Analysis** in the Project Manager tree, and click **Add Solution Setup** to define the solution data.
12. Choose **File** > **Save** to save the project.
13. Choose **RMxprt** > **Analyze** to analyze the design.

> **Note:** When you place the cursor over an entry field in the data windows, a brief description of that field appears in the status bar at the bottom of the **RMxprt** window.

Once analyzed, the model can be viewed in the Maxwell 2D Modeler, or it can be used to create a new Maxwell 2D project, and a new Maxwell 3D project.

Refer to the *Permanent-Magnet DC Motor Problem* application note, on the technical support page of the Ansys web site, for a specific example of a permanent-magnet DC motor problem.

Defining the General Data for PMDC Motors

Use the **General** window to specify the rated output power, voltage values, circuit type, and speed of the DC motor.

To define the general data:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. To open the **General Data Properties** window, double-click the **Machine** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the number of poles for the machine in the **Number of Poles** field. This value is the total number of poles in the stator (or the number of pole pairs multiplied by two).
3. Enter the energy loss due to friction at the given speed in the **Frictional Loss** field.

   > **Note:** To use the **Brush Press** and **Frictional Coefficient** fields when you define the commutator and brush later in the **Commutator/Brush Data** window, enter **0** here for the **Friction Loss**.

4. Enter the wind loss due to air resistance measured at the reference speed in the **Wind Loss** field.
5. Enter the given speed in the **Reference Speed** field.
6. Click **OK** to close the **Properties** window.

General Data for PMDC Motors

To access the general data, double-click the **Machine** entry in the Project Manager tree.

The **General Data Properties** window for a three-phase induction motor contains the following fields:

Machine Type	The machine type you selected when inserting a new RMxprt design (DC Permanent Magnet Motor)
Number of Poles	The number of poles the machine contains
Rotor Position	Select whether the rotor is an **Inner Rotor** or **Outer Rotor**
Frictional Loss	The frictional energy loss (due to friction) measured at the reference speed
Wind Loss	The wind loss (due to air resistance) measured at the reference speed
Reference Speed	The given speed of reference

Defining the Stator Data for a PMDC Motor

The stator is the outer lamination stack where the polyphase voltage windings reside.

To define the general stator data:

1. To open the **Stator Data Properties** window, double-click the **Machine** > **Stator** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

desktop without opening a separate window.)

2. Enter the **Outer Diameter** of the stator.
3. Enter the **Inner Diameter** of the stator.
4. Enter the length of the stator core in the **Length** field.
5. Enter the stacking factor for the stator core in the **Stacking Factor** field. This value is a ratio of the effective magnetic length of the core, and ranges from 0 to 1. It is defined as the total length minus the total insulation from the laminations, divided by the total length. A value of 1 indicates that the stator is not laminated.
6. Click **OK** to close the **Properties** window.

Stator Data for PMDC Motors

To access the general stator data, double-click the **Machine** > **Stator** entry in the Project Manager tree.

The **Stator Data Properties** window contains the following fields:

Outer Diameter	The outer diameter of the stator core
Inner Diameter	The inner diameter of the stator core
Length	The length of the stator core
Stacking Factor	The stacking factor of the stator core

Defining the Stator Pole for a PMDC Motor

The rotor pole drives the electromagnetic field which is coupled with the stator windings. Use the **Stator Pole Data Properties** window to define the stator pole.

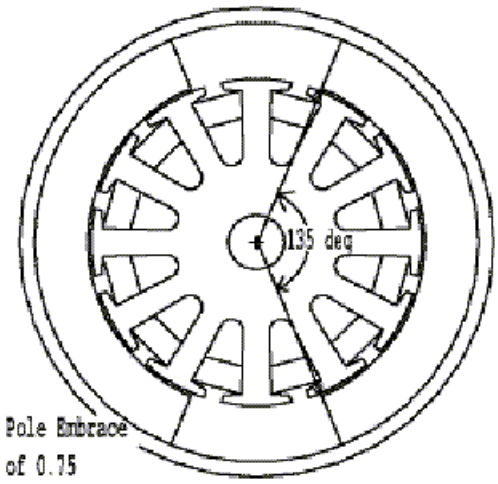
To define the rotor pole:

1. To open the **Stator Pole Data Properties** window, double-click the **Machine** > **Stator** > **Pole** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)

   > **Note:** For a two-pole machine, a pole embrace of 0.75 yields a magnet with a span of 135 degrees (based on 0.75*180 degrees).

2. Enter the ratio of the actual arc distance in relation to the maximum possible arc distance in the **Embrace** field. This value is between 0 and 1.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Enter the distance from the center of the stator to the magnet arc center in the **Offset** field. Enter 0 for a uniform air gap.
4. To select the type of magnet to use in the rotor pole:
    a. Click the **Magnet Type** button.
    The **Select Definition** window appears.
    b. Select or define a material for the magnet type.
    c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.
5. Enter the length of the magnet in the axial direction in the **Magnet Length** field.
6. Enter the maximum radial thickness of the magnet at the center of the pole in the **Magnet Thickness** field. To control the flux, the magnet's thickness may vary.
7. Click **OK** to close the **Properties** window.

Stator Pole Data for PMDC Motors

To access the stator pole data, double-click the **Machine** > **Stator** > **Pole** entry in the Project Manager tree.

The **Stator Pole Data Properties** window contains the following fields:

Embrace	The pole embrace
Offset	The pole-arc center offset from the stator center (0 for a uniform air gap)
Magnet Type	The type of magnet. Click the button to open the **Select Definition** window.
Magnet Length	The maximum length of the magnet
Magnet Thickness	The maximum thickness of the magnet

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining the Rotor Data for a PMDC Motor

The rotor is equipped with slots containing copper conductors that are connected to the commutator. The commutator acts as a mechanical rectifier in the motor.

Use the **Rotor Data Properties**, **Rotor Slot Data Properties**, and **Rotor Winding Data Properties** windows to define the rotor slots, windings, and dimensions.

To define general rotor data:

1. To open the **Rotor Data Properties** window, double-click the **Machine** > **Rotor** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the stacking factor for the rotor core in the **Stacking Factor** field.
3. Enter the **Number of Slots** in the rotor.
4. Select the **Slot Type**:
    a. Click the button for the **Slot Type**.
    b. In the **Select Slot Type** window that appears, select a slot type (available types include 1 through 4).

    > **Note:** When you place the mouse cursor over the slot type, a schematic outline of the slot appears.

    c. Click **OK** to close the **Select Slot Type** window and return to the **Properties** window.
5. Enter the outer diameter of the rotor in the **Outer Diameter** field.
6. Enter the inner diameter of the rotor in the **Inner Diameter** field.
7. Enter the length of the rotor core in the **Length** field.
8. Select a **Steel Type** for the rotor core:
    a. Click the button for **Steel Type**.
    b. In the **Select Definition** window that appears, select a steel type from the list, or define a new steel type.
    c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.
9. Enter the skew width, measured in slot number, in the **Skew Width** field.
10. Click **OK** to close the **Properties** window.

Rotor Data for PMDC Motors

To access the general rotor data, double-click the **Machine** > **Rotor** entry in the Project Manager tree.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Rotor Data Properties** window contains the following fields:

Stacking Factor	The stacking factor of the rotor core
Number of Slots	The number of slots the rotor core contains
Slot Type	The type of slots in the rotor core. Click the button to open the **Select Slot Type** window
Outer Diameter	The outer diameter of the rotor core
Inner Diameter	The inner diameter of the rotor core
Length	The length of the rotor core
Steel Type	The steel type of the rotor core. Click the button to open the **Select Definition** window.
Skew Width	The skew width measured in slot number

Defining the Rotor Slots for a PMDC Motor

To define the physical dimensions of the rotor slots:

1. To open the **Rotor Slot Data Properties** window, double-click the **Machine** > **Rotor** > **Slot** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Optionally, to automatically design the dimensions of slots **Hs2**, **Bs1**, and **Bs2**, select the **Auto Design** check box.
3. Enter the available slot dimensions:
   - **Hs0** Always available
   - **Hs2** Available only when **Auto Design** is cleared. When **Auto Design** is selected, this slot dimension is determined automatically.
   - **Bs0** Always available
   - **Bs1** Available only when **Auto Design** is cleared. When **Auto Design** is selected, this slot dimension is determined automatically. When **Parallel Tooth** is selected, this slot dimension is determined based on the value entered in the **Tooth Width** field.
   - **Bs2** Available only when **Auto Design** is cleared. When **Auto Design** is selected, this slot dimension is determined automatically. When **Parallel Tooth** is selected, this slot dimension is determined based on the value entered in the **Tooth Width** field.
   - **Rs** Rs is added when the slot type is 3 or 4
4. Click **OK** to close the **Properties** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rotor Slot Data for PMDC Motors

To access the stator slot data, double-click the **Machine** > **Rotor** > **Slot** entry in the Project Manager tree.

The **Rotor Slot Data Properties** window contains the following fields:

Auto Design	Select or clear this to enable or disable auto-design of slots **Hs2**, **Bs1**, and **Bs2**. When this check box is selected, only two other fields appear in the window: **Hs0** and **Bs0**.
Hs0	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected)
Hs2	A slot dimension (see the diagram shown in the modeling when **Machine** > **Rotor** > **Slot** is selected)
Bs0	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected)
Bs1	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected)
Bs2	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected)
Rs	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected); Rs is added when the slot type is 3 or 4.

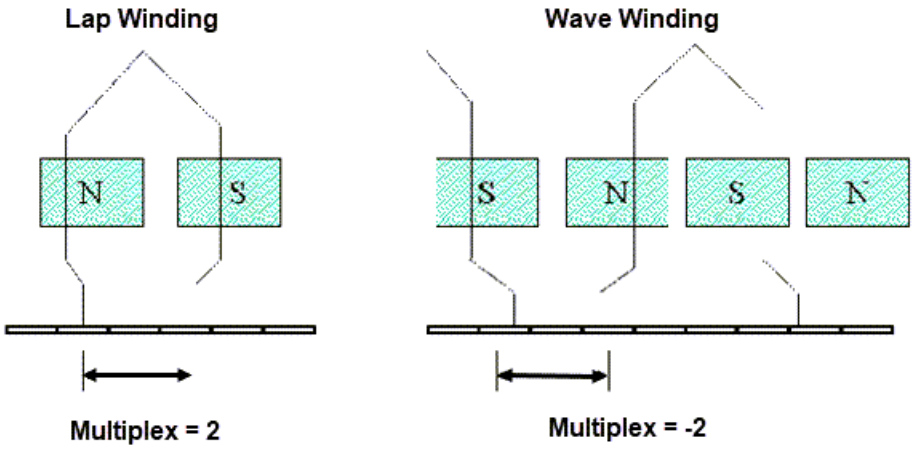
Defining the Rotor Windings and Conductors for a PMDC Motor

To define the rotor windings, wires, and conductors:

1. To open the **Rotor Slot Winding Properties** window, double-click the **Machine** > **Rotor** > **Winding** entry in the Project Manager tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Click the **Winding** tab.
3. Select a **Winding Type**:
    a. Click the button for **Winding Type**.
    b. In the **Winding Type** window that appears, select from one of the following three types of winding:
        - **Lap**
        - **Wave**
        - **Frog Leg**
4. Enter the number of windings in the **Multiplex Number** field (1 for a single winding, 2 for double windings, 3 for triple windings). For a lap winding, the multiplex number is the number of commutators between the start and end of one winding, and the number of

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

parallel branches is equal to the number of poles multiplied by the multiplex number. For a wave winding, the number of parallel branches equals the multiplex number multiplied by two.

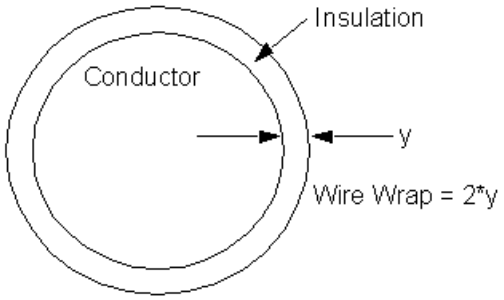


5. Enter the number of virtual slots per each real slot in the **Virtual Slots** field. The rotor is assumed to have two layers of conductors, an upper and a lower layer. Each layer of conductors can have a number of windings, which are referred to as virtual slots.

   > **Note:** For example, the upper and lower layer can have two windings each, which would yield a virtual slot number of two; for a 12-slot machine, this would yield 24 commutation segments.

6. Enter the total number of conductors in each rotor slot in the **Conductors per Slot** field. This value is the number of turns per coil multiplied by the number of layers. This value is the total number of conductors in one real full rotor slot.
7. Enter the coil pitch, measured in number of slots, in the **Coil Pitch** field. The coil pitch is the number of slots separating one winding. For example, if a coil starts in slot 1 and ends in slot 6, it has a coil pitch of 5.
8. Enter the number of wires per conductor in the **Number of Strands** field. Enter 0 to have RMxprt auto-design this value.
9. Enter the thickness of the double-sided wire wrap in the **Wire Wrap** field. Enter 0 to automatically obtain this value from the wire library.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



10. Select the **Wire Size**:
    a. Click the button for **Wire Size**.
    b. In the **Wire Size** window that appears, select a value from the **Wire Diameter** drop-down menu.
    c. Select a wire gauge from the **Gauge** drop-down menu. You can select from the following options:

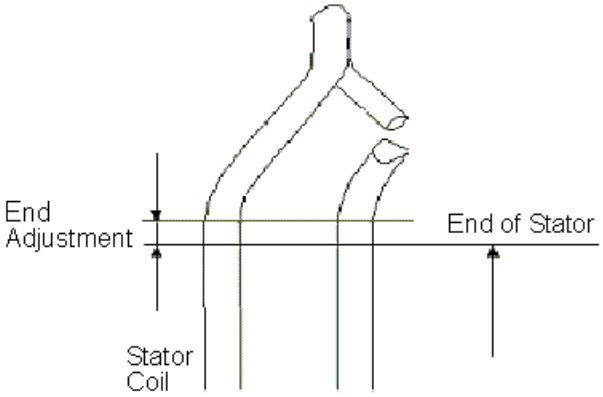
<*number*>	You can select a specific gauge number. When you select a gauge number, the **Wire Diameter** field is automatically updated.
USER	This option allows you to manually enter the **Wire Diameter**. This is useful when you want to enter a diameter that does not correspond to a particular wire gauge.
AUTO	This option sets the **Wire Diameter** to zero, and RMxprt automatically calculates the optimal value. The diameter information is then written to the output file when you analyze the design.
MIXED	This option allows you to define a conductor that is made of different size wires. For example, a single conductor may consist of five wires: three wires with a diameter of 0.21 mm and two wires with a diameter of 0.13 mm.

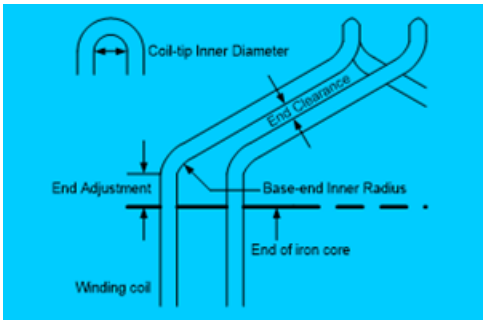
    The gauge number is based on AWG settings. You can create your own wire table using **Machine > Wire**, and then you can select this wire table using the **Tools** > **Options** > **General Options** > **Machines** option. When you are done setting the wire size, click **OK** to close the **Wire Size** window and return to the **Properties** window.

11. Click the **End/Insulation** tab.
12. Select or clear the **Input Half-turn Length** check box.
13. Do one of the following:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you selected **Input Half-turn Length**, then enter the half-turn length of the armature winding in the Half Turn Length field.
- If you cleared **Input Half-turn Length**, then enter the end length adjustment of the stator coils in the **End Adjustment** field. The end adjustment is the distance one end of the conductor extends vertically beyond the end of the stator.

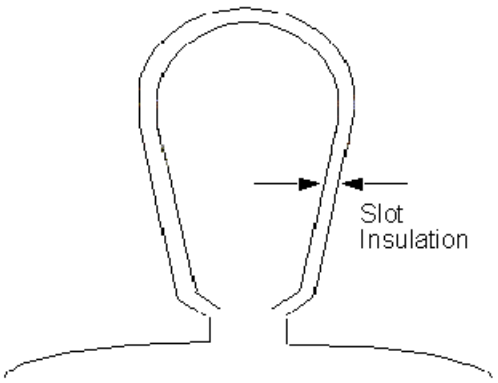




14. Enter the inner radius of the base corner in the **Base Inner Radius** field.
15. Enter the inner diameter of the coil tip in the **Tip Inner Diameter** field.
16. Enter the distance between two rotor coils in the **End Clearance** field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

17. Enter the thickness of the slot liner insulation in the **Slot Liner** field.



18. Enter the thickness of the wedge insulation in the **Wedge Thickness** field.
19. Enter the thickness of the insulation layer in the **Layer Insulation** field.
20. Enter the limited slot fill factor for the wire design in the **Limited Fill Factor** field. The slot fill factor is the ratio between the cross-sectional area of all conductors in one slot and the entire slot area.
21. Select the type of equalizer connection from the **Equalizer Connection** drop-down menu. Select from **None**, **Half**, or **Full**.
22. Click **OK** to close the **Properties** window.

Defining Different Size Wires for a PMDC Motor

Use the **Gauge** option if you have a conductor that is made up different size wires.

To define different size wires:

1. In the **Wire Size** window, select **MIXED** from the **Gauge** drop-down menu.
2. Select either **Round** or **Rectangular** as the **Wire Type**.
3. Enter the appropriate wire data in the table:
    - For a round wire:
        - Enter the **Diameter** in the table.
        - Enter a **Number** in the table to specify how many of the conductor's wires have this diameter.
    - For a rectangular wire:
        - Enter the **Width** of the wire in the table.
        - Enter the **Thickness** of the wire in the table.
        - Enter the **Fillet** value in the table.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Enter a **Number** in the table to specify how many of the conductor's wires have this data.

4. Click **Add** to add the new wire data.
5. Repeat steps 3 and 4 for each size wire you want to add.
6. When you are finished defining the wires, click **OK** to close the **Wire Size** window and return to the RMxprt **Properties** window.

Note: For example, if one conductor is made up of five wires, and three of those wires have a diameter of 0.21 mm, and the other 2 have a diameter of 0.13 mm, then the mixed wire size table will have two lines. The first line will list Diameter = 0.21 and Number = 3. The second line will list Diameter = 0.13 and Number = 2. An equivalent wire diameter is displayed as Wire Size value in the **Winding** tab in the **Properties** window.

Rotor Winding Data for PMDC Motors

To access the stator winding data, double-click the **Machine** > **Rotor** > **Winding** entry in the Project Manager tree.

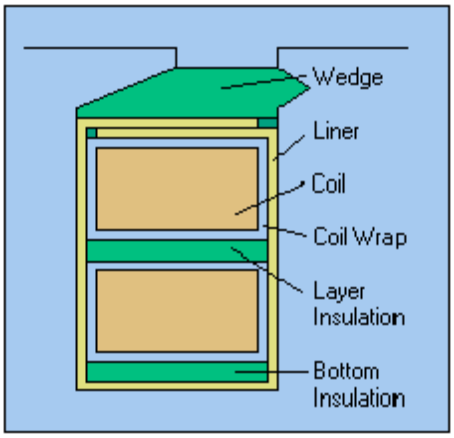
The **Rotor Winding Data Properties** window contains the following fields:

Winding tab	**Winding Type**	The type of rotor winding. Click the button to open the **Winding Type** window and choose from **Lap**, **Wave**, and **Frog Leg**.
	Multiplex Number	Single, double, or triple windings (1, 2, or 3).
	Virtual Slots	The number of virtual slots per real slot.
	Conductors per Slot	The number of conductors per rotor slot (0 for auto-design).
	Coil Pitch	The coil pitch measured in number of slots.
	Number of Strands	The number of wires per conductor (0 for auto-design).
	Wire Wrap	The thickness of the double-sided wire wrap (0 to automatically obtain this value from the wire library).
	Wire Size	The diameter of the wire (0 for auto-design). Click the button to open the **Wire Size** window where you can

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		specify units, wire type, diameter, and gauge.
End/Insulation tab	**Input Half-turn Length**	Select or clear this check box to specify whether or not you want to enter the half-turn length. When this check box is selected, the **Half Turn Length** field appears the next time you open the **Properties** window. When this check box is selected, the **End Adjustment** field appears instead.
	Half Turn Length	The half-turn length of the armature winding
	Base Inner Radius	The inner radius of the base corner
	Tip Inner Diameter	The inner diameter of the coil tip
	End Clearance	The end clearance between two adjacent coils
	Slot Liner	The thickness of the slot liner insulation
	Wedge Thickness	The thickness of the wedge insulation
	Layer Insulation	The thickness of the insulation layer
	Limited Fill Factor	The limited slot fill factor for the wire design. The slot fill factor is the ratio between the cross-sectional area of all conductors in one slot and the entire slot area.
	Equalizer Connection	The connection type of the equalizer. Select from **None**, **Half**, or **Full**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Defining the Commutator and Brush for a PMDC Motor

The commutator allows current transfer between DC terminals or brushes and the rotor coils, providing the current to the system as a function of rotation. Due to the action of the commutator, the corresponding magnetic field has a fixed distribution with respect to the stator.

To define the commutator and brush pairs:

1. To open the **Commutator Data Properties** window, double-click the **Machine** > **Commutator** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Click the **Commutator** tab.
3. Select **Cylinder** or **Pancake Type** as the **Commutator Type**.

   > **Note:** When you place the mouse cursor over the commutator type, an outline of the commutator appears.

4. For **Cylinder** commutators, do the following:
   a. Enter the **Commutator Diameter**.
   b. Enter the **Commutator Length**.
5. For **Pancake** commutators, do the following:
   a. Enter the **Outer Diameter**.
   b. Enter the **Inner Diameter**.
6. Enter the thickness of the insulation between two consecutive commutator segments in the **Commutator Insulation** field.
7. Click the **Brush** tab.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Enter the **Brush Width**.
9. Enter the **Brush Length**.
10. Enter the number of brush pairs when using a wave armature winding in the **Brush Pairs** field.
11. Enter the angle of displacement from the neutral axis, in mechanical degrees, in the **Brush Displacement** field.

    > **Note:** The brush displacement is positive for the counter-clockwise direction. For example, if the rotor turns clockwise and the brush displacement is also clockwise, then the angle is negative; if the rotor turns clockwise but the brush displacement is counter-clockwise, then the angle is positive.

12. Enter the voltage drop across one brush pair in the **Brush Drop** field.
13. Enter the mechanical pressure of the brushes as they press against the commutator in the **Brush Press** field.
14. Enter the **Frictional Coefficient** of the brush.

    > **Note:** If the **Friction Loss** field is used in the **General** window, the **Brush Press** and **Frictional Coefficient** fields will be hidden in the **Commutator/Brush** window. These fields are shown only when the **Friction Loss** field in the **General** window is set to zero.

15. Click **OK** to close the **Properties** window.

Commutator and Brush Data for PMDC Motors

To access the commutator and brush data, double-click the **Machine** > **Commutator** entry in the Project Manager tree.

The **Commutator Data Properties** window contains the following fields:

Commutator tab	**Commutator Type**	The type of commutator. Click the button to open the **Select Commutator Type** window and select from **Cylinder** or **Pancake**.
	Commutator Diameter	For a **Cylinder** commutator type, the diameter of the commutator
	Commutator Length	For a **Cylinder** commutator type, the length of the commutator
	Outer Diameter	For a **Pancake** commutator type, the outer diameter of the commutator

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Inner Diameter	For a **Pancake** commutator type, the inner diameter of the commutator
	Commutator Insulation	The thickness of the insulation between the two commutator bars
Brush tab	**Brush Width**	The width of the brush
	Brush Length	The length of the brush
	Brush Pairs	The number of brush pairs
	Brush Displacement	The displacement of the brush from the neutral position, in mechanical degrees (positive for anti-rotating direction).
	Brush Drop	The voltage drop across a one-pair brush
	Brush Press	The brush press per unit area. (Available only when **Frictional Loss** is set to zero for the machine.)
	Frictional Coefficient	The frictional coefficient of the brush. (Available only when **Frictional Loss** is set to zero for the machine.)

Defining the Shaft Data for a PMDC Motor

To define the shaft:

1. To open the **Shaft Data Properties** window, double-click the **Machine** > **Shaft** entry in the Project Manager tree. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Select or clear the **Magnetic Shaft** check box to specify whether or not the shaft is to be made of magnetic material.
3. Click **OK** to close the **Properties** window.

Shaft Data for PMDC Motors

To access the shaft data, double-click the **Machine** > **Shaft** entry in the Project Manager tree.

The **Shaft Data Properties** window contains the following fields:

Magnetic Shaft	Select or clear this check box to indicate whether or not the shaft is made of magnetic material.

Setting Up Analysis Parameters for a PMDC Motor

To define the solution data:

1. To open the **Solution Setup** window, right-click **Analysis** in the Project Manager tree, and click **Add Solution Setup**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click the **General** tab. The **Operation Type** is automatically set to **Motor** for this machine type.
3. Select the **Load Type** used in the motor from the following options:

Const Speed	The speed remains constant in the motor.
Const Power	The output power remains constant in the motor.
Const Torque	The torque remains constant regardless of the speed. In this case, $T_{load} = T_{rated}$, given by the output power divided by the given rated speed.
Linear Torque	The torque increases linearly with speed. In this case, $T_{load} = T_{rated} * (n/n_{rated})$ where T_{rated} is given by the output power divided by the given rated speed.
Fan Load	The load varies nonlinearly with speed. In this case, $T_{load} = T_{rated} * (n/n_{rated})^2$ where T_{rated} is given by the output power divided by the given rated speed.

4. Enter the output power developed at the shaft of the motor in the **Rated Output Power** field.
5. Enter the RMS line-to-line voltage in the **Rated Voltage** field.
6. Enter the desired output speed of the motor at the load point in the **Rated Speed** field.
7. Enter the temperature at which the system functions in the **Operating Temperature** field.
8. Click **OK** to close the **Solution Setup** window.

Solution Data for PMDC Motors

To access the solution data, right-click **Analysis** in the Project Manager tree, and click **Add Solution Setup**. For this machine type, there is only one tab, the **General** tab.

The **Solution Setup** window contains the following fields:

Operation Type	The operation type is automatically set to **Motor** for this machine type.
Load Type	Select from **Const Speed**, **Const Power**, **Const Torque**, **Linear Torque**, and **Fan Load**. The default is **Const Power**.
Rated Output Power	Type a value for the rated output voltage, and select the units.
Rated Voltage	Type a value for the rated voltage, and select the units.
Rated Speed	Type a value for the rated speed, and select the units.
Operating Temperature	Type a value for the operating temperature, and select the units.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Three-Phase Synchronous Machines

After you have selected **Three-Phase Synchronous Machines** as your model type, you need to define the following:

- General data, such as the unit system, power, and voltage.
- Stator data, such as the slot types and dimensions, stator diameter, skew width, and laminations.
- Optional stator Vent data.
- Winding data, such as the parallel branches, conductors, and wire dimensions.
- Rotor pole data, such as its associated pole-body dimensions and air gaps.
- Optional Rotor damper data, such as the damper dimensions, rings, and material properties.
- Rotor winding data and the winding control parameters.
- Shaft Data
- Optional machine housing.
- Solution data, such as specifying motor or generator application, and rated output voltage and frequency.

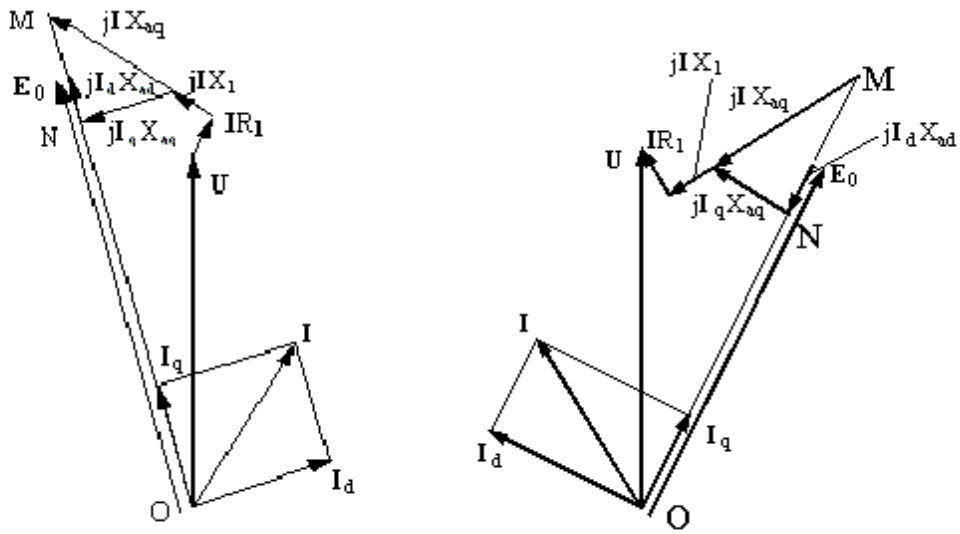
Also see the Analysis Approach for Three-Phase Synchronous Machines.

Analysis Approach for Three-Phase Synchronous Machines

The three-phase salient-pole synchronous electric machine has two types: the generator and the motor. Their basic structures are the same. Three-phase synchronous generators are the main source of electrical energy for industrial, commercial, and private use. They receive mechanical energy at the shaft and transform it into electrical energy. The rotor is equipped with a multipole winding excited by a DC source. The stator is equipped with a three-phase winding that has a sinusoidal spatial distribution. The spinning rotor produces a rotating magnetic field in the air gap of the machine. The frequency of the voltage induced in the stator is given by $f = pv$, where p is the number of pairs of poles, and v is the velocity of the rotor. The machine is capable of producing both active and reactive power as required by the load connected at the stator phasor.

The three-phase salient-pole synchronous electric machine has two types: the generator and the motor. Their basic structures are the same. Usually the frequency-domain phasor diagram is adopted to analyze the characteristics. The phasor diagram for a generator is shown on the left and that for a motor is shown on the right.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the figure, R_1, X_1, X_{ad}, and X_{aq} are armature resistance, armature leakage reactance, d-axis armature reactance, and q-axis armature reactance, respectively. X_{ad} is nonlinear, while a linearized value is used in the phasor diagram. Taking the input voltage **U** as the reference phasor, for a given current:

$$\mathbf{I} = I\angle -\varphi$$

where φ is the power factor angle, a phasor represented by OM can be derived by $\mathbf{U} + \mathbf{I}(R_1 + jX_1 + jX_{aq})$.

The direction of $\mathbf{E}_0$ can, therefore, be obtained. Taking the power angle, the angle that **U** lags $\mathbf{E}_0$, as θ, then the angle that **I** lags $\mathbf{E}_0$ is $\psi = \theta + \varphi$

The d- and q-axis currents are then represented by the following:

$$I_d = I * sin(\psi)$$

$$I_q = I * cos(\psi)$$

The phasor length ON represents the d-axis back EMF from d-axis resultant flux linkage and is used to determine the d-axis field saturation. Then a frozen method is applied to derive $\mathbf{E}_0$, X_{ad}, and exciting current I_f.

The output power (electric power) is directly computed from voltage and current as

$$P_2 = 3*U*I*cos(\varphi)$$

The input power (mechanical power) is defined as

$$P_1 = P_2 + P_{fw} + P_{cua} + P_{Fe} + P_{add} + P_{cuf} + P_{ex}$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where P_{fw}, P_{cua}, P_{Fe}, P_{add}, P_{cuf}, and P_{ex} are frictional and wind loss, armature copper loss, iron-core loss, additional loss, field winding copper loss, and exciter loss, respectively.

The input mechanical shaft torque is $T_1 = P_1/\omega$, where ω is synchronous speed in rad/s.

The efficiency is computed by $eff = P_2/P_1 * 100\%$.

Main Features

- **Adapted to both Synchronous Motor and Generator**: The structures of the salient-pole synchronous motor and the generator are basically the same, but their phasor relationships and the computation methods are slightly different, their output characteristics data are also different. This is specified in the solution setup.
- **Auto Arrangement of Three-phase Windings**: Almost all commonly used three-phase single- and double-layer, half- and whole-type ac windings (including fractional-pitch windings) can be automatically arranged. Users do not need to define coils one by one. RMxprt also supports a double-layer winding with half-turn coils which are auto-arranged in the order of even, odd, even, odd, …, and even, odd, as long as it is physically possible.

When a designer adopts single-layer whole-coiled windings, RMxprt will perform winding arrangement optimization to minimize the average coil pitch. When asymmetric three-phase windings are used, winding arrangement is optimized in such a way that minimum negative-sequence and zero-sequence components are achieved.

- **Winding Editor Supporting Any Single- and Double-Layer Windings**: Besides taking advantage of the winding auto-arrangement function in RMxprt, users can also specify any special winding by using of the Winding Editor function.

In Winding Editor, through modification of phase belonging, number of turns, in-slot and out-slot number of each coil, it is possible to design single- and double-layer winding arrangement for any purposes.

- **Analyze Air-Gap Magnetic Field Distribution**: For both uniform and non-uniform air gaps, Schwarz-Christopher Transformation is adopted to solve for the air-gap magnetic field distribution.
- **Analyze EMF Waveform and Total Harmonic Distortion** (THD): Based on the analysis of the air-gap magnetic field waveform, taking into account coil short pitch, winding distribution, skew slot, winding connection, load effects and other factors, the emf waveforms in the coils and the windings are analyzed to solve for the emf distortion factors.
- **Analyze Dynamic Parameters of Damping Winding**: Different from the squirrel-cage winding of the induction machine, the damping winding of the salient-pole synchronous machine is located in the surface of magnetic field poles, which deviates greatly along the d- and the q-axes. Furthermore, the connection of damping bars has several forms. The bars under each pole could be connected, but not connected with those under other poles.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

All the bars could be connected together. The bars could be connected through the endplate. RMxprt can deal with all those complicated situations and give the dynamic parameters for the damping winding.

Defining a Three-Phase Synchronous Machine

The general procedure for defining a three-phase synchronous machine is as follows:

1. Insert a three-phase synchronous machine into a existing or new project.
2. Double-click the **Machine** entry in the project tree to define the general data.
3. Double-click the **Machine** > **Stator** entry in the project tree to define the stator geometry.
4. Double-click the **Machine** > **Stator** > **Slot** entry in the project tree to define the stator slot dimensions.
5. Optionally, you can add a vent to, or remove an existing vent from the stator. To add a vent, select the stator, and right-click to display the pop-up menu for **Insert Vent**.
6. Double-click the **Machine** > **Stator** > **Winding** entry in the project tree to define the stator windings, conductors, and insulation data.
7. Double-click the **Machine** > **Rotor** entry in the project tree to define the general rotor geometry, the pole data, and the insulation data.
8. Double-click the **Machine** > **Rotor** > **Winding** entry in the project tree to define the rotor conductors and windings.
9. Optionally, you can add a damper to the design or remove an existing damper. To add a damper, use **Machine** > **Insert Damper**. This inserts the damper in the project tree under the rotor. You must then specify the slot type and other properties for the damper.
10. Optionally, you can add a vent to, or remove an existing vent from the rotor. To add a vent select the rotor, and right-click to display the pop-up menu. Use **Insert Vent**.
11. Double-click the **Machine** > **Shaft** entry in the project tree to define the magnetism of the shaft.
12. Optionally, right-click the **Machine** entry in the project tree and click **Insert Housing** to add a machine housing.
13. Right-click **Analysis** in the project tree, and click **Add Solution Setup** to define this solution data.
14. Choose **File** > **Save** to save the project.
15. Choose **RMxprt > Analyze** to analyze the design.

> **Note:** When you place the cursor over an entry field, a brief description of that field appears in the status bar at the bottom of the **RMxprt** window.

Once the design is analyzed, the model can be viewed in the Maxwell 2D Modeler, or it can be used to create a new Maxwell 2D project, and a Maxwell 3D design.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining the General Data for a Three-Phase Synchronous Machine

Use the **General Data Properties** window to define the power settings, speed, and efficiency of the generator. This window allows you to define the basic parameters of the synchronous generator, such as power, voltage, winding connections, and losses.

To define the general data:

1. To open the **General Data Properties** window, double-click the **Machine** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the number of poles for the machine in the **Number of Poles** field. This value is the total number of poles in the stator (or the number of pole pairs multiplied by two).
3. Enter the power lost through frictional forces in the **Frictional Loss** field.
4. Enter the wind loss measured at the reference speed in the **Wind Loss** field.
5. Enter the given speed in the **Reference Speed** field.
6. Click **OK** to close the **Properties** window.

General Data for Three-Phase Synchronous Machines

To access the general data, double-click the **Machine** entry in the project tree.

The **General Data Properties** window for a three-phase induction motor contains the following fields:

Machine Type	The machine type you selected when inserting a new RMxprt design (Three Phase Synchronous Machine)
Number of Poles	The number of poles the machine contains
Frictional Loss	The frictional energy loss (due to friction) measured at the reference speed
Wind Loss	The wind loss (due to air resistance) measured at the reference speed
Reference Speed	The given speed of reference

Defining the Stator for a Three-Phase Synchronous Machine

Use the **Stator** windows to define the slot dimensions, stacking factors, air ducts, and insulation of the stator. The stator is the outer lamination stack where the three-phase windings reside.

To define general stator data:

1. To open the **Stator Data Properties** window, double-click the **Machine** > **Stator** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

desktop without opening a separate window.)

2. Enter the **Outer Diameter** of the stator.
3. Enter the **Inner Diameter** of the stator.
4. Enter the length of the stator core in the **Length** field.
5. Enter the stacking factor for the stator core in the **Stacking Factor** field.
6. Select a **Steel Type** for the stator core:
   a. Click the button for **Steel Type**.
   b. In the **Select Definition** window that appears, select a steel type from the list, or define a new steel type.
   c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.
7. Enter the **Number of Slots** in the stator.
8. Select the**Slot Type**:
   a. Click the button for the **Slot Type**.
   b. In the **Select Slot Type** window that appears, select a slot type (available types include 1 through 6). Slot types **1** though **4** are filled with round wire. Slot types **5** and **6** are filled with rectangular wire. If **Auto Design** is enabled, the software designs an optimum slot geometry; in this case, you can input the tooth width dimension, and the software determines the slot width accordingly.

      > **Note:** When you place the mouse cursor over the slot type, a schematic of the selected type appears, displaying the slot dimension variables.

   c. Click **OK** to close the **Select Slot Type** window and return to the **Properties** window.
9. Enter the number of sectors in the **Lamination Sectors** field.
10. Enter the thickness of the magnetic pressboard in the **Pressboard Thickness** field. Enter **0** for a non-magnetic pressboard.
11. Enter the skew width, measured in slot number, in the **Skew Width** field.
12. Click **OK** to close the **Properties** window.

Stator Data for Three-Phase Synchronous Machines

To access the general stator data, double-click the **Machine** > **Stator** entry in the Project Manager tree.

The **Stator Data Properties** window contains the following fields:

Outer Diameter	The outer diameter of the stator core
Inner Diameter	The inner diameter of the stator core

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Length	The length of the stator core
Stacking Factor	The stacking factor of the stator core
Steel Type	The steel type of the stator core. Click the button to open the **Select Definition** window.
Number of Slots	The number of slots the stator core contains
Slot Type	The type of slots in the stator core. Click the button to open the **Select Slot Type** window.
Lamination Sectors	The number of lamination sectors
Pressboard Thickness	The magnetic press board thickness (0 for a non-magnetic press board)
Skew Width	The skew width measured in slot number

Defining Stator Slots for a Three-Phase Synchronous Machine

To define the slot dimensions:

1. To open the **Stator Slot Data Properties** window, double-click the **Machine** > **Stator** > **Slot** entry in the Project Manager tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Optionally, to automatically design the dimensions of slots **Hs2**, **Bs1**, and **Bs2**, select the **Auto Design** check box.
3. Optionally, to design dimensions of slots **Bs1** and **Bs2** based on the stator tooth width, select the **Parallel Tooth** check box, and enter a value in the **Tooth Width** field.
4. Enter the available slot dimensions:
   - **Hs0** Always available
   - **Hs2** Available only when **Auto Design** is cleared. When **Auto Design** is selected, this slot dimension is determined automatically.
   - **Bs0** Always available
   - **Bs1** Available only when **Auto Design** and **Parallel Tooth** are both cleared. When **Auto Design** is selected, this slot dimension is determined automatically. When **Parallel Tooth** is selected, this slot dimension is determined based on the value entered in the **Tooth Width** field.
   - **Bs2** Available only when **Auto Design** and **Parallel Tooth** are both cleared. When **Auto Design** is selected, this slot dimension is determined automatically. When **Parallel Tooth** is selected, this slot dimension is determined based on the value entered in the **Tooth Width** field.
5. Click **OK** to close the **Properties** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stator Slot Data for Three-Phase Synchronous Machines

To access the stator slot data, double-click the **Machine** > **Stator** > **Slot** entry in the Project Manager tree.

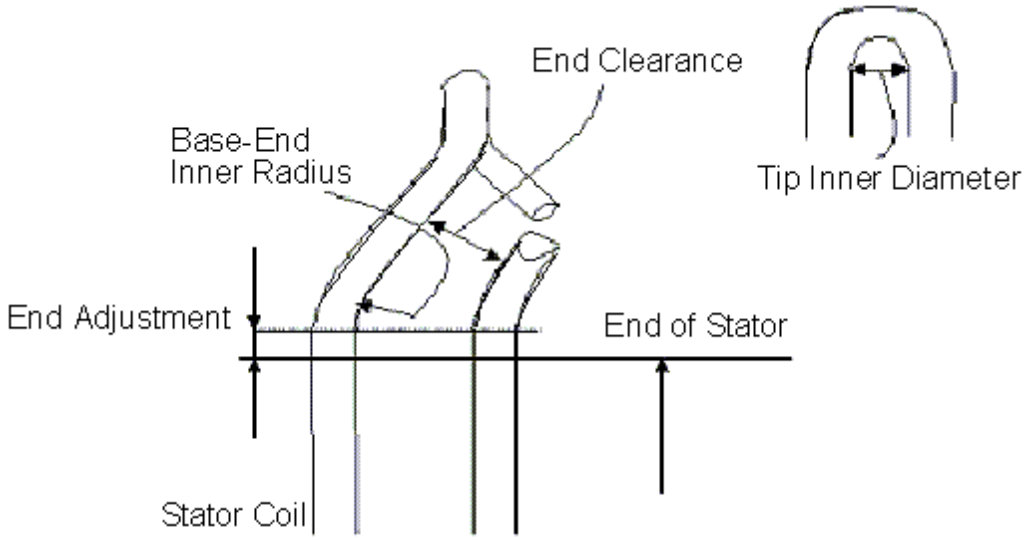
The **Stator Slot Data Properties** window contains the following fields:

Auto Design	Select or clear this to enable or disable auto-design of slots **Hs2**, **Bs1**, and **Bs2**. When this check box is selected, only two other fields appear in the window: **Hs0** and **Bs0**.
Parallel Tooth	Select this to design **Bs1** and **Bs2** based on the tooth width. When this check box is selected, the **Bs1** and **Bs2** fields are removed, and the **Tooth Width** field is added.
Tooth Width	The tooth width for the parallel tooth, on which **Bs1** and **Bs2** are designed
Hs0	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected)
Hs2	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected)
Bs0	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected)
Bs1	A slot dimension (see the diagram shown in the modeling window when **Machine > Stator > Slot is selected**)
Bs2	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected).
Rs	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected); Rs is added when the slot type is 3 or 4.

Defining Stator Windings and Insulation for a Three-Phase Synchronous Machine

Use the **Stator Winding** window to define the stator winding data, such as the coils, wires, insulation, number of parallel branches, and physical dimensions of the windings.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The stator winding data defines the configuration of one phase of the three-phase windings.

To define the stator windings and insulation:

1. To open the **Stator Slot Winding Properties** window, double-click the **Machine** > **Stator** > **Winding** entry in the Project Manager tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Click the **Winding** tab.
3. Enter the number of layers in the stator winding in the **Winding Layers** field.
4. Select a **Winding Type**:
    a. Click the button for **Winding Type**.
    b. In the **Winding Type** window that appears, select from one of the following three types of winding:
        - **Whole Coiled**
        - **Half Coiled**
        - **Editor**

    When you place the mouse cursor over a winding button, an outline of the selected winding appears. The following table describes the six types of windings that are possible (three for one-layer and three for two-layer):

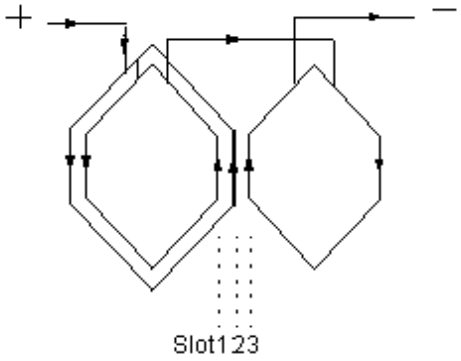
Type	Description
Editor	A user-defined one-layer winding arrangement. You need to set up the winding arrangement for each slot. For this winding type, the following letters are used for the phase windings: • Phase A/A return uses A/X.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Phase B/B return uses B/Y.
- Phase C/C return uses C/Z.

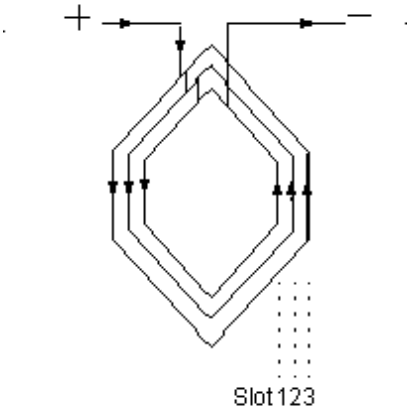
A one-layer, whole-coiled winding:

Whole Coiled



A one-layer concentric half-coiled winding:

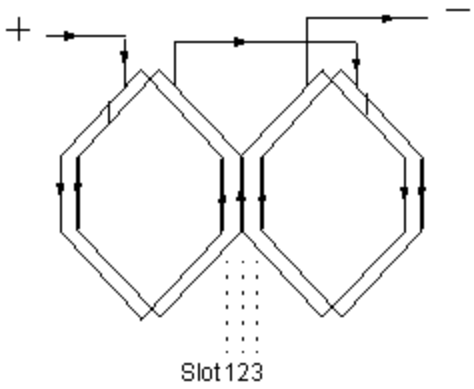
Half Coiled



Editor A user-defined two-layer winding arrangement. When you select **20**, the Winding Editor opens, where you can specify a different winding arrangement for each slot.

Whole Coiled A two-layer wave winding:

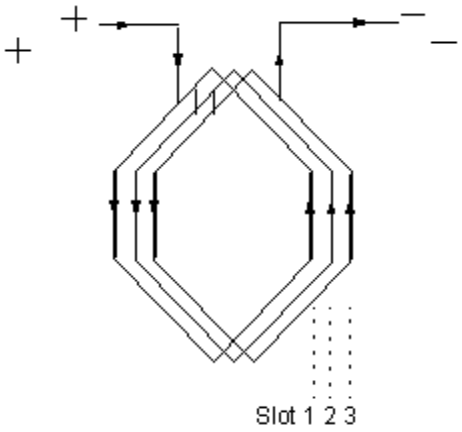
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The phase belt for this winding configuration is equal to 360/2*m*, where *m* is the phase number.

A two-layer, half-coiled winding:

Half Coiled



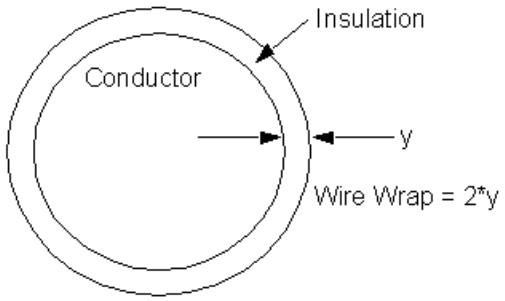
There is only one coil per phase per pair of poles.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
> **Example 1:** A one layer winding arranged in 12 slots should be defined as type **10**, with the following arrangement: **AAZZBBXXCCYY**
>
> **Example 2:** A two layer winding arranged in 12 slots should be defined as type **20**, with the following arrangement: **AAZZBBXXCCYY**
>
> Only the top layer needs to be defined; the bottom layer will be determined according to the coil pitch.

c. Once you have clicked a button to select a winding, click **OK** to close the **Winding Type** window and return to the **Properties** window.

5. Select or enter the number of parallel branches in one phase of the winding in the **Parallel Branches** field.
6. Enter the total number of conductors in each stator slot in the **Conductors per Slot** field. This value is the number of turns per coil multiplied by the number of layers.
7. Enter the coil pitch, measured in number of slots, in the **Coil Pitch** field. The coil pitch is the number of slots separating one winding. For example, if a coil starts in slot 1 and ends in slot 6, it has a coil pitch of 5.
8. Enter the number of wires per conductor in the **Number of Strands** field. Enter 0 to have RMxprt auto-design this value.
9. Enter the thickness of the double-sided wire wrap in the **Wire Wrap** field. Enter 0 to automatically obtain this value from the wire library.



10. Select the **Wire Size**: Click the button for **Wire Size**. The **Wire Size** window appears. Select a value from the **Wire Diameter** drop-down menu. Select a wire gauge from the **Gauge** drop-down menu. You can select from the following options:

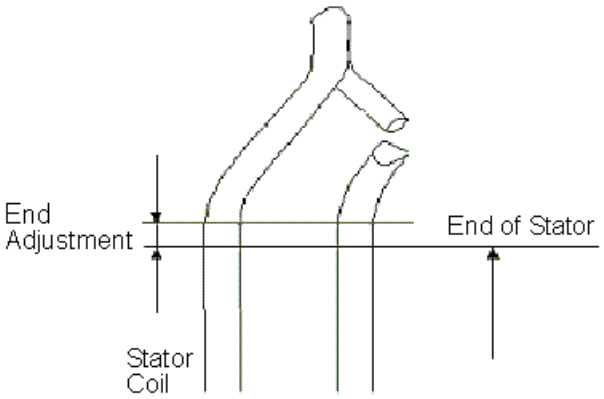
<*number*>	You can select a specific gauge number. When you select a gauge number, the **Wire Diameter** field is automatically updated.
USER	This option allows you to manually enter the **Wire Diameter**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

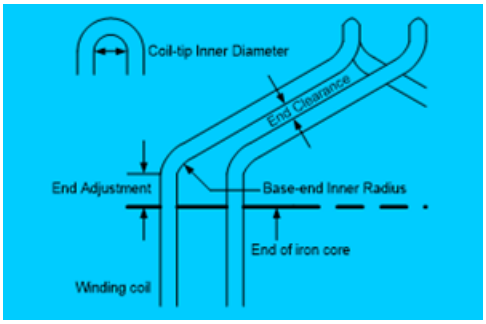
	This is useful when you want to enter a diameter that does not correspond to a particular wire gauge.
AUTO	This option sets the **Wire Diameter** to zero, and RMxprt automatically calculates the optimal value. The diameter information is then written to the output file when you analyze the design.
MIXED	This option allows you to define a conductor that is made of different size wires. For example, a single conductor may consist of five wires: three wires with a diameter of 0.21 mm and two wires with a diameter of 0.13 mm.

The gauge number is based on AWG settings. You can create your own wire table using **Machine > Wire**, and then you can select this wire table using the **Tools** > **Options** > **General Options** > **Machines** command. When you are done setting the wire size, click **OK** to close the **Wire Size** window and return to the **Properties** window.

11. Click the **End/Insulation** tab.
12. Select or clear the **Input Half-turn Length** check box.
13. Do one of the following:
    - If you selected **Input Half-turn Length**, then enter the half-turn length of the armature winding in the Half Turn Length field.
    - If you cleared **Input Half-turn Length**, then enter the end length adjustment of the stator coils in the **End Adjustment** field. The end adjustment is the distance one end of the conductor extends vertically beyond the end of the stator.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



14. Enter the inner radius of the base corner in the **Base Inner Radius** field.
15. Enter the inner diameter of the coil tip in the **Tip Inner Diameter** field.
16. Enter the distance between two stator coils in the **End Clearance** field.
17. Enter the thickness of the slot liner insulation in the **Slot Liner** field.
18. Enter the thickness of the wedge insulation in the **Wedge Thickness** field.
19. Enter the thickness of the insulation layer in the **Layer Insulation** field.
20. Enter the limited slot fill factor for the wire design in the **Limited Fill Factor** field. The slot fill factor is the ratio between the cross-sectional area of all conductors in one slot and the entire slot area.
21. Click **OK** to close the **Properties** window.

Stator Winding and Insulation for Three-Phase Synchronous Machines

To access the stator winding and insulation data, double-click the **Machine** > **Stator** > **Winding** entry in the Project Manager tree.

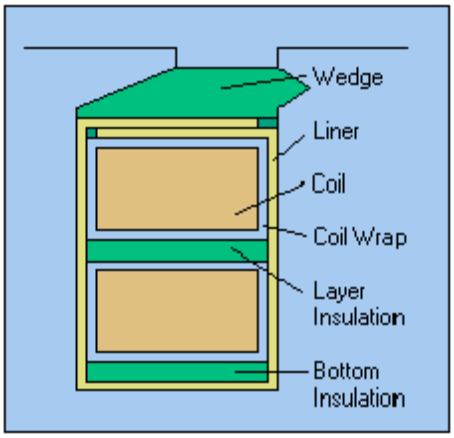
The **Stator Winding Data Properties** window contains the following fields:

Winding tab	**Winding Layers**	The number of winding layers.
	Winding Type	The type of stator winding. Click the button to open the **Winding Type** window and choose from **Whole Coiled**, **Half Coiled**, and **Editor**.
	Parallel Branches	The number of parallel branches in the stator winding.
	Conductors per Slot	The number of conductors per stator slot (0 for auto-design).
	Coil Pitch	The coil pitch measured in number of slots.
	Number of Strands	The number of wires per conductor (0 for auto-design).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Wire Wrap	The thickness of the double-sided wire wrap (0 to automatically obtain this value from the wire library).
	Wire Size	The diameter of the wire (0 for auto-design). Click the button to open the **Wire Size** window where you can specify units, wire type, diameter, and gauge.
End/Insulation tab	**Input Half-turn Length7**	Select or clear this check box to specify whether or not you want to enter the half-turn length. When this check box is selected, the **Half Turn Length** field appears the next time you open the **Properties** window. When this check box is selected, the **End Adjustment** field appears instead.
	Half Turn Length	The half-turn length of the armature winding.
	End Adjustment	The end length adjustment of the stator coils, which is the distance one end of the conductor extends vertically beyond the end of the stator.
	Base Inner Radius	The inner radius of the base corner.
	Tip Inner Diameter	The inner diameter of the coil tip.
	End Clearance	The end clearance between two adjacent coils.
	Slot Liner	The thickness of the slot liner insulation.
	Wedge Thickness	The thickness of the wedge insulation.
	Layer Insulation	The thickness of the insulation layer.
	Limited Fill Factor	The limited slot fill factor for the wire design. The slot fill factor is the ratio between the cross-sectional area of all conductors in one slot and the entire slot area.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Winding Editor for a Three-Phase Synchronous Machine

For a three-phase synchronous machine, you may want to specify a different number of conductors for each stator slot. The Winding Editor makes this possible by enabling you to specify the number of turns for each coil.

To specify the number of turns for each coil:

1. Click **Machine** > **Winding** > **Edit Layout**.
2. In the **Winding Editor** window that appears, from the table in the upper left, set which phase you want for each coil and which slot is the “in” and “out” slot for the current in each coil.
3. If you are working on a quarter or half model, you may want to specify a multiplier by clicking the **Periodic Multiplier** check box and specifying a value.
4. Select or deselect the **Constant Turns** or **Constant Pitch** check boxes, depending on whether you want to be able to change these setting in the table above. When these options are selected, you cannot change the turns or pitch.
5. When you are satisfied with the coil settings, click **OK** to close the **Winding Editor** window.

Defining Different Size Wires for a Three-Phase Synchronous Machine

Use the **Gauge** option if you have a conductor that is made up different size wires.

To define different size wires:

1. In the **Wire Size** window, select **MIXED** from the **Gauge** drop-down menu.
2. Select either **Round** or **Rectangular** as the **Wire Type**.
3. Enter the appropriate wire data in the table:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For a round wire:
  - Enter the **Diameter** in the table.
  - Enter a **Number** in the table to specify how many of the conductor's wires have this diameter.
- For a rectangular wire:
  - Enter the **Width** of the wire in the table.
  - Enter the **Thickness** of the wire in the table.
  - Enter the **Fillet** value in the table.
  - Enter a **Number** in the table to specify how many of the conductor's wires have this data.

4. Choose **Add** to add the new wire data.
5. Repeat steps 3 and 4 for each size wire you want to add.
6. When you are finished defining the wires, click **OK** to close the **Wire Size** window.

> **Note:** For example, if one conductor is made up of five wires, and three of those wires have a diameter of 0.21 mm, and the other 2 have a diameter of 0.13 mm, then the mixed wire size table will have two lines. The first line will list Diameter = 0.21 and Number = 3. The second line will list Diameter = 0.13 and Number = 2. An equivalent wire diameter is displayed as Wire Size value in the Winding tab in the Properties window.

Stator Vent Data for Three-Phase Synchronous Machines

To insert a vent on a stator for a three-phase synchronous machine:

1. Right-click on the stator icon in the Project Manager tree to display the shortcut menu.
2. Click **Insert Vent**.

The vent icon appears in the Project Manager tree under the stator.

To remove an existing vent item:

1. Right-click on the stator icon in the Project Manager tree to display the shortcut menu.
2. Click **Remove Vent**.

This removes the vent item from the Project Manager tree.

To access the Vent properties for a vent, double-click on a vent item. The **Vent Properties** window contains the following fields.

Vent Ducts	The number of radial vent ducts
Duct Width	The width of the radial vent ducts

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Magnetic spacer width	Width of magnetic spacer that holds vent ducts; 0 for non-magnetic spacer.
Duct pitch	Center-to-Center distance between two adjacent Vent ducts

Defining the Rotor for a Three-Phase Synchronous Machine

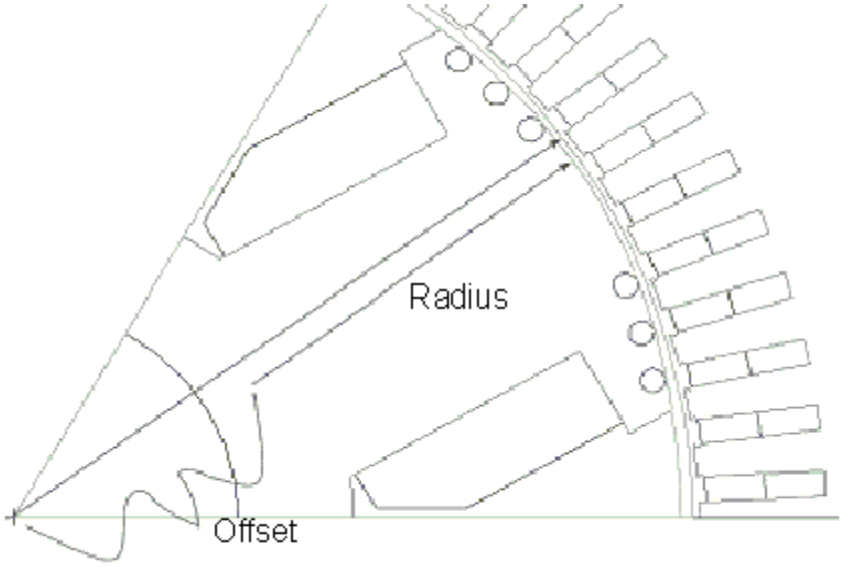
The rotor consists of copper bars in which current is induced by the magnetic fields produced by the stator windings.

To define the general rotor data:

1. To open the **Rotor Data Properties** window, double-click the **Machine** > **Rotor** entry in the Project Manager tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Click the **Rotor** tab.
3. Enter the outer diameter of the rotor in the **Outer Diameter** field.
4. Enter the inner diameter of the rotor in the **Inner Diameter** field.
5. Enter the length of the rotor core in the **Length** field.
6. Select a **Steel Type** for the rotor core:
    a. Click the button for **Steel Type**.
    b. In the **Select Definition** window that appears, select a steel type from the list, or define a new steel type.
    c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.
7. Enter the stacking factor for the rotor core in the **Stacking Factor** field.
8. Click the **Pole** tab.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

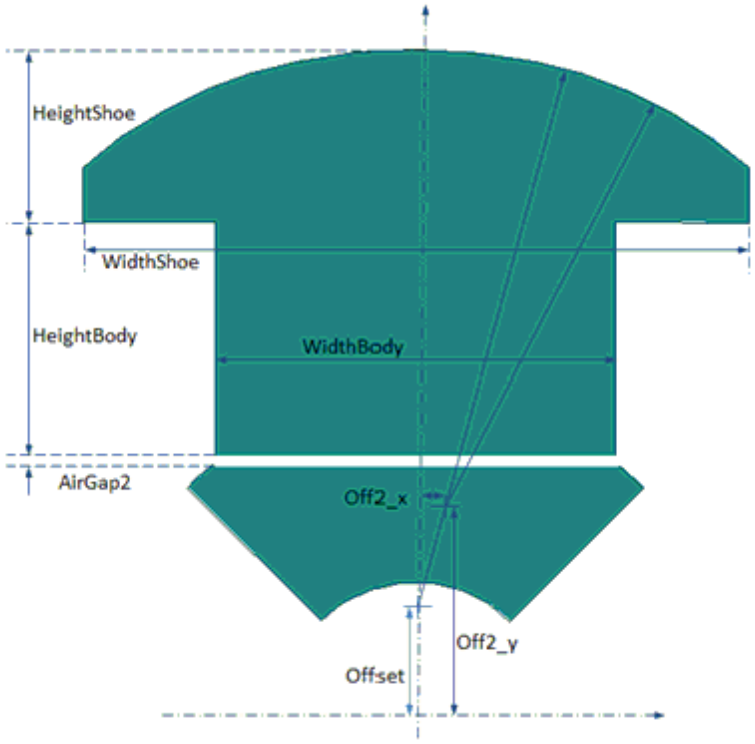
9. Enter the pole-arc center offset from the rotor center in the **Pole Arc Offset** field.



10. Enter the width of the pole shoe in the **Pole Shoe Width** field.
11. Enter the height of the pole shoe in the **Pole Shoe Height** field.
12. Enter the width of the pole body in the **Pole Body Width** field.
13. Enter the height of the pole body in the **Pole Body Height** field.
14. Enter the width between the rotor pole and rotor yoke in the **Second Air Gap** field.
15. To include the two arcs in the half-pole range, do the following:
    a. Select the **Select Pole Arc** check box.
    b. Enter the offset of the second arc perpendicular to the pole-center line in the **Off2_x** field.
    c. Enter the offset of the second arc parallel with the pole-center line in the **Off2_y** field.

> **Note:** If the Off2_x and Off2_y parameters for the second arc are invalid, the second arc is ignored and the pole includes only the first arc defined by Pole Arc Offset.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



16. Select or clear the **Magnetic PressBoard** check box to specify whether or not the press board is made of magnetic material.
17. Enter the thickness of the press board in the **Press Board Thickness** field.
18. Click the **Insulation** tab.
19. Enter the thickness of the insulating material beneath the shoe pole in the **Shoe Insulation** field.
20. Enter the thickness of the insulating material on the side of the pole body in the **Pole Insulation** field.
21. Enter the clearance distance between the windings in the **Winding Clearance** field.
22. Click **OK** to close the **Properties** window.

Rotor, Rotor Pole, and Insulation for Three-Phase Synchronous Machines

To access the general rotor data, pole data, and insulation data double-click the **Machine** > **Rotor** entry in the Project Manager tree.

The **Rotor Data Properties** window contains the following fields:

Rotor tab	**Outer Diameter**	The outer diameter of the rotor core

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Inner Diameter	The inner diameter of the rotor core
	Length	The length of the rotor core
	Steel Type	The steel type of the rotor core. Click the button to open the **Select Definition** window.
	Stacking Factor	The stacking factor of the rotor core
Pole tab	**Pole Arc Offset**	The pole-arc center offset from the rotor center
	Pole Shoe Width	The width of the pole shoe
	Pole Shoe Height	The height of the pole shoe
	Pole Body Width	The width of the pole body.
	Pole Body Height	The height of the pole body
	Second Air Gap	The width of the second air gap, between the rotor pole and rotor yoke
	Second Pole Arc	Select or clear this option to specify whether or not the pole surface includes the two arcs in the half-pole range. When you select this check box, two additional fields appear: **Off2_x** and **Off2_y**.
	Off2_x	The offset of the second arc perpendicular to the pole-center line. This field is only available when **Second Pole Arc** is selected.
		Note: If **Off2_x** for the second arc is invalid, the second arc is ignored and the pole includes only the first arc defined by **Pole Arc Offset**.
	Off2_y	The offset of the second arc parallel with the pole-center line. This field is only available when **Second Pole Arc** is selected.
		Note: If **Off2_y** for the second arc is invalid, the second arc is ignored and the pole includes only the first arc defined by **Pole Arc Offset**.
	Magnetic PressBoard	Select or clear this option to specify whether or not the press board is made of magnetic material.
	Press Board	The thickness of the press board

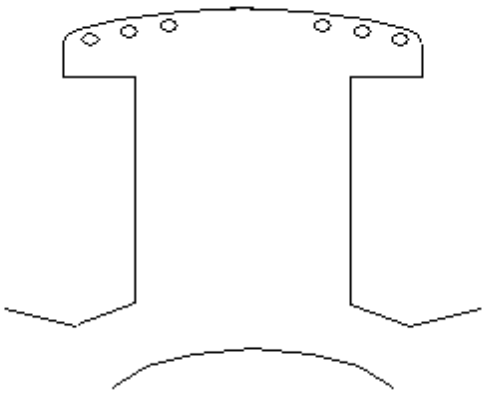
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Thickness	
	Steel Type	The steel type of the rotor pole. Click the button to open the **Select Definition** window. Default value is the same as the rotor core.
	Stacking Factor	The stacking factor of the rotor pole. Default value is the same as the rotor core.
Insulation tab	**Shoe Insulation**	The thickness of the insulating material beneath the pole shoe
	Pole Insulation	The thickness of the insulating material on the side of the pole body
	Winding Clearance	The clearance distance between the windings

Defining the Rotor Pole for a Three-Phase Synchronous Machine

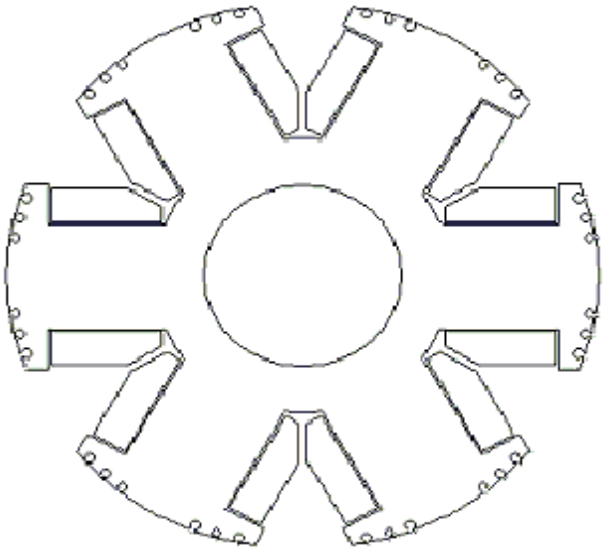
The rotor pole drives the electromagnetic field that is coupled with the stator windings.

The following figure shows a partial diagram of a rotor pole:



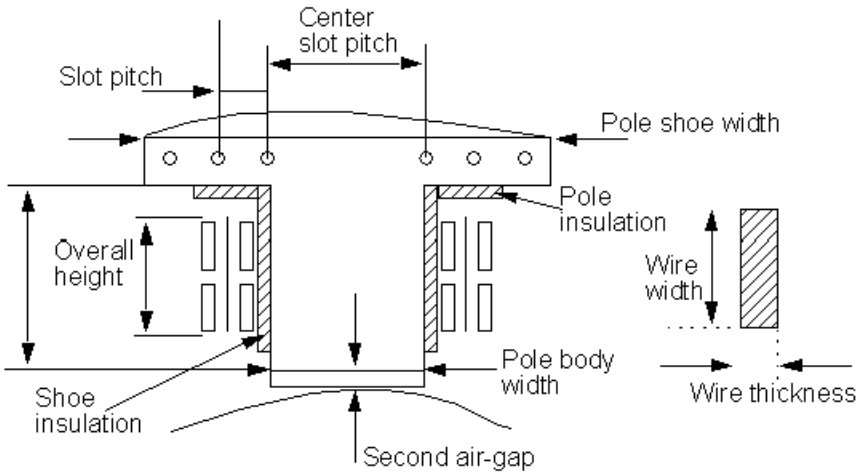
The following figure shows a diagram of an entire rotor:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Defining the Rotor Winding Data for a Three-Phase Synchronous Machine

Use the **Rotor Winding** window to define the wires and physical dimensions of the rotor winding. The rotor winding provides the excitation for the electromagnetic field that produces the rotor pole.



To define the rotor windings:

1. To open the **Rotor Winding Properties** window, double-click the **Machine** > **Rotor** > **Winding** entry in the Project Manager tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Select the **Winding Type** for the rotor:

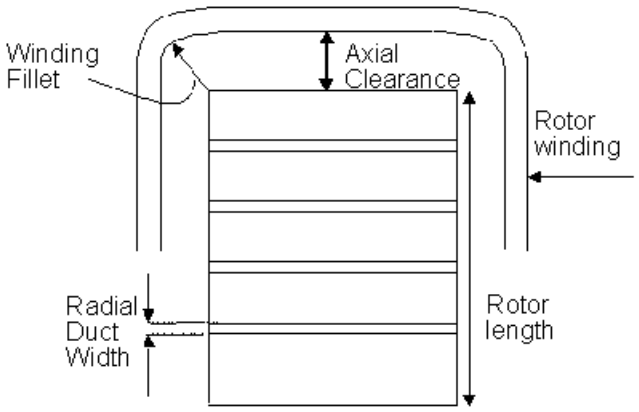
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Click the button.
b. In the **Winding Type** window that appears, click to select the type of winding, from **Round**, **Cylinder**, or **EdgeWise**. Note that when you place the mouse cursor over the winding type, its schematic appears.
c. Click **OK** to return to the **Properties** window.

3. Enter the number of parallel branches for the winding in the **Parallel Branches** field.
4. Enter a value for **Conductors per Pole**.
5. If the **Round** winding type is selected, specify the Number of Strands per conductor. Enter 0 for auto-design.
6. Enter the width of the insulating wire wrap in the **Wire Wrap** field.
7. For an **EdgeWise** winding type, enter the thickness of the inter-turn insulation in the Interturn Insulation field.

> **Note:** The Inter-turn Insulation field is not available when either the Round or Cylinder winding type has been selected.

8. Enter the gauge of the wire in the **Wire Size** field.
9. Enter the **Axial Clearance** to specify the axial distance between the core and the coil at the end of the lamination stack.



10. Enter the limited cross-section width for winding design or arrangement in the **Limited Cross Width** field. Enter 0 for maximum available area.
11. Enter the limited cross-section height for winding design or arrangement in the **Limited Cross Height** field. Enter 0 for maximum available area.
12. Enter the inner radius at winding fillet in the **Winding Fillet** field.
13. Click **OK** to close the **Properties** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rotor Winding Data for Three-Phase Synchronous Machines

To access the rotor winding data, double-click the **Machine** > **Rotor** > **Winding** entry in the Project Manager tree.

The **Rotor Winding Data Properties** window contains the following fields:

Winding Type	The type of rotor winding. Click the button to open the **Winding Type** window and choose from **Round**, **Cylinder**, and **Edgewise**.
Parallel Branches	The number of parallel branches in the rotor winding
Conductors per Pole	The number of conductors per rotor pole (0 for auto-design)
Number of Strands	The number of wires per conductor (0 for auto-design); only available with **Round** Winding Type
Wire Wrap	The thickness of the double-sided wire wrap (0 to automatically obtain this value from the wire library)
Interturn Insulation	The thickness of the inter-turn insulation of an edgewise winding; only available with the **EdgewiseCoil** Winding Type.
Wire Size	The diameter of the wire (0 for auto-design). Click the button to open the **Wire Size** window where you can specify units, wire type, diameter, and gauge.
Axial Clearance	The axial gap between the field winding and the pole body or inner coil
Limited Cross Width	The limited cross-section width for the winding design or arrangement (0 for available maximum area)
Limited Cross Height	The limited cross-section height for the winding design or arrangement (0 for available maximum area)
Winding Fillet	The size of the winding fillet

Defining the Rotor Damper Data

To define a rotor damper for a machine that permits one:

1. Click **Machine** > **Insert Damper**.

   The Damper icon appears in the Project Manager tree under the rotor icon. A slot icon appears in the hierarchy under the damper.

2. Double-click on the Damper icon to display the properties window for the damper.
3. Enter the appropriate values for the damper. The slot type, the bar conductor type, and

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

end conductor type are entered by clicking on buttons that open other windows.

4. Click **OK** to close the properties window.

Damper Data for Three-Phase Synchronous Machines

You can add a damper to or remove damper from the rotor of a three-phase machine.

To add a damper:

1. Right-click on the rotor icon in the Project Manager tree to display the shortcut menu.
2. Click **Insert Damper** on the menu.

The damper appears in the Project Manager tree under the rotor. The damper also includes an associated slot.

To remove a damper:

1. Right-click on the rotor icon in the Project Manager tree to display the shortcut menu.
2. Click **Remove Damper** on the menu.

The damper and associated slot are removed from the Project Manager tree.

The damper data contains the following fields:

Damper Slots Per Pole	Number of damper slots per pole
Slot Type	Damper slot type; specify this by clicking the button in the properties field and selecting from the Select Slot Type window.
Cast Rotor	Defines the rotor squirrel cage winding as cast
Bar Conductor Type	Specify this by clicking the button in the properties field, and using the Select Definition window to find and assign materials.
End Length	Single side end extended bar length
End Ring Width	Axial width of end ring
End Ring Height	Radial height of end ring
End Ring Conductor Type	Specify this by clicking the button in the properties field and using the Select Definition window to find and assign the material.
End Ring	Type of end ring for the damper. Specify this by clicking the button in the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type	properties field and use the Select Pole type window to select from the available types.
Slot Pitch	Slot pitch in mechanical degrees in reference to the offset center

Defining the Shaft Data for a Three-Phase Synchronous Machine

To define the shaft:

1. To open the **Shaft Data Properties** window, double-click the **Machine** > **Shaft** entry in the Project Manager tree. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Select or clear the **Magnetic Shaft** check box to specify whether or not the shaft is to be made of magnetic material.
3. Click **OK** to close the **Properties** window.

Shaft Data for Three-Phase Synchronous Machines

To access the shaft data, double-click the **Machine** > **Shaft** entry in the Project Manager tree.

The **Shaft Data Properties** window contains the following fields:

Magnetic Shaft	Select or clear this check box to indicate whether or not the shaft is made of magnetic material.

Setting Up Analysis Parameters for a Three-Phase Synchronous Machine

To define the solution data:

1. To open the **Solution Setup** window, right-click **Analysis** in the Project Manager tree, and click **Add Solution Setup**.
2. Click the **General** tab.
3. Select **Motor** or **Generator** from the **Operation Type** drop-down list.
4. Select the **Load Type** used in the motor from the following options:

Infinite Bus	When the generator is connected to an infinite bus of the power system, the terminal voltages remain constant and the output power is determined by the output currents.
Independent Generator	The generator supplies electrical power to a load inductor. For an AC generator, the resistance and inductance of the load inductor are determined by the rated output power, the rated voltage, and the rated power factor. For a DC generator, the resistance of the load resistor is determined by the rated output power and the rated voltage.
Const Speed	*For Motors.* The speed remains constant in the motor.
Const Power	*For Motors.* The output power remains constant in the motor.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Const Torque	*For Motors.* The torque remains constant regardless of the speed. In this case, $T_{load} = T_{rated}$, given by the output power divided by the given rated speed.
Linear Torque	*For Motors.* The torque increases linearly with speed. In this case, $T_{load} = T_{rated} * (n/n_{rated})$ where T_{rated} is given by the output power divided by the given rated speed.
Fan Load	*For Motors.* The load varies nonlinearly with speed. In this case, $T_{load} = T_{rated} * (n/n_{rated})^2$ where T_{rated} is given by the output power divided by the given rated speed.

5. Enter the output power developed at the shaft of the machine in the **Rated Output Power** field.
6. Enter the RMS line-to-line voltage in the **Rated Voltage** field.
7. Enter the desired output speed of the motor at the load point in the **Rated Speed** field.
8. Enter the temperature at which the system functions in the **Operating Temperature** field.
9. Click the **Three-Phase Synchronous Machine** tab.
10. Enter a value in the **Rated Power Factor** field.
11. Select **Wye** or **Delta** from the **Winding Connection** drop-down menu.
12. In the **Exciter Efficiency** field, enter the efficiency of the exciter used to supply the rotor winding with DC current if it is mechanically connected to the shaft of the generator. The efficiency value ranges between 0 and 1 and will only affect the total efficiency result.
13. To enter an **Input Exciting Current**, select the check box, enter a value, and select the units.
14. Click **OK** to close the **Solution Setup** window.

Solution Data for Three-Phase Synchronous Machines

To access the solution data, right-click **Analysis** in the project tree, and click **Add Solution Setup**.

The **Solution Setup** window contains the following fields:

Operation Type	On the **General** tab. Select from **Motor** or **Generator**.
Load Type	On the **General** tab. For a motor, select from **Const Speed**, **Const Power**, **Const Torque**, **Linear Torque**, and **Fan Load**. The default is **Const Power**. For a generator, select from **Infinite Bus** and **Independent Generator**.
Rated Output Power	On the **General** tab. Type a value for the rated output voltage, and select the units.
Rated Voltage	On the **General** tab. Type a value for the rated voltage, and select the units.
Rated Speed	On the **General** tab. Type a value for the rated speed, and select the units.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Operating Temperature	On the **General** tab. Type a value for the operating temperature, and select the units.
Rated Power Factor	On the **Three-Phase Synchronous Machine** tab. Type a value for the rated power factor.
Winding Connection	On the **Three-Phase Synchronous Machine** tab. Select from **Wye** or **Delta**.
Exciter Efficiency	On the **Three-Phase Synchronous Machine** tab. Type a percent for the exciter efficiency.
Input Exciting Current	On the **Three-Phase Synchronous Machine** tab. If you select this check box, then enter the exciting current, and select the units.

Brushless Permanent-Magnet DC Motors

After you have selected **Brushless Permanent-Magnet DC Motors** as your model type, you need to define the following:

- General data, such as the voltage, speed, and circuit type of the model.
- Circuit data, such as lead trigger angle, transistor drop, and control circuit information.
- Stator data, such as the diameter, slot dimensions, winding data, and skew width of the stator.
- Rotor data
- Rotor pole data, such as the magnet dimensions and stacking factor.
- Shaft data
- Optional machine housing.
- Solution data, such as rated output voltage and frequency.

Analysis Approach for Brushless PMDC Motors

The stator of a brushless DC motor is equipped with a polyphase winding. The phases are connected to the DC bus through a switching circuit. The switching sequence is controlled so that it is synchronized with the position of the rotor. As a result, the stator produces a rotating magnetic field.

The rotor is equipped with permanent magnets, creating a structure with the same number of poles at the stator. The stator switches act like a commutator in a classic DC motor.

In brushless permanent-magnet DC (BLDC) motors, the armature currents are commutated exactly according to rotor position. The signal of rotor position may be obtained from a position sensor, or from induced voltages for sensor-less control system.

The performance of BLDC motors is analyzed via a time-domain simulation. The voltage equation in the time domain is

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\begin{bmatrix} v_d \\ v_q \\ v_0 \end{bmatrix} - \begin{bmatrix} e_d \\ e_q \\ e_0 \end{bmatrix} = \begin{bmatrix} R_1 + L_d p & -L_q \omega_e & 0 \\ -L_d \omega_e & R_1 + L_q p & 0 \\ 0 & 0 & R_1 + L_0 p \end{bmatrix} \cdot \begin{bmatrix} i_d \\ i_q \\ i_0 \end{bmatrix}$$

where R_1, L_d, L_q, and L_0 are armature resistance, d-axis synchronous inductance, q-axis synchronous inductance, and 0-axis inductance, respectively. ω_e is rotor speed in electrical rad/s, and p represents for d/dt.

The transformations for terminal voltages, induced voltages, and winding currents are given by the following three equations:

$$\begin{bmatrix} v_d \\ v_q \\ v_0 \end{bmatrix} = C^T \cdot \begin{bmatrix} v_a \\ v_b \\ ? \end{bmatrix} : \quad \begin{bmatrix} e_d \\ e_q \\ e_0 \end{bmatrix} = C^T \cdot \begin{bmatrix} e_a \\ e_b \\ ? \end{bmatrix} \quad : \begin{bmatrix} i_a \\ i_b \\ ? \end{bmatrix} = C \cdot \begin{bmatrix} i_d \\ i_q \\ i_0 \end{bmatrix}$$

The transformation matrices for two-phase, three-phase, and four-phases systems, noted as C_2, C_3, and C_4, are as follows:

$$C_2 = \begin{bmatrix} cos\,\theta & sin\,\theta & 0 \\ \sin\theta & cos\,\theta & 0 \end{bmatrix}$$

$$C_3 = \sqrt{\tfrac{2}{3}} \begin{bmatrix} cos\,\theta & sin\,\theta & \frac{1}{\sqrt{2}} \\ \cos(\theta - \alpha) & sin\,(\theta - \alpha) & \frac{1}{\sqrt{2}} \\ cos\,(\theta - 2\alpha) & sin\,(\theta - 2\alpha) & \frac{1}{\sqrt{2}} \end{bmatrix}$$

$$C_4 = \begin{bmatrix} cos\,\theta & sin\,\theta & 0 \\ sin\,\theta & -cos\,\theta & 0 \\ -cos\,\theta & -sin\,\theta & 0 \\ -sin\,\theta & cos\,\theta & 0 \end{bmatrix}$$

where $\alpha = 2\pi/3$.

The input power (electric power) can now be computed from the voltage and current as

$$P_1 = \tfrac{1}{T} \int_0^T (v_d i_d + v_q i_q + v_0 i_0) dt$$

The output power (mechanical power) is

$$P_2 = P_1 - (P_{fw} + P_{Cua} + P_t + P_{Fe})$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where P_{fw}, P_{Cua}, P_t, and P_{Fe} are frictional and wind loss, armature copper loss, transistor/diode loss, and iron-core loss, respectively.

The output mechanical shaft torque T_2 is $T_2 = P_2/\omega$, where ω is the rotor speed in mechanical rad/s.

The efficiency is computed by

$$\eta = \frac{P_2}{P_1} \times 100\%$$

Defining a Brushless Permanent-Magnet DC Motor

The general procedure for defining a brushless permanent-magnet DC motor is as follows:

1. Insert the permanent magnet brushless DC motor into a new or existing project.
2. Double-click the **Machine** entry in the project tree to define the general data.
3. Double-click the **Machine** > **Circuit** entry in the project tree to define the control circuit.
4. Double-click the **Machine** > **Stator** entry in the project tree to define the stator geometry.
5. Double-click the **Machine** > **Stator** > **Slot** entry in the project tree to define the stator slot dimensions.
6. Double-click the **Machine** > **Stator** > **Winding** entry in the project tree to define the stator windings and conductors.
7. Double-click the **Machine** > **Rotor** entry in the project tree to define the rotor geometry.
8. Double-click the **Machine** > **Rotor** > **Pole** entry in the project tree to define the pole, embrace, offset, and air gap data for the rotor pole.
9. Double-click the **Machine** > **Shaft** entry in the project tree to define the magnetism of the shaft.
10. Optionally, right-click the **Machine** entry in the project tree and click **Insert Housing** to add a machine housing.
11. Right-click **Analysis** in the project tree, and click **Add Solution Setup** to define the solution data.
12. Choose **File** > **Save** to save the project.
13. Choose **RMxprt** > **Analyze** to analyze the design.

> **Note:** When you place the cursor over an entry field in the data windows, a brief description of that field appears in the status bar at the bottom of the **RMxprt** window.

Once analyzed, the model can be viewed in the Maxwell 2D Modeler, or it can be used to create a new Maxwell 2D project, and a new Maxwell 3D design.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For a specific example of a brushless permanent-magnet DC motor problem, see the *Brushless Permanent-Magnet DC Motor Problem* application note, on the technical support page of the Ansys web site.

Defining the General Data for a Brushless PMDC Motor

Use the **General** window to specify the rated output power, voltage values, circuit type, and speed of the brushless DC motor.

To define the general data:

1. To open the **General Data Properties** window, double-click the **Machine** entry in the Project Manager tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the number of poles for the machine in the **Number of Poles** field. This value is the total number of poles in the stator (or the number of pole pairs multiplied by two).
3. Enter the energy loss due to friction at the given speed in the **Frictional Loss** field.
4. Enter the wind loss due to air resistance measured at the reference speed in the **Wind Loss** field.
5. Enter the given speed in the **Reference Speed** field.
6. Select **DC** or **CCC** from the **Control Type** drop-down menu.
7. Select a **Circuit Type** from the following types:

   Y3 Y-connected, three-phase
   L3 Loop-type, three-phase
   S3 Star-type, three-phase
   C2 Cross-type, two-phase
   L4 Loop-type, four-phase
   S4 Star-type, four-phase

   The circuit types are based on industry standards. By default, type **Y3**, a three-phase, six-status circuit, is selected as the circuit type.

   > **Note:** When you place the mouse cursor over a circuit type, an outline schematic of the circuit appears.

8. Click **OK** to close the **Properties** window.

General Data for Brushless PMDC Motors

To access the general data, double-click the **Machine** entry in the project tree.

The **General Data Properties** window for a three-phase induction motor contains the following fields:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

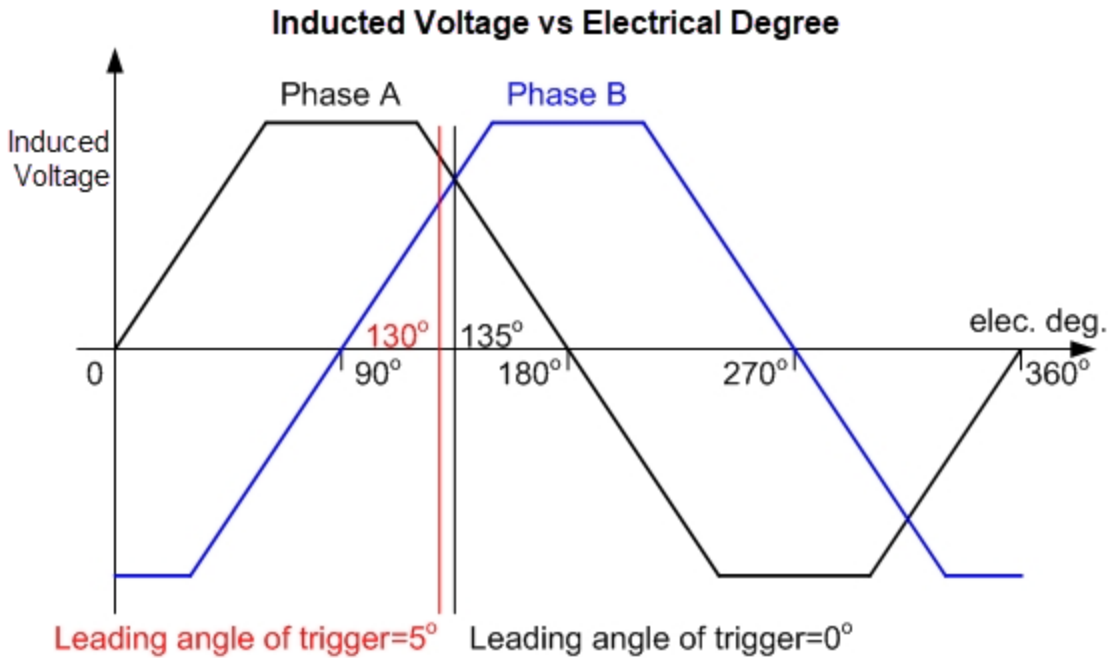
Machine Type	The machine type you selected when inserting a new RMxprt design (Brushless Permanent-Magnet DC Motor)
Number of Poles	The number of poles the machine contains
Rotor Position	Select whether the rotor is an **Inner Rotor** or **Outer Rotor**
Frictional Loss	The frictional energy loss (due to friction) measured at the reference speed
Wind Loss	The wind loss (due to air resistance) measured at the reference speed
Reference Speed	The given speed of reference
Control Type	The way the circuit is controlled. Select from **DC** or **CCC** (chopped current control).
Circuit Type	The drive circuit type. Click the button to open the **Circuit Type** window and select from the following six types: • **Y3**: Y-Type, 3-Phase • **L3**: Loop-Type, 3-Phase • **S3**: Star-Type, 3-Phase • **C2**: Cross-Type, 2-Phase • **L4**: Loop-Type, 4-Phase • **S4**: Star-Type, 4-Phase

Defining the Circuit Data for a Brushless PMDC Motor

Use the **Circuit Data Properties** window to define the circuit data for a brushless PMDC Motor.

1. To open the **Circuit Data Properties** window, double-click the **Machine** > **Circuit** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the trigger's lead angle in electrical degrees in the **Lead Angle of Trigger** field. The trigger's lead angle is shown in the following plot of the open circuit induced voltage versus position. An angle of 0 means that the induced voltage in the triggered phase is at a maximum:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:** A positive value represents a lead angle, and a negative value represents a lag angle.

3. Enter the period from on-status to off-status of a transistor, in electrical degrees, in the **Trigger Pulse Width** field.
4. Enter the voltage drop across one transistor when the transistor is turned on in the **Transistor Drop** field. Refer to the figures of the different circuit types in step 2.
5. Enter the voltage drop of one diode in the discharge loop in the **Diode Drop** field. If you selected a star-type circuit (**S3** or **S4**) as the **Circuit Type**, enter the total discharge voltage in this field.
6. If you selected CCC (chopped current control) as the **Control Type**, then enter the maximum and minimum current values in the **Maximum Current** and **Minimum Current** fields.
7. Click **OK** to close the **Properties** window.

Circuit Data for Brushless PMDC Motors

To access the **Circuit Data Properties** window, double-click the **Machine** > **Circuit** entry in the Project Manager tree.

Lead Angle of Trigger	The trigger's lead angle, in electrical degrees

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Trigger Pulse Width	The period from on-status to off-status for a transistor, in electrical degrees
Transistor Drop	The voltage drop across one transistor when the transistor is turned on
Diode Drop	The voltage drop across one diode in the discharge loop
Maximum Current	The maximum current for the chopped current control. This field is not available for a DC circuit.
Minimum Current	The minimum current for the chopped current control. This field is not available for a DC circuit.

Defining the Stator Data for a Brushless PMDC Motor

The stator is the outer lamination stack where the polyphase voltage windings reside.

To define the general stator data:

1. To open the **Stator Data Properties** window, double-click the **Machine** > **Stator** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the **Outer Diameter** of the stator.
3. Enter the **Inner Diameter** of the stator.
4. Enter the length of the stator core in the **Length** field.
5. Enter the stacking factor for the stator core in the **Stacking Factor** field.
6. Select a **Steel Type** for the stator core:
    a. Click the button for **Steel Type**.
    b. In the **Select Definition** window that appears, select a steel type from the list, or define a new steel type.
    c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.
7. Enter the **Number of Slots** in the stator.
8. Select the **Slot Type**:
    a. Click the button for the **Slot Type**.
    b. In the **Select Slot Type** window that appears, select a slot type (available types include 1 through 4).

    > **Note:** When you place the mouse cursor over the slot type, a schematic of the selected type appears, displaying the slot dimension variables.

    c. Click **OK** to close the **Select Slot Type** window and return to the **Properties** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. Enter the skew width, measured in slot number, in the **Skew Width** field.
10. Click **OK** to close the **Properties** window.

Stator Data for Brushless PMDC Motors

To access the general stator data, double-click the **Machine** > **Stator** entry in the Project Manager tree.

The **Stator Data Properties** window contains the following fields:

Outer Diameter	The outer diameter of the stator core
Inner Diameter	The inner diameter of the stator core
Length	The length of the stator core
Stacking Factor	The stacking factor of the stator core
Steel Type	The steel type of the stator core. Click the button to open the **Select Definition** window.
Number of Slots	The number of slots the stator core contains
Slot Type	The type of slots in the stator core. Click the button to open the **Select Slot Type** window.
Skew Width	The skew width measured in slot number

Defining the Stator Slots for a Brushless PMDC Motor

To define the physical dimensions of the stator slots:

1. To open the **Stator Slot Data Properties** window, double-click the **Machine** > **Stator** > **Slot** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Optionally, to automatically design the dimensions of slots **Hs2**, **Bs1**, and **Bs2**, select the **Auto Design** check box.
3. Optionally, to design dimensions of slots **Bs1** and **Bs2** based on the stator tooth width, select the **Parallel Tooth** check box, and enter a value in the **Tooth Width** field.
4. Enter the available slot dimensions:

   Hs0 Always available

   Hs2 Available only when **Auto Design** is cleared. When **Auto Design** is selected, this slot dimension is determined automatically.

   Bs0 Always available

   Bs1 Available only when **Auto Design** and **Parallel Tooth** are both cleared. When

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Auto Design is selected, this slot dimension is determined automatically. When **Parallel Tooth** is selected, this slot dimension is determined based on the value entered in the **Tooth Width** field.

Bs2 Available only when **Auto Design** and **Parallel Tooth** are both cleared. When **Auto Design** is selected, this slot dimension is determined automatically. When **Parallel Tooth** is selected, this slot dimension is determined based on the value entered in the **Tooth Width** field.

5. Click **OK** to close the **Properties** window.

Stator Slot Data for Brushless PMDC Motors

To access the stator slot data, double-click the **Machine** > **Stator** > **Slot** entry in the Project Manager tree.

The **Stator Slot Data Properties** window contains the following fields:

Auto Design	Select or clear this to enable or disable auto-design of slots **Hs2**, **Bs1**, and **Bs2**. When this check box is selected, only two other fields appear in the window: **Hs0** and **Bs0**.
Parallel Tooth	Select this to design **Bs1** and **Bs2** based on the tooth width. When this check box is selected, the **Bs1** and **Bs2** fields are removed, and the **Tooth Width** field is added.
Tooth Width	The tooth width for the parallel tooth, on which **Bs1** and **Bs2** are designed
Hs0	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected)
Hs1	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected)
Hs2	A slot dimension (see the diagram shown in the modeling window when **Machine-Stator-Slot** is selected)
Bs0	A slot dimension (see the diagram shown in the modeling window when **Machine > Stator > Slot is selected**)
Bs1	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected)
Bs2	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected).
Rs	A slot dimension. (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected). Rs is added when the slot type is 3 or 4.

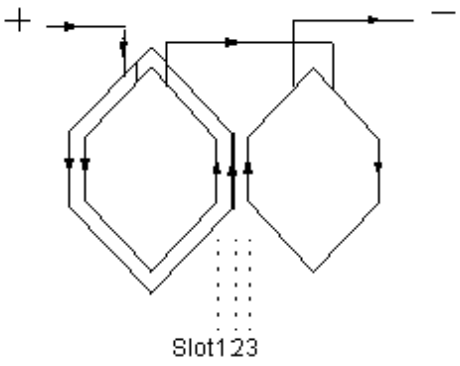
Defining the Stator Windings and Conductors for a Brushless PMDC Motor

To define the stator windings, wires, and conductors:

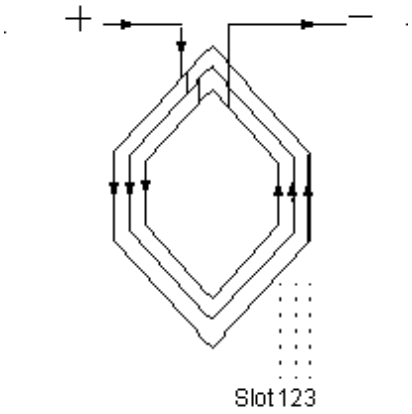
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. To open the **Stator Slot Winding Properties** window, double-click the **Machine** > **Stator** > **Winding** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Click the **Winding** tab.
3. Enter the number of layers in the stator winding in the **Winding Layers** field.
4. Select a **Winding Type**:
   a. Click the button for **Winding Type**.
   b. In the **Winding Type** window that appears, select from one of the following three types of winding:
      - **Whole Coiled**
      - **Half Coiled**
      - **Editor**

   When you place the mouse cursor over a winding button, an outline of the selected winding appears. The following table describes the six types of windings that are possible (three for one-layer and three for two-layer):

Type	Description
Editor	A user-defined one-layer winding arrangement. You need to set up the winding arrangement for each slot. For this winding type, the following letters are used for the phase windings: • Phase A/A return uses A/X. • Phase B/B return uses B/Y. • Phase C/C return uses C/Z.
Whole Coiled	A one-layer whole-coiled winding: 
Half Coiled	A one-layer concentric half-coiled winding:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

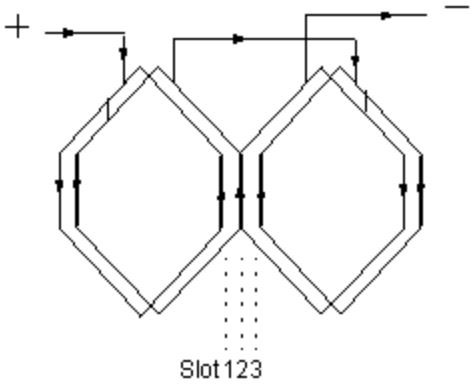


Editor

A user-defined two-layer winding arrangement. When you select **20**, the Winding Editor opens, where you can specify a different winding arrangement for each slot.

A two-layer wave winding:

Whole Coiled

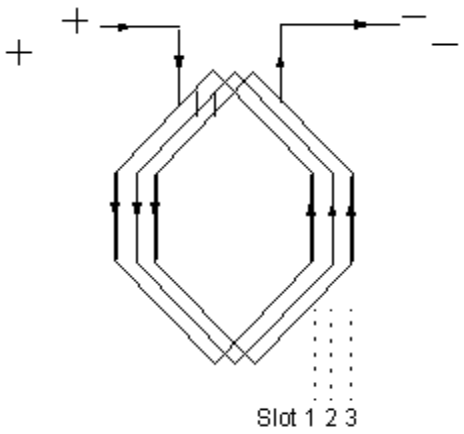


The phase belt for this winding configuration is equal to 360/2*m*, where *m* is the phase number.

Half Coiled

A two-layer half-coiled winding:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



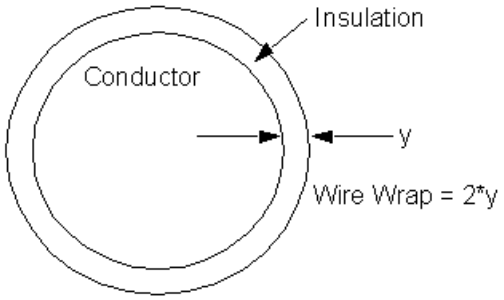
There is only one coil per phase per pair of poles.

> **Note: For a two-layer winding,** if you check Constant Pitch in the Winding Editor, only the top layer needs to be defined; the bottom layer will be determined according to the coil pitch.

c. Once you have clicked a button to select a winding, click **OK** to close the **Winding Type** window and return to the **Properties** window.

5. Select or enter the number of parallel branches in one phase of the winding in the **Parallel Branches** field.
6. Enter the total number of conductors in each stator slot in the **Conductors per Slot** field. This value is the number of turns per coil multiplied by the number of layers.
7. Enter the coil pitch, measured in number of slots, in the **Coil Pitch** field. The coil pitch is the number of slots separating one winding. For example, if a coil starts in slot 1 and ends in slot 6, it has a coil pitch of 5.
8. Enter the number of wires per conductor in the **Number of Strands** field. Enter 0 to have RMxprt auto-design this value.
9. Enter the thickness of the double-sided wire wrap in the **Wire Wrap** field. Enter 0 to automatically obtain this value from the wire library.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



10. Select the **Wire Size**:
    a. Click the button for **Wire Size**.
    b. In the **Wire Size** window that appears, select a value from the **Wire Diameter** drop-down menu.
    c. Select a wire gauge from the **Gauge** drop-down menu. You can select from the following options:

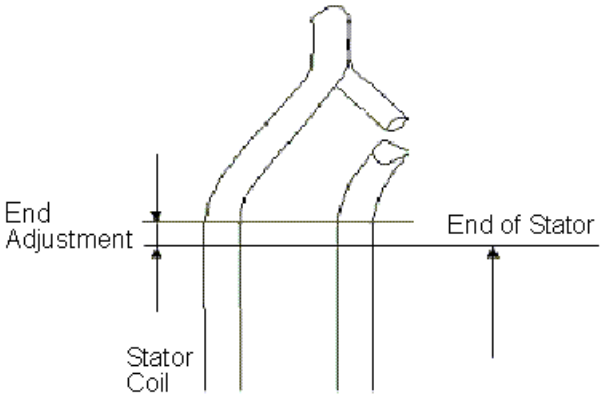
< *number* >	You can select a specific gauge number. When you select a gauge number, the **Wire Diameter** field is automatically updated.
USER	This option allows you to manually enter the **Wire Diameter**. This is useful when you want to enter a diameter that does not correspond to a particular wire gauge.
AUTO	This option sets the **Wire Diameter** to zero, and RMxprt automatically calculates the optimal value. The diameter information is then written to the output file when you analyze the design.
MIXED	This option allows you to define a conductor that is made of different size wires. For example, a single conductor may consist of five wires: three wires with a diameter of 0.21 mm and two wires with a diameter of 0.13 mm.

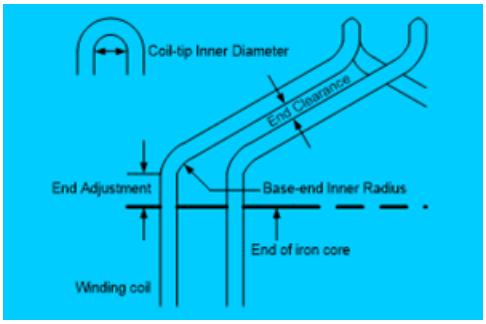
    The gauge number is based on AWG settings. You can create your own wire table using **Machine > Wire**, and then you can select this wire table using **Tools** > **Options** > **General Options** > **Machines**.

11. When you are done setting the wire size, click **OK** to close the **Wire Size** window and return to the **Properties** window.
12. Click the **End/Insulation** tab.
13. Select or clear the **Input Half-turn Length** check box.
14. Do one of the following:
    - If you selected **Input Half-turn Length**, then enter the half-turn length of the armature winding in the Half-Turn Length field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you cleared **Input Half-turn Length**, then enter the end length adjustment of the stator coils in the **End Adjustment** field. The end adjustment is the distance one end of the conductor extends vertically beyond the end of the stator.

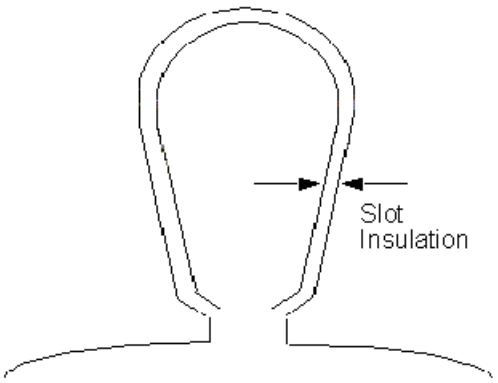




15. Enter the inner radius of the base corner in the **Base Inner Radius** field.
16. Enter the inner diameter of the coil tip in the **Tip Inner Diameter** field.
17. Enter the distance between two stator coils in the **End Clearance** field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

18. Enter the thickness of the slot liner insulation in the **Slot Liner** field.



19. Enter the thickness of the wedge insulation in the **Wedge Thickness** field.
20. Enter the thickness of the insulation layer in the **Layer Insulation** field.
21. Enter the limited slot fill factor for the wire design in the **Limited Fill Factor** field. The slot fill factor is the ratio between the cross-sectional area of all conductors in one slot and the entire slot area.
22. Click **OK** to close the **Properties** window.

Winding Editor for a Brushless DC Motor

For a brushless DC motor, you may want to specify a different number of conductors for each stator slot. The Winding Editor makes this possible by enabling you to specify the number of turns for each coil.

To specify the number of turns for each coil:

1. Click **Machine** > **Winding** > **Edit Layout**.
2. In the **Winding Editor** window that appears, from the table in the upper left, set which phase you want for each coil and which slot is the “in” and “out” slot for the current in each coil.
3. If you are working on a quarter or half model, you may want to specify a multiplier by clicking the **Periodic Multiplier** check box and specifying a value.
4. Select or deselect the **Constant Turns** or **Constant Pitch** check boxes, depending on whether you want to be able to change these setting in the table above. When these options are selected, you cannot change the turns or pitch.
5. When you are satisfied with the conductor settings, click **OK** to close the **Winding Editor** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining Different Size Wires for a Brushless DC Motor

Use the **Gauge** option in the **Wire Size** dialog box if you have a conductor that is made up different size wires.

To define different size wires:

1. In the **Wire Size** window, select **MIXED** from the **Gauge** drop-down menu.
2. Select either **Round** or **Rectangular** as the **Wire Type**.
3. Enter the appropriate wire data in the table:
    - For a round wire:
        - Enter the **Diameter** in the table.
        - Enter a **Number** in the table to specify how many of the conductor's wires have this diameter.
    - For a rectangular wire:
        - Enter the **Width** of the wire in the table.
        - Enter the **Thickness** of the wire in the table.
        - Enter the **Fillet** value in the table.
        - Enter a **Number** in the table to specify how many of the conductor's wires have this data.
4. Click **Add** to add the new wire data.
5. Repeat steps 3 and 4 for each size wire you want to add.
6. When you are finished defining the wires, click **OK** to close the **Wire Size** window and return to the RMxprt **Properties** window.

> **Note:** For example, if one conductor is made up of five wires, and three of those wires have a diameter of 0.21 mm, and the other 2 have a diameter of 0.13 mm, then the mixed wire size table will have two lines. The first line will list Diameter = 0.21 and Number = 3. The second line will list Diameter = 0.13 and Number = 2. An equivalent wire diameter is displayed as Wire Size value in the Winding tab in the Properties window.

Stator Winding Data for Brushless PMDC Motors

To access the stator winding data, double-click the **Machine** > **Stator** > **Winding** entry in the Project Manager tree.

The **Stator Winding Data Properties** window contains the following fields:

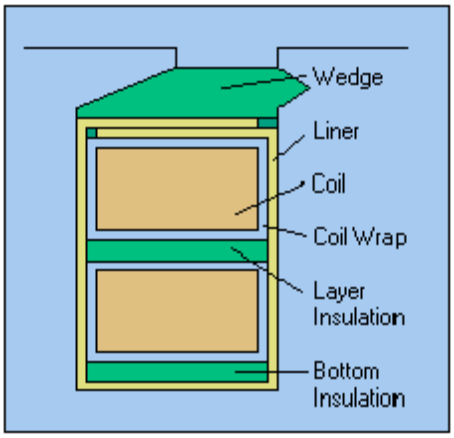
Winding tab	**Winding Layers**	The number of winding layers
	Winding Type	The type of stator winding. Click the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		button to open the **Winding Type** window and choose from **Whole Coiled**, **Half Coiled**, and **Editor**.
	Parallel Branches	The number of parallel branches in the stator winding
	Conductors per Slot	The number of conductors per stator slot (0 for auto-design)
	Coil Pitch	The coil pitch measured in number of slots
	Number of Strands	The number of wires per conductor (0 for auto-design)
	Wire Wrap	The thickness of the double-sided wire wrap (0 to automatically obtain this value from the wire library).
	Wire Size	The diameter of the wire (0 for auto-design). Click the button to open the **Wire Size** window where you can specify units, wire type, diameter, and gauge.
End/Insulation tab	**Input Half-turn Length**	Select or clear this check box to specify whether or not you want to enter the half-turn length. When this check box is selected, the **Half Turn Length** field appears the next time you open the **Properties** window. When this check box is selected, the **End Adjustment** field appears instead.
	Half Turn Length	The half-turn length of the armature winding
	End Adjustment	The end length adjustment of the stator coils, which is the distance one end of the conductor extends vertically beyond the end of the stator.
	Base Inner Radius	The inner radius of the base corner
	Tip Inner Diameter	The inner diameter of the coil tip
	End Clearance	The end clearance between two adjacent coils
	Slot Liner	The thickness of the slot liner insulation

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Wedge Thickness	The thickness of the wedge insulation
Layer Insulation	The thickness of the insulation layer
Limited Fill Factor	The limited slot fill factor for the wire design. The slot fill factor is the ratio between the cross-sectional area of all conductors in one slot and the entire slot area.



Defining the Rotor Data for a Brushless PMDC Motor

The rotor consists of copper bars in which current is induced by the magnetic fields produced by the stator windings. In the project tree, double-click **Machine** > **Rotor** and **Machine** > **Rotor** > **Pole** to define the rotor and the pole.

To define general rotor data:

1. To open the **Rotor Data Properties** window, double-click the **Machine** > **Rotor** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the outer diameter of the rotor in the **Outer Diameter** field.
3. Enter the inner diameter of the rotor in the **Inner Diameter** field.
4. Enter the length of the rotor core in the **Length** field.
5. Select a **Steel Type** for the rotor core:
    a. Click the button for **Steel Type**.
    b. In the **Select Definition** window that appears, select a steel type from the list, or define a new steel type.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.

6. Enter the stacking factor for the rotor core in the **Stacking Factor** field.
7. Select a **Pole Type**:
   a. Click the button.
   b. In the **Select Pole Type** window that appears, click a button to select the desired pole type (1, 2, 3, 4, or 5).

      > **Note:** When you place the mouse cursor over a pole type, an outline of the selected circuit type appears.

   c. Click **OK** to close the **Select Pole Type** window and return to the **Properties** window.
8. Click **OK** to close the **Properties** window.

Rotor Data for Brushless PMDC Motors

To access the general rotor data, double-click the **Machine** > **Rotor** entry in the Project Manager tree.

The **Rotor Data Properties** window contains the following fields:

Outer Diameter	The outer diameter of the rotor core
Inner Diameter	The inner diameter of the rotor core
Length	The length of the rotor core
Steel Type	The steel type of the rotor core. Click the button to open the **Select Definition** window.
Stacking Factor	The stacking factor of the rotor core
Pole Type	The pole type for the rotor. Click this button to open the **Select Pole Type** window and select from the following types: **1**, **2**, **3**, **4**, **5**.

Defining the Rotor Pole for a Brushless PMDC Motor

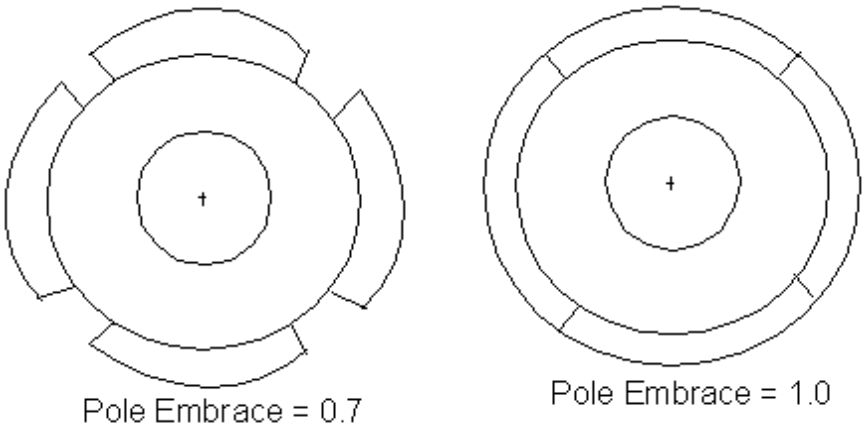
The rotor pole drives the electromagnetic field, which is coupled with the stator windings. Use the **Rotor Pole Data Properties** window to define the rotor pole.

> **Note:** Some of the fields in the **Rotor Pole** window change, or are inactive, depending on the **Rotor Type** you select.

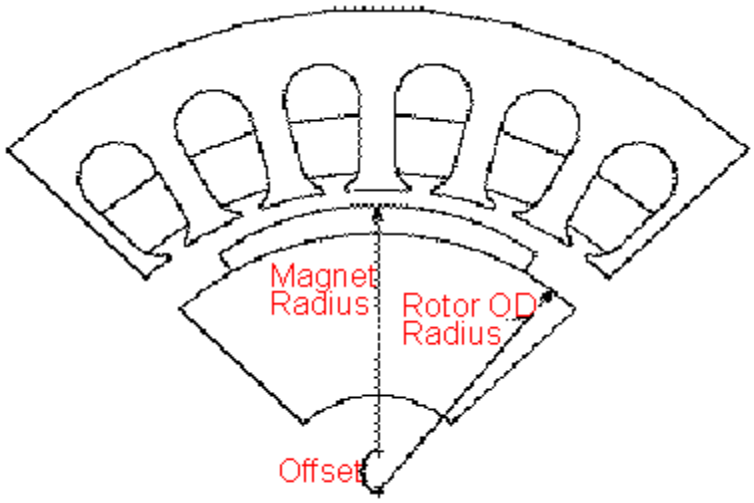
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To define the rotor pole:

1. To open the **Rotor Pole Data Properties** window, double-click the **Machine** > **Rotor** > **Pole** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. For all pole types except type **4**, enter the ratio of the actual arc distance in relation to the maximum possible arc distance in the **Embrace** field. This value is between 0 and 1.



3. For pole type **4**, enter the shaft diameter of the rotor in the **Shaft Diameter** field.
4. For pole types **1**, **2**, and **3**, enter the distance from the center of the rotor to the polar arc center in the **Offset** field. Enter 0 for a uniform air gap.



5. For pole type **5**, enter the thickness of the bridge across the two poles in the **Bridge** field.
6. For pole type **5**, enter the width of the rib supporting the bridge in the **Rib** field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Select the type of magnet to use in the rotor pole from the **Magnet Type** drop-down menu.
8. For pole types **4** and **5**, enter the width of the magnet in the **Magnet Width** field.
9. Enter the maximum radial thickness of the magnet in the **Magnet Thickness** field.
10. Click **OK** to close the **Properties** window.

Rotor Pole Data for Brushless PMDC Motors

To access the pole rotor data, double-click the **Machine** > **Rotor** > **Pole** entry in the Project Manager tree.

The **Rotor Pole Data Properties** window may contain the following fields, depending on the pole type specified.

Embrace	The pole embrace; for pole types **1**, **2**, **3**, and **5**
Shaft Diameter	The shaft diameter of the rotor; for pole type **4**
Offset	The pole-arc center offset from the rotor center (0 for a uniform air gap); for pole types **1**, **2**, and **3**
Bridge	The thickness of the bridge across two adjacent poles; for pole type **5**
Rib	The width of the rib at the center of two adjacent poles that support the bridge; for pole type **5**
Magnet Type	The type of magnet. Click the button to open the **Select Definition** window; for all pole types.
Magnet Width	The maximum width of the magnet; for pole types **4** and **5**
Magnet Thickness	The maximum thickness of the magnet; for all pole types

Defining the Shaft Data for a Brushless PMDC Motor

To define the shaft:

1. To open the **Shaft Data Properties** window, double-click the **Machine** > **Shaft** entry in the Project Manager tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Select or clear the **Magnetic Shaft** check box to specify whether or not the shaft is to be made of magnetic material.
3. Click **OK** to close the **Properties** window.

Shaft Data for Brushless PMDC Motors

To access the shaft data, double-click the **Machine** > **Shaft** entry in the Project Manager tree.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Shaft Data Properties** window contains the following fields:

Magnetic Shaft	Select or clear this check box to indicate whether or not the shaft is made of magnetic material.

Setting Up Analysis Parameters for a Brushless PMDC Motor

To define the solution data:

1. To open the Solution Setup window, right-click **Analysis** in the project tree, and click **Add Solution Setup**.
2. Click the **General** tab. The **Operation Type** is **Motor** for this machine type.
3. Select the **Load Type** used in the motor from the following options:

Const Speed	The speed remains constant in the motor.
Const Power	The output power remains constant in the motor.
Const Torque	The torque remains constant regardless of the speed. In this case, $T_{load} = T_{rated}$, given by the output power divided by the given rated speed.
Linear Torque	The torque increases linearly with speed. In this case, $T_{load} = T_{rated} * (n/n_{rated})$ where T_{rated} is given by the output power divided by the given rated speed.
Fan Load	The load varies nonlinearly with speed. In this case, $T_{load} = T_{rated} * (n/n_{rated})^2$ where T_{rated} is given by the output power divided by the given rated speed.

4. Enter the output power developed at the shaft of the motor in the **Rated Output Power** field.
5. Enter the RMS line-to-line voltage in the **Rated Voltage** field.
6. Enter the desired output speed of the motor at the load point in the **Rated Speed** field.
7. Enter the temperature at which the system functions in the **Operating Temperature** field.
8. Click **OK** to close the **Solution Setup** window.

Analyses Offered

- **Adapted to both Synchronous Motor and Generator**: The structures of the salient-pole synchronous motor and the generator are basically the same, but their phasor relationships and the computation methods are slightly different, their output characteristics data are also different. Therefore, RMxprt divides the synchronous machine into two design modules: Synchronous Motor and Synchronous Generator.
- **Auto Arrangement of Three-phase Windings**: Almost all commonly used three-phase single- and double-layer, half- and whole-type ac windings (including fractional-pitch windings) can be automatically arranged. Users do not need to define coils one by one.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RMxprt also supports a double-layer winding with half-turn coils, which are auto-arranged in the order of even, odd, even, odd, …, and even, odd, as long as it is physically possible.

When a designer adopts single-layer whole-coiled windings, RMxprt will perform winding arrangement optimization to minimize the average coil pitch. When asymmetric three-phase windings are used, winding arrangement is optimized in such a way that minimum negative-sequence and zero-sequence components are achieved.

- **Winding Editor Supporting Any Single- and Double-Layer Windings**: Besides taking advantage of the winding auto-arrangement function in RMxprt, users can also specify any special winding by using of the Winding Editor function.

  In the Winding Editor, through modification of phase belonging, number of turns, in-slot and out-slot number of each coil, it is possible to design single- and double-layer winding arrangements for any purposes.

- **Analyze Air-Gap Magnetic Field Distribution**: For both uniform and non-uniform air gaps, Schwarz-Christopher Transformation is adopted to solve for the air-gap magnetic field distribution.
- **Analyze EMF Waveform and Total Harmonic Distortion (THD)**: Based on the analysis of the air-gap magnetic field waveform, taking into account coil short pitch, winding distribution, skew slot, winding connection, load effects and other factors, the emf waveforms in the coils and the windings are analyzed to solve for the emf distortion factors.
- **Analyze Dynamic Parameters of Damping Winding**: Different from the squirrel-cage winding of the induction machine, the damping winding of the salient-pole synchronous machine is located in the surface of magnetic field poles, which deviates greatly along the d- and the q-axes. Furthermore, the connection of damping bars has several forms. The bars under each pole could be connected, but not connected with those under other poles. All the bars could be connected together. The bars could be connected through end-plate. RMxprt can deal with all those complicated situations and give the dynamic parameters for the damping winding.

Solution Data for Brushless PMDC Motors

To access the solution data, right-click **Analysis** in the project tree, and click **Add Solution Setup**. For this machine type, there is only one tab, the **General** tab.

The **Solution Setup** window contains the following fields:

Operation Type	The operation type is automatically set to **Motor** for this machine type.
Load Type	Select from **Const Speed**, **Const Power**, **Const Torque**, **Linear Torque**, and **Fan Load**. The default is **Const Power**.
Rated Output Power	Type a value for the rated output voltage, and select the units.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rated Voltage	Type a value for the rated voltage, and select the units.
Rated Speed	Type a value for the rated speed, and select the units.
Operating Temperature	Type a value for the operating temperature, and select the units.

Switched Reluctance Motors

After you have selected **Switched Reluctance Motors** as your model type, define the following:

- General data, such as the power, voltage, and speed of the motor
- Circuit data
- Stator core data, such as the number of poles, diameter, and yoke thickness
- Stator coil data, such as the slot liner thickness, number of parallel branches, and number of wires in each conductor
- Rotor core data, such as the air gap dimensions and number of poles in the rotor
- Shaft data
- Optional machine housing
- Solution data

Analysis Approach for Switched Reluctance Motors

This motor type operates with shaft position feedback to synchronize the commutation of the phase currents with precise rotor position. Typically, both the stator and the rotor are salient to increase the torque-producing characteristics of the motor. The rotor has no windings; the torque is produced by the alignment tendency of the rotor to the stator so that the stator flux linkage is maximized.

In these motors, the stator and rotor have different numbers of poles. The stator phase windings are energized at precise moments synchronized with the position of the rotor. The task of energizing the stator windings is performed by a complex electronic system.

The number of phases in the winding is the ratio of the stator number of poles to the smallest common divider of the stator and the rotor number of poles.

In switched reluctance motors (SRM), the stator and the rotor have a different number of poles, and the stator currents are commutated exactly according to rotor position. The signal of the rotor position is obtained from a position sensor. The stator windings are triggered one by one, and normally the current in a winding has finished or almost finished freewheeling when the next winding is triggered. Therefore, the mutual effects between two phases can be neglected. The voltage equation of one phase is

$$u = u_T + R_S \cdot i + \frac{d\Psi(\theta,i)}{dt}$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where u_T is the transistor or diode voltage drop, and R_s is the stator winding resistance. $\Psi(\theta, i)$ is the flux linkage of the winding at rotor position θ and winding current i, as is shown in Figure 8, where the rotor position when the center of the rotor slot is aligned to the winding axis is defined as 0.

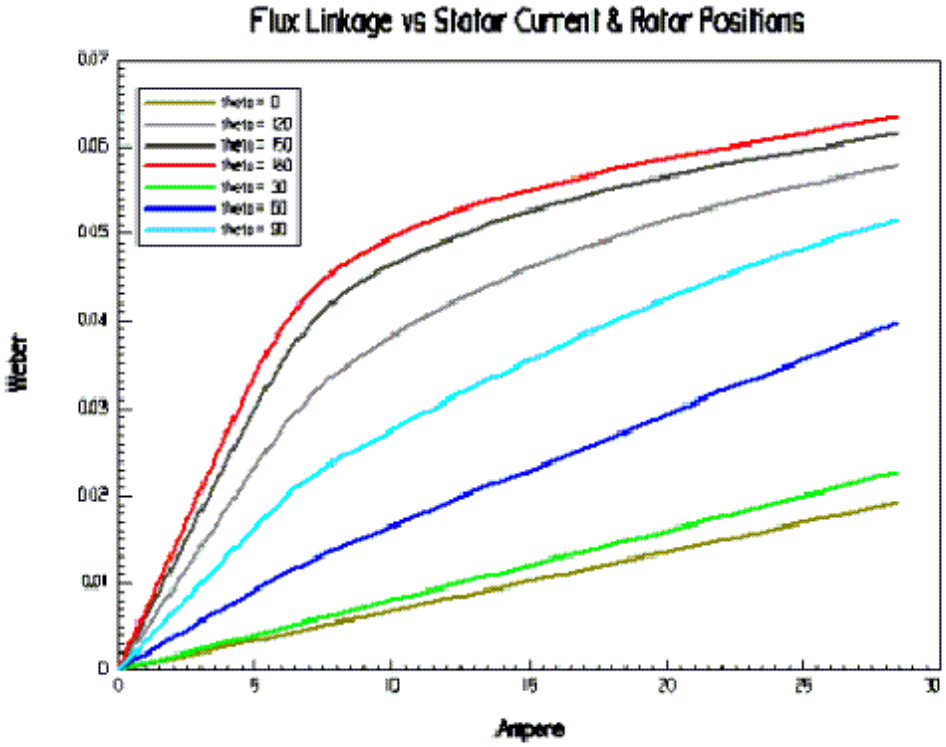


Figure 8

Let

$$\tilde{L}_{?} = \frac{\partial \Psi(\theta, i)}{\partial i}$$

and

$$G = \frac{\partial(\Psi / i)}{\partial \theta} = \frac{\partial \tilde{L}_{?}}{\partial \theta}$$

Then

$$u = u_T + R_S \cdot i + \tilde{L}_{?} p i + G \omega_e i$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where ω_e is the rotor speed in electrical rad/s, and p is the differential operator as given by

$$p = \frac{\mathrm{d}}{\mathrm{d}t}$$

The instant electromagnetic torque t_2 is

$$t_2 = \frac{1}{2} Gi^2$$

The input electric power is computed from voltage and current as

$$P_1 = \frac{1}{T}\int_0^T (u \cdot i \cdot dt)$$

The output mechanical power is

$$P_2 = P_1 - \left(P_{fw} + P_{Cua} + P_t + P_{Fe}\right)$$

where P_{fw}, P_{Cua}, P_t, and P_{Fe} are frictional and wind loss, armature copper loss, transistor/diode loss, and iron core loss, respectively.

The average output mechanical shaft torque T_2 is

$$T_2 = \frac{P_2}{\omega}$$

where ω is the rotor angular speed in mechanical rad/s.

The efficiency of the electric machine is computed by

$$\eta = \frac{P_2}{P_1} \times 100\%$$

Defining a Switched Reluctance Motor

The general procedure for defining a switched reluctance motor is as follows:

1. Insert a Switched Reluctance motor into a new or existing project.
2. Double-click the **Machine** entry in the project tree to define the general data.
3. Double-click the **Machine** > **Circuit** entry in the project tree to define the control circuit.
4. Double-click the **Machine** > **Stator** entry in the project tree to define the stator geometry.
5. Double-click the **Machine** > **Stator** > **Winding** entry in the project tree to define the stator windings and conductors.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Double-click the **Machine** > **Rotor** entry in the project tree to define the rotor geometry.
7. Double-click the **Machine** > **Shaft** entry in the project tree to define the magnetism of the shaft.
8. Optionally, right-click the **Machine** entry in the project tree and click **Insert Housing** to add a machine housing.
9. Right-click **Analysis** in the project tree, and click **Add Solution Setup** to define the solution data.
10. Choose **File** > **Save** to save the project.
11. Choose **RMxprt** > **Analyze** to analyze the design.

> **Note:** When you place the cursor over an entry field in the data windows, a brief description of that field appears in the status bar at the bottom of the **RMxprt** window.

Once analyzed, the model can be viewed in the Maxwell 2D Modeler, or it can be used to create a new Maxwell 2D project, and a new Maxwell 3D project.

For a specific example, see the *Switched Reluctance Motor Problem* application note, on the technical support page of the Ansys web site.

Defining the General Data for a Switched Reluctance Motor

Use the **General** window to define the power settings, speed, and period of the motor.

To define the general data:

1. To open the **General Data Properties** window, double-click the **Machine** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the energy loss due to friction at the given speed in the **Frictional Loss** field.
3. Enter the wind loss due to air resistance measured at the reference speed in the **Wind Loss** field.
4. Enter the given speed in the **Reference Speed** field.
5. Select **DC** or **CCC** from the **Control Type** drop-down menu.
6. Select a **Circuit Type** from the following types:
   - **Full-Voltage**
   - **Half-Voltage**
   - **Coupled-Coil**

   The circuit types are based on industry standards. By default, type **Full-Voltage**, is selected as the circuit type.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** When you place the mouse cursor over a circuit type, an outline schematic of the circuit appears.

7. Click **OK** to close the **Properties** window.

General Data for Switched Reluctance Motors

To access the general data, double-click the **Machine** entry in the Project Manager tree.

The **General Data Properties** window for a three-phase induction motor contains the following fields:

Machine Type	The machine type you selected when inserting a new RMxprt design (Switched Reluctance Motor)
Frictional Loss	The frictional energy loss (due to friction) measured at the reference speed
Wind Loss	The wind loss (due to air resistance) measured at the reference speed
Reference Speed	The given speed of reference
Control Type	The way the circuit is controlled. Select from **DC** or **CCC** (chopped current control, which forces the current to fall between the minimum and maximum values specified).
Circuit Type	The drive circuit type. Click the button to open the **Circuit Type** window and select from the following three types: • **Full-Voltage** • **Half-Voltage** • **Coupled-Coil**

Defining the Circuit Data for a Switched Reluctance Motor

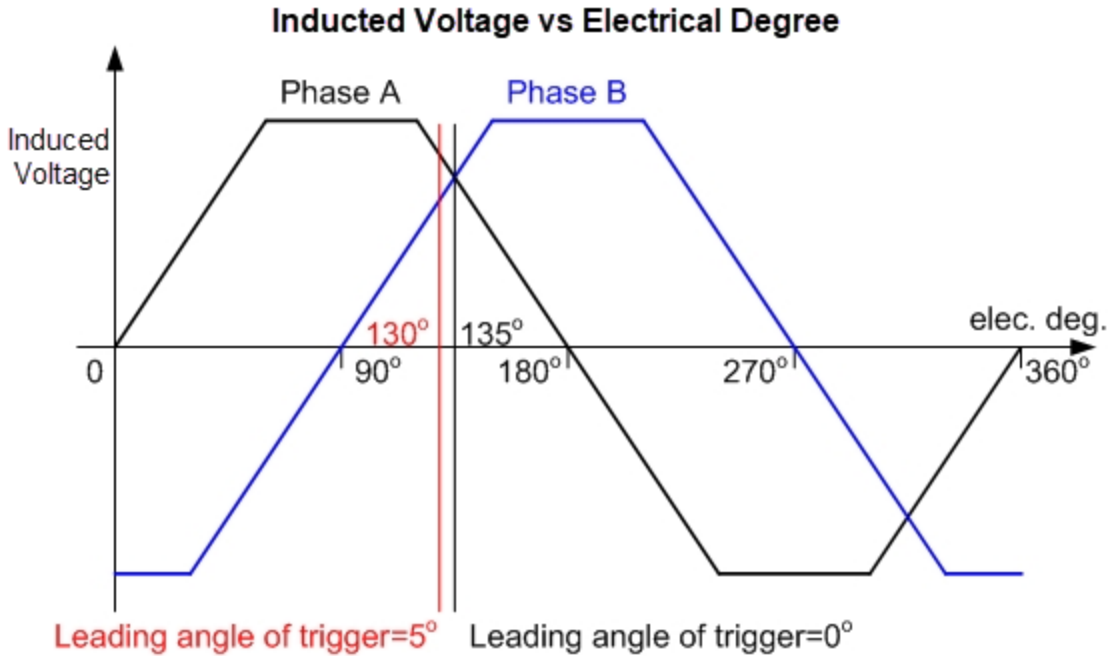
Use the **Circuit Data Properties** window to specify the rated output power, voltage values, circuit type, and speed of the brushless DC motor.

To define the general data:

1. To open the **Circuit Data Properties** window, double-click the **Machine** > **Circuit** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the trigger's lead angle in electrical degrees in the **Lead Angle of Trigger** field. The trigger angle is the point at which the magnetic poles interact to begin the motion of the motor. An angle of 0 means that each phase is triggered when its axis is aligned with the rotor slot center. The trigger's lead angle is shown in the following plot of the open circuit

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

induced voltage versus position. An angle of 0 means that the induced voltage in the triggered phase is at a maximum:



/>

> **Note:** A positive value represents a lead angle, and a negative value represents a lag angle.

3. Enter the period from on-status to off-status of a transistor, in electrical degrees, in the **Trigger Pulse Width** field. The trigger pulse width is the width of the energizing pulse applied to the winding, or the period for an ‘on’ status of the transistors. The maximum ‘on’ period is given by 180 degrees plus the value for the lead angle of trigger.
4. Enter the voltage drop across one transistor when the transistor is turned on in the **Transistor Drop** field. Refer to the figures of the different circuit types in step 2. This value is over one conduction path when the transistors are triggered.
5. Enter the voltage drop on all anti-parallel diodes in the discharge path in the **Diode Drop** field. If you selected a star-type circuit (**S3** or **S4**) as the **Circuit Type**, enter the total discharge voltage in this field.
6. If you selected CCC (chopped current control) as the **Control Type**, then enter the maximum and minimum current values in the **Maximum Current** and **Minimum Current** fields.
7. Click **OK** to close the **Properties** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Circuit Data for Switched Reluctance Motors

To access the **Circuit Data Properties** window, double-click the **Machine** > **Circuit** entry in the project tree. When **AC** is selected at the **Control Type**, now circuit data properties exist.

Lead Angle of Trigger	The trigger's lead angle, in electrical degrees
Trigger Pulse Width	The period from on-status to off-status for a transistor, in electrical degrees
Transistor Drop	The voltage drop across one transistor when the transistor is turned on
Diode Drop	The voltage drop across one diode in the discharge loop
Maximum Current	The maximum current for the chopped current control. This field is not available for a DC circuit.
Minimum Current	The minimum current for the chopped current control. This field is not available for a DC circuit.

Defining the Stator Data for a Switched Reluctance Motor

The stator is the outer lamination stack where the polyphase voltage windings reside.

To define the general stator data:

1. To open the **Stator Data Properties** window, double-click the **Machine** > **Stator** entry in the Project Manager tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the **Outer Diameter** of the stator.
3. Enter the **Inner Diameter** of the stator.
4. Enter the total length of the stator core in the **Length** field.
5. Enter the effective magnetic length of the core in the **Stacking Factor** field. This value typically ranges from between 0.93 and 1.0, and is defined as the total length minus the total lamination insulation, divided by the total length.
6. Select a **Steel Type** for the stator core:
    a. Click the button for **Steel Type**.
    b. In the **Select Definition** window that appears, select a steel type from the list, or define a new steel type.
    c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.
7. Enter the number of poles the stator core contains in the **Number of Poles** field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Enter the pole embrace in the **Embrace** field. The pole embrace is the ratio of the actual pole arc angle to the maximum possible pole angle in the field. This value ranges from between 0 and 1.
9. Enter the thickness of the stator coil yoke in the **Yoke Thickness** field.
10. Click **OK** to close the **Properties** window.

Stator Data for Switched Reluctance Motors

To access the general stator data, double-click the **Machine** > **Stator** entry in the Project Manager tree.

The **Stator Data Properties** window contains the following fields:

Outer Diameter	The outer diameter of the stator core
Inner Diameter	The inner diameter of the stator core
Length	The length of the stator core
Stacking Factor	The stacking factor of the stator core
Steel Type	The steel type of the stator core. Click the button to open the **Select Definition** window
Number of Poles	The number of poles in the stator core
Embrace	The stator pole embrace
Yoke Thickness	The thickness of the yoke at the stator core

Defining the Stator Winding Data for a Switched Reluctance Motor

The stator coils provide the excitation for the rotating magnetic poles.

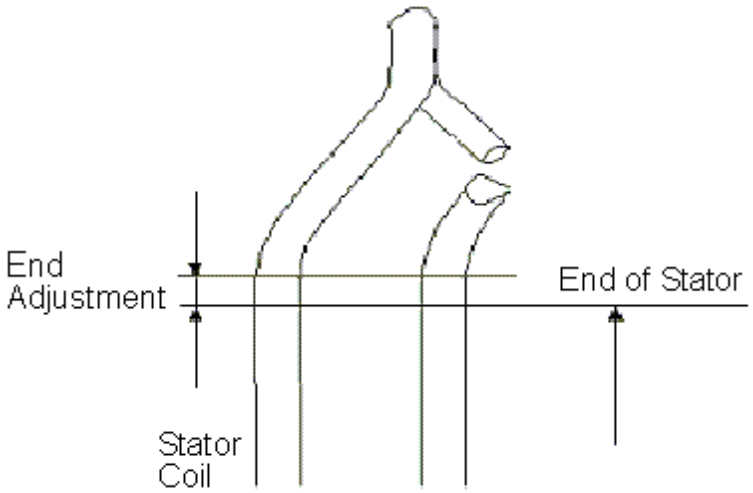
Use the **Stator Coil** window to define the parallel branches, wire specifications, and slot liner for the stator coil.

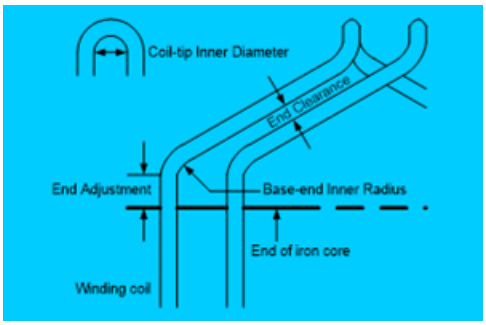
To define the stator coils:

1. To open the **Stator Slot Winding Properties** window, double-click the **Machine** > **Stator** > **Winding** entry in the project tree on the desktop. You can also enter values in the **Properties** section of the desktop without opening a separate window.
2. Enter the thickness of the insulation between the stator core and the field winding in the **Insulation Thickness** field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Enter the end length adjustment of the stator coils in the **End Adjustment** field. The end adjustment is the distance one end of the conductor extends vertically beyond the end of the stator.

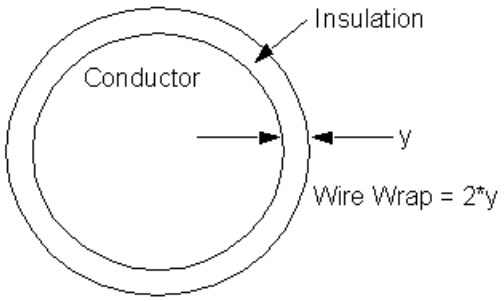




4. Select or enter the number of parallel branches in one phase of the winding in the **Parallel Branches** field.
5. Enter the number of turns per stator pole in the **Turns per Pole** field.
6. Enter the number of wires per conductor in the **Number of Strands** field. Enter 0 to have RMxprt auto-design this value.
7. Enter the inner radius of the base corner in the **Base Inner Radius** field.
8. Enter the inner diameter of the coil tip in the **Tip Inner Diameter** field.
9. Enter the distance between two stator coils in the **End Clearance** field.
10. Enter the thickness of the slot liner insulation in the **Slot Liner** field.
11. Enter the thickness of the wedge insulation in the **Wedge Thickness** field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12. Enter the thickness of the insulation layer in the **Layer Insulation** field.
13. Enter the limited slot fill factor for the wire design in the **Limited Fill Factor** field. The slot fill factor is the ratio between the cross-sectional area of all conductors in one slot and the entire slot area.
14. Enter the thickness of the double-sided wire wrap in the **Wire Wrap** field. Enter 0 to automatically obtain this value from the wire library.



15. Select the **Wire Size**:
    a. Click the button for **Wire Size**.
    b. In the **Wire Size** window that appears, select a value from the **Wire Diameter** drop-down menu.
    c. Select a wire gauge from the **Gauge** drop-down menu. You can select from the following options:

<number>	You can select a specific gauge number. When you select a gauge number, the **Wire Diameter** field is automatically updated.
USER	This option allows you to manually enter the **Wire Diameter**. This is useful when you want to enter a diameter that does not correspond to a particular wire gauge.
AUTO	This option sets the **Wire Diameter** to zero, and RMxprt automatically calculates the optimal value. The diameter information is then written to the output file when you analyze the design.
MIXED	This option allows you to define a conductor that is made of different size wires. For example, a single conductor may consist of five wires: three wires with a diameter of 0.21 mm and two wires with a diameter of 0.13 mm.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The gauge number is based on AWG settings. You can create your own wire table using **Machine > Wire**, and then you can select this wire table using **Tools** > **Options** > **General Options** > **Machines**.

d. When you are done setting the wire size, click **OK** to close the **Wire Size** window and return to the **Properties** window.
e. Enter the conductor area ratio of the coupled circuit to the main circuit in the **Coupled Ratio** field.
f. Click **OK** to close the **Properties** window.

Defining Different Size Wires for a Switched Reluctance Motor

Use the **Gauge** option in the **Wire Size** window if you have a conductor that is made up different size wires.

To define different size wires:

1. In the **Wire Size** window, select **MIXED** from the **Gauge** drop-down list.
2. Select either **Round** or **Rectangular** as the **Wire Type**.
3. Enter the appropriate wire data in the table:
    - For a round wire:
        - Enter the **Diameter** in the table.
        - Enter a **Number** in the table to specify how many of the conductor's wires have this diameter.
    - For a rectangular wire:
        - Enter the **Width** of the wire in the table.
        - Enter the **Thickness** of the wire in the table.
        - Enter the **Fillet** value in the table.
        - Enter a **Number** in the table to specify how many of the conductor's wires have this data.
4. Click **Add** to add the new wire data.
5. Repeat steps 3 and 4 for each size wire you want to add.
6. When you are finished defining the wires, click **OK** to close the **Wire Size** window and return to the RMxprt **Properties** window.

> **Note:** For example, if one conductor is made up of five wires, and three of those wires have a diameter of 0.21 mm, and the other 2 have a diameter of 0.13 mm, then the mixed wire size table will have two lines. The first line will list Diameter = 0.21 and Number = 3. The second line will list Diameter = 0.13 and Number = 2. An equivalent wire diameter is displayed as Wire Size value in the Winding tab in the Properties window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stator Winding Data for Switched Reluctance Motors

To access the stator winding data, double-click the **Machine** > **Stator** > **Winding** entry in the Project Manager tree.

The **Stator Winding Data Properties** window contains the following fields:

Insulation Thickness	Thickness of the insulation between the stator core and the field winding
End Adjustment	End length adjustment of the stator coils, which is the distance one end of the conductor extends vertically beyond the end of the stator
Parallel Branches	Number of parallel branches in the stator winding
Turns per Pole	Number of turns per stator pole (0 for auto-design)
Number of Strands	Number of wires per conductor (0 for auto-design)
Wire Wrap	Thickness of the double-sided wire wrap (0 to automatically obtain this value from the wire library)
Wire Size	Diameter of the wire (0 for auto-design); click the button to open the **Wire Size** window where you can specify units, wire type, diameter, and gauge.
Coupled Ratio	Conductor area ratio of the coupled circuit to the main circuit

Defining the Rotor Data for a Switched Reluctance Motor

The rotor core channels the flux generated by stator windings and provides shaft torque. The rotor consists of copper bars in which current is induced by the magnetic fields produced by the stator windings. Use the **Rotor Data Properties** window to define the air gaps, rotor dimensions, and type of steel used in the rotor core. In the Project Manager tree, double-click **Machine** > **Rotor** to define the rotor.

To define general rotor data:

1. To open the **Rotor Data Properties** window, double-click the **Machine** > **Rotor** entry in the Project Manager tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the outer diameter of the rotor in the **Outer Diameter** field.
3. Enter the inner diameter of the rotor in the **Inner Diameter** field.
4. Enter the length of the rotor core in the **Length** field.
5. Select a **Steel Type** for the rotor core:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Click the button for **Steel Type**.
b. In the **Select Definition** window that appears, select a steel type from the list, or define a new steel type.
c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.

6. Enter the effective magnetic length of the core in the **Stacking Factor** field. This value ranges from 0 to 1, and is defined as the total length minus the total lamination insulation, divided by the total length.
7. Enter the number of poles the rotor core contains in the **Number of Poles** field.
8. Enter the ratio of the actual pole angle in relation to the maximum possible pole angle in the **Embrace** field. The value ranges from 0 to 1.
9. Enter the thickness of the rotor yoke in the **Yoke Thickness** field.
10. Click **OK** to close the **Properties** window.

Rotor Data for Switched Reluctance Motors

To access the general rotor data, double-click the **Machine** > **Rotor** entry in the Project Manager tree.

The **Rotor Data Properties** window contains the following fields:

Outer Diameter	Outer diameter of the rotor core
Inner Diameter	Inner diameter of the rotor core
Length	Length of the rotor core
Steel Type	Steel type of the rotor core; click the button to open the **Select Definition** window.
Stacking Factor	Stacking factor of the rotor core
Number of Poles	Number of poles the rotor core contains
Embrace	Rotor pole embrace
Yoke Thickness	Thickness of the rotor core yoke

Defining the Shaft Data for a Switched Reluctance Motor

To define the shaft:

1. To open the **Shaft Data Properties** window, double-click the **Machine** > **Shaft** entry in the Project Manager tree on the desktop. (You can also enter values in the **Properties**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

section of the desktop without opening a separate window.)

2. Select or clear the **Magnetic Shaft** check box to specify whether or not the shaft is to be made of magnetic material.
3. Click **OK** to close the **Properties** window.

Shaft Data for Switched Reluctance Motors

To access the shaft data, double-click the **Machine** > **Shaft** entry in the Project Manager tree.

The **Shaft Data Properties** window contains the following fields:

Magnetic Shaft	Select or clear this check box to indicate whether or not the shaft is made of magnetic material.

Setting Up Analysis Parameters for a Switched Reluctance Motor

To define the solution data:

1. To open the Solution Setup window, right-click **Analysis** in the project tree, and click **Add Solution Setup**.
2. Click the **General** tab. The **Operation Type** is automatically set to **Motor** for this machine type.
3. Select the **Load Type** used in the motor from the following options:

Const Speed	The speed remains constant in the motor.
Const Power	The output power remains constant in the motor.
Const Torque	The torque remains constant regardless of the speed. In this case, $T_{load} = T_{rated}$, given by the output power divided by the given rated speed.
Linear Torque	The torque increases linearly with speed. In this case, $T_{load} = T_{rated} * (n/n_{rated})$ where T_{rated} is given by the output power divided by the given rated speed.
Fan Load	The load varies nonlinearly with speed. In this case, $T_{load} = T_{rated} * (n/n_{rated})^2$ where T_{rated} is given by the output power divided by the given rated speed.

4. Enter the output power developed at the shaft of the motor in the **Rated Output Power** field.
5. Enter the RMS line-to-line voltage in the **Rated Voltage** field.
6. Enter the desired output speed of the motor at the load point in the **Rated Speed** field.
7. Enter the temperature at which the system functions in the **Operating Temperature** field.
8. Click **OK** to close the **Solution Setup** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution Data for Switched Reluctance Motors

To access the solution data, right-click **Analysis** in the Project Manager tree, and click **Add Solution Setup**. For this machine type, there is only one tab, the **General** tab.

The **Solution Setup** window contains the following fields:

Operation Type	The operation type is automatically set to **Motor** for this machine type.
Load Type	Select from **Const Speed**, **Const Power**, **Const Torque**, **Linear Torque**, and **Fan Load**. The default is **Const Power**.
Rated Output Power	Type a value for the rated output voltage, and select the units.
Rated Voltage	Type a value for the rated voltage, and select the units.
Rated Speed	Type a value for the rated speed, and select the units.
Operating Temperature	Type a value for the operating temperature, and select the units.

Line-Start Permanent-Magnet Synchronous Motors

Once you have selected **Line-Start Permanent-Magnet Synchronous Motors** as your motor type, you can define the following:

- General data, such as the frequency, winding connection, number of poles, and voltage
- Stator data, such as the slot type and dimensions, stator diameter, and winding data
- Rotor pole data, such as its associated dimensions, stacking factor, and magnet type
- Shaft data
- Solution data

By option, you can

- add a machine housing
- from a stator
- from a rotor

Analysis Approach for Line-Start PM Synchronous Motors

Synchronous motors use a three-phase sinusoidal voltage source to induce a rotating magnetic field in the stator. Applying this three-phase sinusoidal voltage source to the stator winding of a synchronous motor yields the rotational magnetic field in the air gap. The permanent magnet poles mounted on the rotor try to align in this rotating field, producing a synchronous torque on

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the rotor. Upon starting, the damping winding on the rotor generates the asynchronous starting torque, creating a self-starting feature.

The phasor diagram for the line-start permanent-magnet synchronous motor (LSSM) in the frequency domain is shown in Figure 6.

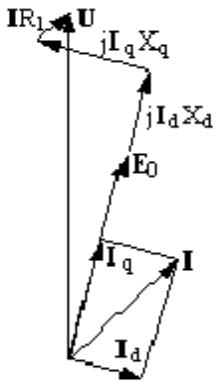


Figure 6

In Figure 6, R_1, X_d, and X_q are armature resistance, d-axis synchronous reactance, and q-axis synchronous reactance, respectively. X_d is the sum of leakage reactance, X_1 and d-axis armature reactance X_{ad}, and X_q is the sum of X_1 and q-axis armature reactance X_{aq}:

$$X_d = X_1 + X_{ad}$$

$$X_q = X_1 + X_{aq}$$

For a given torque angle θ, the angle that $\mathbf{E}_0$ lags **U**:

$$I_d X_d + I_q R_1 = U cos\theta - E_0$$

$$-I_d R_1 + I_q X_q = U sin\theta$$

Solving for I_d and I_q yields

$$I_d = \frac{X_q(U cos\theta - E_0) - R_1 U sin\theta}{R_1^2 + X_d X_q}$$

$$I_q = \frac{R_1(U cos\theta - E_0) - X_d U sin\theta}{R_1^2 + X_d X_q}$$

The angle that **I** lags $\mathbf{E}_0$ is

$$\psi = \tanh \frac{I_d}{I_q}$$

The power factor angle (or torque angle) that **I** lags **U**, is

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\varphi = \psi \pm \theta$$

The input power (electric power) can now be computed from voltage and current as

$$P_1 = 3UIcos\varphi$$

The output power (mechanical power) is

$$P_2 = P_1 - \left(P_{fw} + P_{Cua} + P_{Fe}\right)$$

where P_{fw}, P_{Cu}, and P_{Fe} are frictional and wind loss, armature copper loss, and iron-core loss, respectively.

The output mechanical power (torque) T_2 is

$$T_2 = \frac{P_2}{\omega}$$

where ω is the synchronous speed in rad/s.

The efficiency is computed by

$$\eta = \frac{P_2}{P_1} \times 100\%$$

The motor is started the same way as for an induction motor, by using a squirrel-cage-type winding – called a damper winding in this case – that is mounted on the rotor, producing the starting torque.

Defining a Line-Start Permanent Magnet Synchronous Motor

The general procedure for defining a line-start synchronous motor is as follows:

1. Insert a line-start synchronous motor into a new or existing project.
2. Double-click the **Machine** entry in the project tree to define the general data.
3. Double-click the **Machine** > **Stator** entry in the project tree to define the stator geometry.
4. Double-click the **Machine** > **Stator** > **Slot** entry in the project tree to define the stator slot dimensions.
5. Double-click the **Machine** > **Stator** > **Winding** entry in the project tree to define the stator windings and conductors.
6. Double-click the **Machine** > **Rotor** entry in the project tree to define the rotor geometry.
7. Double-click the **Machine** > **Rotor** > **Pole** entry in the project tree to define the pole, embrace, offset, and air gap data for the rotor pole.
8. Double-click the **Machine** > **Shaft** entry in the project tree to define the magnetism of the shaft.
9. Optionally, right-click the **Machine** entry in the project tree and click **Insert Housing** to add a machine housing.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10. Right-click **Analysis** in the project tree, and click **Add Solution Setup** to define the solution data.
11. Choose **File > Save** to save the project.
12. Choose **RMxprt** > **Analyze** to analyze the design.

> **Note:** When you place the cursor over an entry field in the data windows, a brief description of that field appears in the status bar at the bottom of the **RMxprt** window.

Once analyzed, the model can be viewed in the Maxwell 2D Modeler, or it can be used to create a new Maxwell 2D project, and a new Maxwell 3D design.

Defining the General Data for a Line-Start PM Synchronous Motor

Use the **General** window to define the basic parameters of the motor, such as the motor's rated output power, rated voltage, losses, and connection type.

To define the general data:

1. To open the **General Data Properties** window, double-click the **Machine** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the number of poles for the machine in the **Number of Poles** field. This value is the total number of poles in the stator (or the number of pole pairs multiplied by two).
3. Enter the energy loss due to friction at the given speed in the **Frictional Loss** field.
4. Enter the wind loss due to air resistance measured at the reference speed in the **Wind Loss** field.
5. Enter the given speed in the **Reference Speed** field.
6. Click **OK** to close the **Properties** window.

General Data for Line-Start PM Synchronous Motors

To access the general data, double-click the **Machine** entry in the project tree.

The **General Data Properties** window for a three-phase induction motor contains the following fields:

Machine Type	The machine type you selected when inserting a new RMxprt design (Line-Start PM Synchronous Motor).
Number of Poles	The number of poles the machine contains.
Frictional Loss	The frictional energy loss (due to friction) measured at the reference speed.
Wind Loss	The wind loss (due to air resistance) measured at the reference speed.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reference Speed	The given speed of reference.

Defining the Stator Data for a Line-Start PM Synchronous Motor

The stator is the outer lamination stack where the polyphase voltage windings reside.

Use the **Stator Data**, **Stator Slot Data**, and **Stator Winding Data** windows to define the stator data, such as physical dimensions of the lamination, windings, and conductors.

To define the general stator data:

1. To open the **Stator Data Properties** window, double-click the **Machine** > **Stator** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the **Outer Diameter** of the stator.
3. Enter the **Inner Diameter** of the stator.
4. Enter the length of the stator core in the **Length** field.
5. Enter the effective magnetic length of the core in the **Stacking Factor** field.
6. Select a **Steel Type** for the stator core:
    a. Click the button for **Steel Type**.
    b. In the **Select Definition** window that appears, select a steel type from the list, or define a new steel type.
    c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.
7. Enter the **Number of Slots** in the stator.
8. Select the **Slot Type**:
    a. Click the button for the **Slot Type**.
    b. In the **Select Slot Type** window that appears, select a slot type (available types include 1 through 4).

    > **Note:** When you place the mouse cursor over the slot type, a schematic of the selected type appears, displaying the slot dimension variables.

    c. Click **OK** to close the **Select Slot Type** window and return to the **Properties** window.
9. Enter the skew width, measured in slot number, in the **Skew Width** field.
10. Click **OK** to close the **Properties** window.

Stator Data for Line-Start PM Synchronous Motors

To access the general stator data, double-click the **Machine** > **Stator** entry in the project tree.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Stator Data Properties** window contains the following fields:

Outer Diameter	The outer diameter of the stator core
Inner Diameter	The inner diameter of the stator core
Length	The length of the stator core
Stacking Factor	The stacking factor of the stator core
Steel Type	The steel type of the stator core. Click the button to open the **Select Definition** window.
Number of Slots	The number of slots the stator core contains
Slot Type	The type of slots in the stator core. Click the button to open the **Select Slot Type** window.
Skew Width	The skew width measured in slot number

Defining the Stator Slots for a Line-Start PM Synchronous Motor

To define the slot type:

1. To open the **Stator Slot Data Properties** window, double-click the **Machine** > **Stator** > **Slot** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Optionally, to automatically design the dimensions of slots **Hs2**, **Bs1**, and **Bs2**, select the **Auto Design** check box.
3. Optionally, to design dimensions of slots **Bs1** and **Bs2** based on the stator tooth width, select the **Parallel Tooth** check box, and enter a value in the **Tooth Width** field.
4. Enter the available slot dimensions:

Hs0	Always available
Hs2	Available only when **Auto Design** is cleared. When **Auto Design** is selected, this slot dimension is determined automatically.
Bs0	Always available
Bs1	Available only when **Auto Design** and **Parallel Tooth** are both cleared. When **Auto Design** is selected, this slot dimension is determined automatically. When **Parallel Tooth** is selected, this slot dimension is determined based on the value entered in the **Tooth Width** field.
Bs2	Available only when **Auto Design** and **Parallel Tooth** are both cleared. When **Auto Design** is selected, this slot dimension is determined automatically. When **Parallel Tooth** is selected, this slot dimension is determined based on the value entered in the **Tooth Width** field.
Rs	Rs is added when the slot type is 3 or 4.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click **OK** to close the **Properties** window.

Stator Slot Data for Line-Start PM Synchronous Motors

To access the stator slot data, double-click the **Machine** > **Stator** > **Slot** entry in the project tree.

The **Stator Slot Data Properties** window contains the following fields:

Auto Design	Select or clear this to enable or disable auto-design of slots **Hs2**, **Bs1**, and **Bs2**. When this check box is selected, only two other fields appear in the window: **Hs0** and **Bs0**.
Parallel Tooth	Select this to design **Bs1** and **Bs2** based on the tooth width. When this check box is selected, the **Bs1** and **Bs2** fields are removed, and the **Tooth Width** field is added.
Tooth Width	The tooth width for the parallel tooth, on which **Bs1** and **Bs2** are designed.
Hs0	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected).
Hs1	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected).
Hs2	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected).
Bs0	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected).
Bs1	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected).
Bs2	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected).
Rs	A slot dimension. (see the diagram shown in the modeling window when **Machine** > **Stator** > **Slot** is selected). Rs is added when the slot type is 3 or 4.

Defining the Stator Windings and Conductors for a Line-Start PM Synchronous Motor

To define the stator windings and conductors:

1. To open the **Stator Slot Winding Properties** window, double-click the **Machine** > **Stator** > **Winding** entry in the Project Manager tree on the desktop. You can also enter values in the **Properties** section of the desktop without opening a separate window.
2. Click the **Winding** tab.
3. Enter the number of layers in the stator winding in the **Winding Layers** field.
4. Select the **Winding Type** for the stator:

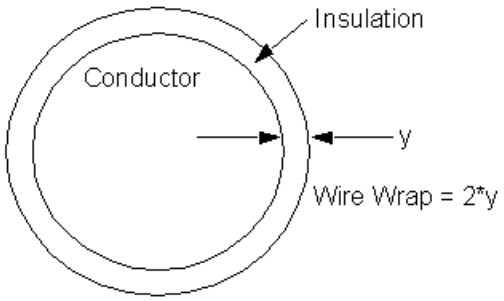
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Click the button for **Winding Type**.
b. In the **Winding Type** window that appears, select from one of the following three types of winding:
   - **Whole Coiled**
   - **Half Coiled**
   - **Editor**

> **Note:** When you place the mouse cursor over the winding type, a schematic of that type appears.

c. Click **OK** to close the **Winding Type** window and return to the **Properties** window.

5. Select or enter the number of parallel branches in one phase of the winding in the **Parallel Branches** field.
6. Enter the total number of conductors in each stator slot in the **Conductors per Slot** field. This value is the number of turns per coil multiplied by the number of layers.
7. Enter the coil pitch, measured in number of slots, in the **Coil Pitch** field. The coil pitch is the number of slots separating one winding. For example, if a coil starts in slot 1 and ends in slot 6, it has a coil pitch of 5.
8. Enter the number of wires per conductor in the **Number of Strands** field. Enter 0 to have RMxprt auto-design this value.
9. Enter the thickness of the double-sided wire wrap in the **Wire Wrap** field. Enter 0 to automatically obtain this value from the wire library.



10. Select the **Wire Size**:
    a. Click the button for **Wire Size**.
    The **Wire Size** window appears.
    b. Select a value from the **Wire Diameter** drop-down list.
    c. Select a wire gauge from the **Gauge** drop-down list. You can select from the following options:

< You can select a specific gauge number. When you select

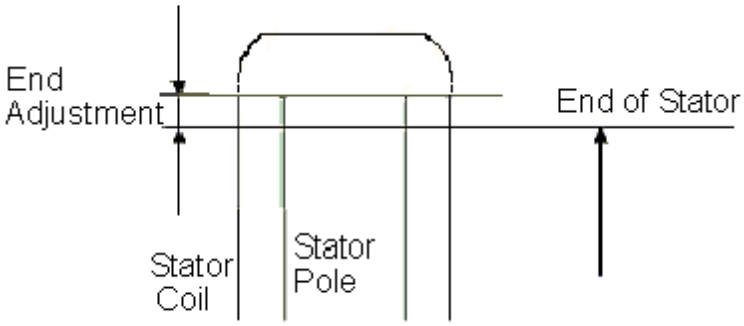
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

number >	a gauge number, the **Wire Diameter** field is automatically updated.
USER	This option allows you to manually enter the **Wire Diameter**. This is useful when you want to enter a diameter that does not correspond to a particular wire gauge.
AUTO	This option sets the **Wire Diameter** to zero, and RMxprt automatically calculates the optimal value. The diameter information is then written to the output file when you analyze the design.
MIXED	This option allows you to define a conductor that is made of different size wires. For example, a single conductor may consist of five wires: three wires with a diameter of 0.21 mm and two wires with a diameter of 0.13 mm.

The gauge number is based on AWG settings. You can create your own wire table using **Machine > Wire**, and then you can select this wire table using **Tools** > **Options** > **General Options** > **Machines**.

d. When you are done setting the wire size, click **OK** to close the **Wire Size** window and return to the **Properties** window.

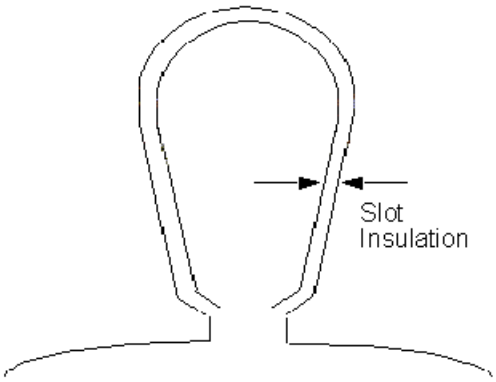
11. Click the **End/Insulation** tab.
12. Select or clear the **Input Half-turn Length** check box.
13. Do one of the following:
    - If you selected **Input Half-turn Length**, then enter the half-turn length of the armature winding in the Half Turn Length field.
    - If you cleared **Input Half-turn Length**, then enter the end length adjustment of the stator coils in the **End Adjustment** field. The end adjustment is the distance one end of the conductor extends vertically beyond the end of the stator.

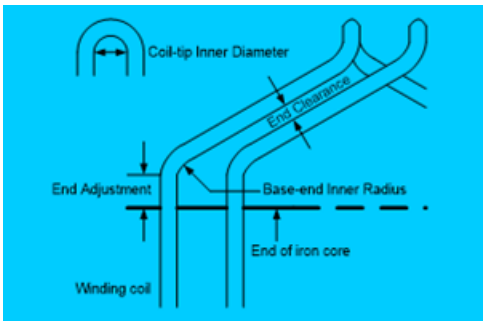


14. Enter the inner radius of the base corner in the **Base Inner Radius** field.
15. Enter the inner diameter of the coil tip in the **Tip Inner Diameter** field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16. Enter the distance between two stator coils in the **End Clearance** field.
17. Enter the thickness of the slot liner insulation in the **Slot Liner** field.





18. Enter the thickness of the wedge insulation in the **Wedge Thickness** field.
19. Enter the thickness of the insulation layer in the **Layer Insulation** field.
20. Enter the limited slot fill factor for the wire design in the **Limited Fill Factor** field. The slot fill factor is the ratio between the cross-sectional area of all conductors in one slot and the entire slot area.
21. Click **OK** to close the **Properties** window.

Winding Editor for a Line-Start Synchronous Motor

For a line-start synchronous motor, you may want to specify a different number of conductors for each stator slot. The Winding Editor makes this possible by enabling you to specify the number of turns for each coil.

To specify the number of turns for each coil:

1. Click **Machine** > **Winding** > **Edit Layout**.

   The **Winding Editor** window appears.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the table in the upper left, set which phase you want for each coil and which slot is the “in” and “out” slot for the current in each coil.
3. If you are working on a quarter or half model, you may want to specify a multiplier by clicking the **Periodic Multiplier** check box and specifying a value.
4. Select or deselect the **Constant Turns** or **Constant Pitch** check boxes, depending on whether you want to be able to change these setting in the table above. When these options are selected, you cannot change the turns or pitch.
5. When you are satisfied with the coil settings, click **OK** to close the **Winding Editor** window.

Defining Different Size Wires for a Line-Start Synchronous Motor

Use the **Gauge** option in the **Wire Size** window if you have a conductor that is made up different size wires.

To define different size wires:

1. In the **Wire Size** window, select **MIXED** from the **Gauge** drop-down menu.
2. Select either **Round** or **Rectangular** as the **Wire Type**.
3. Enter the appropriate wire data in the table:
    - For a round wire:
        - Enter the **Diameter** in the table.
        - Enter a **Number** in the table to specify how many of the conductor’s wires have this diameter.
    - For a rectangular wire:
        - Enter the **Width** of the wire in the table.
        - Enter the **Thickness** of the wire in the table.
        - Enter the **Fillet** value in the table.
        - Enter a **Number** in the table to specify how many of the conductor’s wires have this data.
4. Click **Add** to add the new wire data.
5. Repeat steps 3 and 4 for each size wire you want to add.
6. When you are finished defining the wires, click **OK** to close the **Wire Size** window and return to the RMxprt **Properties** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: For example, if one conductor is made up of five wires, and three of those wires have a diameter of 0.21 mm, and the other 2 have a diameter of 0.13 mm, then the mixed wire size table will have two lines. The first line will list Diameter = 0.21 and Number = 3. The second line will list Diameter = 0.13 and Number = 2. An equivalent wire diameter is displayed as Wire Size value in the Winding tab in the Properties window.

Stator Winding Data for Line-Start PM Synchronous Motors

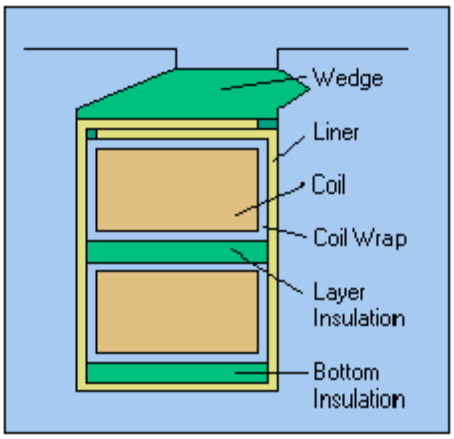
To access the stator winding data, double-click the **Machine** > **Stator** > **Winding** entry in the project tree.

The **Stator Winding Data Properties** window contains the following fields:

Winding tab	**Winding Layers**	The number of winding layers.
	Winding Type	The type of stator winding. Click the button to open the **Winding Type** window and choose from **Whole Coiled**, **Half Coiled**, and **Editor**.
	Parallel Branches	The number of parallel branches in the stator winding.
	Conductors per Slot	The number of conductors per stator slot (0 for auto-design).
	Coil Pitch	The coil pitch measured in number of slots.
	Number of Strands	The number of wires per conductor (0 for auto-design).
	Wire Wrap	The thickness of the double-sided wire wrap (0 to automatically obtain this value from the wire library).
	Wire Size	The diameter of the wire (0 for auto-design). Click the button to open the **Wire Size** window where you can specify units, wire type, diameter, and gauge.
End/Insulation tab	**Input Half-turn Length**	Select or clear this check box to specify whether or not you want to enter the half-turn length. When this check box is selected, the **Half Turn Length** field appears the next time you open the **Properties** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	When this check box is selected, the **End Adjustment** field appears instead.
Half Turn Length	The half-turn length of the armature winding.
End Adjustment	The end length adjustment of the stator coils, which is the distance one end of the conductor extends vertically beyond the end of the stator.
Base Inner Radius	The inner radius of the base corner.
Tip Inner Diameter	The inner diameter of the coil tip.
End Clearance	The end clearance between two adjacent coils.
Slot Liner	The thickness of the slot liner insulation.
Wedge Thickness	The thickness of the wedge insulation.
Layer Insulation	The thickness of the insulation layer.
Limited Fill Factor	The limited slot fill factor for the wire design. The slot fill factor is the ratio between the cross-sectional area of all conductors in one slot and the entire slot area.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Optional Vent for Line-Start PM Synchronous Motor Stator

To add a Vent to the stator, select the stator icon and right-click to display the pop-up menu with **Insert Vent**.

The vent is shown in the project tree under the stator.

To remove an existing Vent, select the stator and right-click to display the up-up menu with **Remove Vent**.

The Vent Data properties window contains the following fields.

Vent Ducts	Number of radial vent ducts
Duct Width	Width of radial vent ducts
Magnetic spacer width	Width of magnetic spacer which hold vent ducts. 0 for non-magnetic spacer.
Duct pitch	Vent ducts.

Defining the Rotor Data for a Line-Start PM Synchronous Motor

The rotor consists of copper bars in which current is induced by the magnetic fields produced by the stator windings. In the project tree, double-click **Machine** > **Rotor** and **Machine** > **Rotor** > **Pole** to define the rotor and the pole.

To define general rotor data:

1. To open the **Rotor Data Properties** window, double-click the **Machine** > **Rotor** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the outer diameter of the rotor in the **Outer Diameter** field.
3. Enter the inner diameter of the rotor in the **Inner Diameter** field.
4. Enter the length of the rotor core in the **Length** field.
5. Select a **Steel Type** for the rotor core:
    a. Click the button for **Steel Type**.
    b. In the **Select Definition** window that appears, select a steel type from the list, or define a new steel type.
    c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.
6. Enter the effective magnetic length of the rotor core in the **Stacking Factor** field. This value ranges from 0 to 1 and is defined as the total length minus the total lamination insulation, divided by the total length. A value of 1 indicates that the rotor is not laminated.
7. Select a **Pole Type**:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Click the button.
b. In the **Select Pole Type** window that appears, click a button to select the desired pole type (1, 2, 3, 4, 5, 6, 7, or 8).

> **Note:** When you place the mouse cursor over a pole type, an outline of the selected circuit type appears.

c. Click **OK** to close the **Select Pole Type** window and return to the **Properties** window.

8. Click **OK** to close the **Properties** window.

Rotor Data for Line-Start PM Synchronous Motors

To access the general rotor data, double-click the **Machine** > **Rotor** entry in the Project Manager tree.

The **Rotor Data Properties** window contains the following fields:

Outer Diameter	The outer diameter of the rotor core
Inner Diameter	The inner diameter of the rotor core
Length	The length of the rotor core
Steel Type	The steel type of the rotor core. Click the button to open the **Select Definition** window.
Stacking Factor	The stacking factor of the rotor core
Pole Type	The pole type for the rotor. Click this button to open the **Select Pole Type** window and select from the following types: **1**, **2**, **3**, **4**, **5**, **6**, **7**, **8**. When you mouse over each button, a diagram appears for that pole type, showing the arrangement and dimensions.

Defining the Rotor Pole for a Line-Start PM Synchronous Motor

The rotor pole drives the electromagnetic field which is coupled with the stator windings. Use the **Rotor Pole Data Properties** window to define the rotor pole.

> **Note:** Some of the fields in the **Rotor Pole** window change, or are inactive, depending on the **Rotor Type** you select.

To define the rotor pole:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. To open the **Rotor Pole Data Properties** window, double-click the **Machine** > **Rotor** > **Pole** entry in the Project Manager tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the limited diameter for the magnet ducts in the **D1** field.
3. Enter one or more of the following magnet duct dimensions, depending on the pole type selected: **O1**, **O2**, **B1**.
4. For all pole types except number **8**, enter the width of the rib supporting the bridge in the **Rib** field.
5. Select the type of magnet to use in the rotor pole:
    a. Click **Magnet Type** button.
    The **Select Definition** window appears.
    b. Select a material.
    c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.
6. Enter the total width of all magnets per pole in the **Magnet Width** field.
7. Enter the maximum radial thickness of the magnet in the **Magnet Thickness** field.
8. Click **OK** to close the **Properties** window.

Rotor Pole Data for Line-Start PM Synchronous Motors

To access the pole rotor data, double-click the **Machine** > **Rotor** > **Pole** entry in the Project Manager tree.

The **Rotor Pole Data Properties** window contains the following fields:

D1	The limited diameter for the magnet ducts. See the diagrams in the **Select Pole Type** window for the location of each dimension and which pole types require which dimensions.
O1	A magnet duct dimension. See the diagrams in the **Select Pole Type** window for the location of each dimension and which pole types require which dimensions.
O2	A magnet duct dimension. See the diagrams in the **Select Pole Type** window for the location of each dimension and which pole types require which dimensions.
B1	A magnet duct dimension. See the diagrams in the **Select Pole Type** window for the location of each dimension and which pole types require which dimensions.
Rib	The width of the rib at the center of two adjacent poles that support the bridge. For pole types except number **8**.
Magnet Type	The type of magnet. Click the button to open the **Select Definition** window. For all pole types.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Magnet Width	The maximum width of the magnet. For all pole types.
Magnet Thickness	The maximum thickness of the magnet. For all pole types.

Optional Rotor Damper for Line-Start PM Synchronous Motor

- To add a damper, right-click on the rotor item in the project tree to display the pop-up menu with **Insert Damper**.
- To remove an existing damper, right-click on the rotor icon in the project tree to display the shortcut menu with **Remove Damper**.

The **Damper Data** properties window contains the following fields.

Damper Slots per Pole	Number of damper slots per pole
Slot Type	Damper slot type. Click the field button open the Slot selection window and select one of the four types.
Cast Rotor	Specify whether the rotor squirrel cage winding is cast
Bar conductor type	Click the field button to open the Materials Selection window to specify the material for the bar conductor.
End Length	Single side end extended bar length
End Ring Width	Axial width of end ring
End Ring Height	Radial height of end ring
End Ring Conductor type	Click the field button to open the Materials Selection window to specify the material for the end ring conductor.

Defining the Shaft Data for a Line-Start PM Synchronous Motor

To define the shaft:

1. To open the **Shaft Data Properties** window, double-click the **Machine** > **Shaft** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Select or clear the **Magnetic Shaft** check box to specify whether or not the shaft is to be made of magnetic material.
3. Click **OK** to close the **Properties** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Shaft Data for Line-Start PM Synchronous Motors

To access the shaft data, double-click the **Machine** > **Shaft** entry in the Project Manager tree.

The **Shaft Data Properties** window contains the following fields:

Magnetic Shaft	Select or clear this check box to indicate whether or not the shaft is made of magnetic material.

Setting Up Analysis Parameters for a Line-Start PM Synchronous Motor

To define the solution data:

1. To open the Solution Setup window, right-click **Analysis** in the Project Manager tree, and click **Add Solution Setup**.
2. Click the **General** tab. The **Operation Type** is automatically set to **Motor** for this machine type.
3. Select the **Load Type** used in the motor from the following options:

Const Speed	The speed remains constant in the motor
Const Power	The output power remains constant in the motor.
Const Torque	The torque remains constant regardless of the speed. In this case, $T_{load} = T_{rated}$, given by the output power divided by the given rated speed.
Linear Torque	The torque increases linearly with speed. In this case, $T_{load} = T_{rated} * (n/n_{rated})$ where T_{rated} is given by the output power divided by the given rated speed.
Fan Load	The load varies nonlinearly with speed. In this case, $T_{load} = T_{rated} * (n/n_{rated})^2$ where T_{rated} is given by the output power divided by the given rated speed.

4. Enter the output power developed at the shaft of the motor in the **Rated Output Power** field.
5. Enter the RMS line-to-line voltage in the **Rated Voltage** field.
6. Enter the desired output speed of the motor at the load point in the **Rated Speed** field.
7. Enter the temperature at which the system functions in the **Operating Temperature** field.
8. Click the **Line-Start PM Synchronous Motor** tab.
9. Select **Wye** or **Delta** from the **Winding Connection** drop-down menu.
10. Click **OK** to close the **Solution Setup** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution Data for Line-Start PM Synchronous Motors

To access the solution data, right-click **Analysis** in the project tree, and click **Add Solution Setup**.

The **Solution Setup** window contains the following fields:

Operation Type	**General** tab. The operation type is automatically set to **Motor** for this machine type.
Load Type	**General** tab. Select from **Const Speed**, **Const Power**, **Const Torque**, **Linear Torque**, and **Fan Load**. The default is **Const Power**.
Rated Output Power	**General** tab. Type a value for the rated output voltage, and select the units.
Rated Voltage	**General** tab. Type a value for the rated voltage, and select the units.
Rated Speed	**General** tab. Type a value for the rated speed, and select the units.
Operating Temperature	**General** tab. Type a value for the operating temperature, and select the units.
Winding Connection	**Line-Start PM Synchronous Motor** tab. Select **Wye** or **Delta** from the **Winding Connection** drop-down menu.

Universal Motors

After you have selected **Universal Motors** as your model type, enter the motor data to define the following:

- General data, such as the number of poles, frictional loss, and reference speed
- Stator pole and winding data, such as its associated pole dimensions, type of steel, and wire definitions
- Rotor data, such as the slot types and dimensions, rotor diameter, laminations, and windings and conductors
- Commutator and brush data, such as the commutator dimensions and brush length
- Shaft data
- Optional machine housing
- Solution data

Analysis Approach for Universal Motors

For a DC motor, if its field winding is connected in series with its armature winding, it becomes a series motor. When the polarity of the terminal voltage changes, the direction of the produced electromagnetic torque does not change because the armature and the exciting currents alternate their directions at the same time. That means the motor can operate not only with a DC

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

source but also with an AC source. Because it can operate with both DC and AC sources, a series motor is also called universal motor (UniM).

For a universal motor, the stator is equipped with *p* pairs of coil-wound poles, creating *P* pairs of alternating north and south poles. The coil excitation may be either AC or DC. The rotor is equipped with a distributed winding connected to a commutator that revolves together with the rotor.

A system of brushes is kept in permanent electrical contact with the commutator. When AC or DC current is applied to the rotor winding (via the brushes and commutator) a torque is produced by the interaction of the rotor (armature) currents and the field produced by the stator poles.

The commutator causes the armature to create a magnetic flux distribution whose axis is perpendicular to the axis of the field flux produced by the permanent magnets. For these motors, the commutator acts as a mechanical rectifier.

The performance of a universal motor is analyzed in the frequency domain. The voltage equation of a universal motor is

$$U = ZI = (R_a + R_f + R_b)I + j\omega\left(L_a + L_f + 2M_{af}\right)I + \omega_e\left(G_{aa} + G_{af}\right)I$$

where, R_a,R_f, and R_b are the armature resistance, field winding resistance, and the brush contact resistance, respectively. L_a, L_f, and M_{af} are the armature self inductance, field winding self inductance, and their mutual inductance, respectively, and are linearized nonlinear parameters. Gaa and Gaf are the coefficients of motion induced voltages by the armature and field winding currents, respectively, and are also linearized nonlinear parameters. ω is the radian frequency, and ω_e the rotor speed in electric rad/s. Z is equivalent input impedance. When the brush axis is aligned with q-axis, $M_{af} = G_{aa} = 0$.

For a given rotor speed ω_e, armature current can be computed based on the applied voltage *U*, as

$$I = \frac{U}{Z}$$

The input power (electric power) is directly computed from voltage and current as

$$P_1 = UI\cos\varphi$$

The output power (mechanical power) is

$$P_2 = P_1 - (P_{fw} + P_b + P_{cuf} + P_{Fe})$$

where P_{fw}, P_b, P_{cua}, P_{cuf}, and P_{Fe} are frictional and wind loss, brush drop loss, armature copper loss, field winding copper loss, and iron-core loss, respectively.

The output mechanical shaft torque T_2 is

$$T_2 = \frac{P_2}{\omega}$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The efficiency is computed by

$$\eta = \frac{P_2}{P_1} \times 100\%$$

Defining a Universal Motor

The general procedure for defining a universal motor is as follows:

1. Insert a universal motor into a new or existing project.
2. Double-click the **Machine** entry in the Project Manager tree to define the general data.
3. Double-click the **Machine** > **Stator** entry in the Project Manager tree to define the stator geometry.
4. Double-click the **Machine** > **Stator** > **Pole** entry in the Project Manager tree to define the stator pole dimensions.
5. Double-click the **Machine** > **Stator** > **Winding** entry in the Project Manager tree to define the stator windings and conductors.
6. Double-click the **Machine** > **Rotor** entry in the Project Manager tree to define the rotor geometry.
7. Double-click the **Machine** > **Rotor** > **Slot** entry in the Project Manager tree to define the rotor slot dimensions.
8. Double-click the **Machine** > **Rotor** > **Winding** entry in the Project Manager tree to define the rotor windings and conductors.
9. Double-click the **Machine** > **Commutator** entry in the project tree to define the commutator and brush data.
10. Double-click the **Machine** > **Shaft** entry in the project tree to define the magnetism of the shaft.
11. Optionally, right-click the **Machine** entry in the project tree and click **Insert Housing** to add a machine housing.
12. Right-click **Analysis** in the project tree, and click **Add Solution Setup** to define the solution data.
13. Choose **File** > **Save** to save the project.
14. Choose **RMxprt > Analyze** to analyze the design.

> **Note:** When you place the cursor over an entry field in the data windows, a brief description of that field appears in the status bar at the bottom of the **RMxprt** window.

Once analyzed, the model can be viewed in the Maxwell 2D Modeler, or it can be used to create a new Maxwell 2D project, and a new Maxwell 3D design.

Refer to the *Universal Motor Problem* application note, on the technical support page of the Ansys web site, for a specific example.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining the General Data for a Universal Motor

Use the **General** window to define the basic parameters of the universal motor such as the power settings, speed, and rated voltage.

To define the general data:

1. To open the **General Data Properties** window, double-click the **Machine** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the number of poles for the machine in the **Number of Poles** field. This value is the total number of poles in the stator (or the number of pole pairs multiplied by two).
3. Enter the energy loss due to friction at the given speed in the **Frictional Loss** field.
4. Enter the wind loss due to air resistance measured at the reference speed in the **Wind Loss** field.
5. Enter the given speed in the **Reference Speed** field.
6. Click **OK** to close the **Properties** window.

General Data for Universal Motors

To access the general data, double-click the **Machine** entry in the project tree.

The **General Data Properties** window for a three-phase induction motor contains the following fields:

Machine Type	The machine type you selected when inserting a new RMxprt design (Switched Reluctance Motor)
Number of Poles	Number of poles for this machine
Frictional Loss	The frictional energy loss (due to friction) measured at the reference speed
Wind Loss	The wind loss (due to air resistance) measured at the reference speed
Reference Speed	The given speed of reference

Defining the Stator Data for a Universal Motor

Use the **Stator Properties** windows to define the stator dimensions, slots, windings, and conductors.

The stator is the outer lamination stack where the polyphase voltage windings reside.

To define the general stator data:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. To open the **Stator Data Properties** window, double-click the **Machine** > **Stator** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the **Outer Diameter** of the stator.
3. Enter the overall width of the stator outer profile in the **Overall Width** field.
4. Enter the **Inner Diameter** of the stator.
5. Enter the length of the stator core in the **Length** field.
6. Enter the stacking factor for the stator core in the **Stacking Factor** field.
7. Select a **Steel Type** for the stator core:
    a. Click the button for **Steel Type**.
    b. In the **Select Definition** window that appears, select a steel type from the list, or define a new steel type.
    c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.
8. Click **OK** to close the **Properties** window.

Stator Data for Universal Motors

To access the general stator data, double-click the **Machine** > **Stator** entry in the project tree.

The **Stator Data Properties** window contains the following fields:

Outer Diameter	The outer diameter of the stator core
Overall Width	The overall width of the stator outer profile
Inner Diameter	The inner diameter of the stator core
Length	The length of the stator core
Stacking Factor	The effective magnetic length of the stator core
Steel Type	The steel type of the stator core. Click the button to open the **Select Definition** window.

Defining the Stator Pole for a Universal Motor

The rotor pole drives the electromagnetic field which is coupled with the stator windings. Use the **Stator Pole Data Properties** window to define the stator pole.

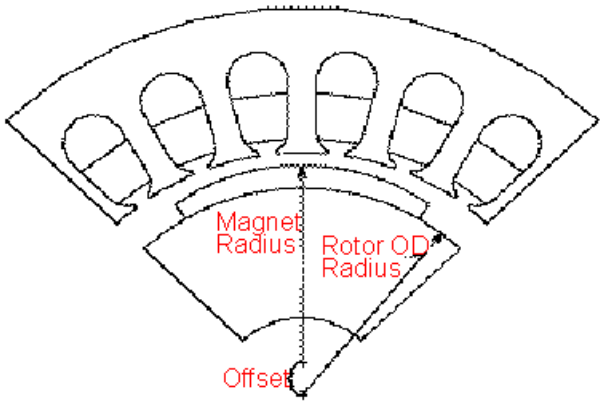
To define the rotor pole:

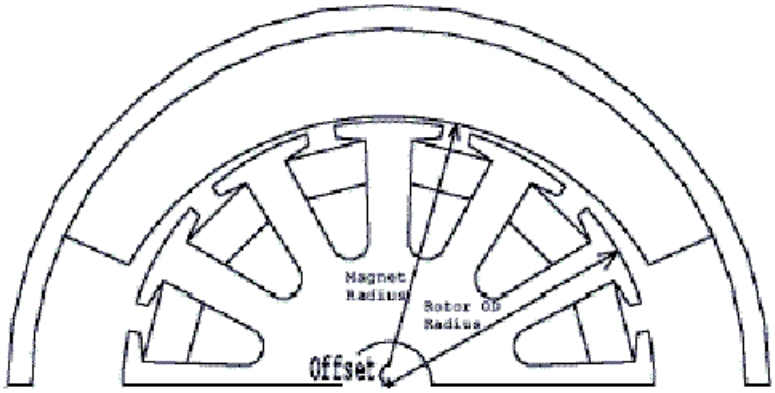
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. To open the **Stator Pole Data Properties** window, double-click the **Machine** > **Stator** > **Pole** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)

   > **Note:** For a two-pole machine, a pole embrace of 0.75 yields a magnet with a span of 135 degrees (based on 0.75*180 degrees).

2. Enter the ratio of the actual arc distance in relation to the maximum possible arc distance in the **Embrace** field. This value is between 0 and 1.
3. Enter the distance from the center of the stator to the magnet arc center in the **Offset** field. Enter 0 for a uniform air gap.





4. Enter the minimum pole width in the **PoleWidth** field.
5. Enter the yoke thickness in the **Ty** field.
6. Enter the shoe-tip thickness in the **Ts** field.
7. Enter the pole's hole radius in the **R1** field. If there is no hole in the design, enter **0**.
8. Enter the pole's side fillet radius in the **R2** field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. Enter the radius of the pole's center side fillet arcs in the **R3** field.
10. Enter the radius of the shoe connecting arc in the **R4** field. To auto-design this dimension, enter **0**. For a linear connection, enter **0**.
11. Enter the inner radius of the screw hole between the two poles in the **R5** field. If there is no hole in the design, enter **0**.
12. Enter the outer radius of the screw hole between the two poles in the **R6** field. If there is no hole in the design, enter **0**.
13. Click **OK** to close the **Properties** window.

Stator Pole Data for Universal Motors

To access the stator pole data, double-click the **Machine** > **Stator** > **Pole** entry in the Project Manager tree.

The **Stator Pole Data Properties** window contains the following fields:

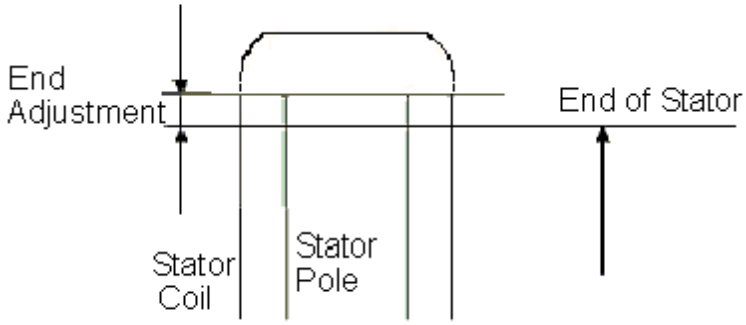
Embrace	The pole embrace
Offset	The pole-arc center offset from the stator center (0 for a uniform air gap)
PoleWidth	The minimum pole width
Ty	The yoke thickness
Ts	The shoe-tip thickness
R1	The hole radius in the pole (0 for no hole)
R2	The radius of the pole side fillet
R3	The radius of the center of the pole side fillet arcs
R4	The radius of the shoe connecting arc (0 for auto-design or for a linear connection)
R5	The inner radius of the screw hole between two poles (0 for no hole)
R6	The outer radius of the screw hole between two poles (0 for no hole)

Defining the Stator Windings and Conductors for a Universal Motor

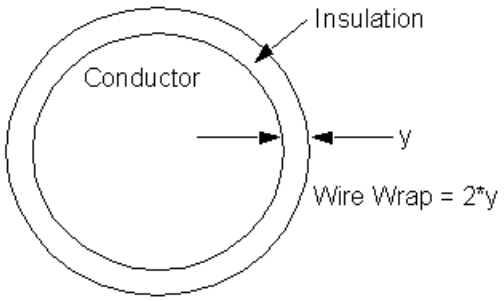
To define the stator windings and conductors:

1. To open the **Stator Winding Properties** window, double-click the **Machine** > **Stator** > **Winding** entry in the Project Manager tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the thickness of the insulation between the stator core and the field winding in the **Insulation Thickness** field.
3. Enter the end length adjustment of the stator coils in the **End Adjustment** field. The end adjustment is the distance one end of the conductor extends vertically beyond the end of the stator.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Enter the number of parallel branches in the stator winding in the **Parallel Branches** field.
5. Enter the number of turns per stator pole in the **Turns per Pole** field. To auto-design the number of turns, enter **0**.
6. Enter the number of wires per conductor in the **Number of Strands** field. Enter **0** to have RMxprt auto-design this value.
7. Enter the thickness of the double-sided wire wrap in the **Wire Wrap** field. Enter 0 to automatically obtain this value from the wire library.



8. Select the **Wire Size**:
    a. Click the button for **Wire Size**.
    b. In the **Wire Size** window that appears, select a value from the **Wire Diameter** drop-down menu.
    c. Select a wire gauge from the **Gauge** drop-down menu. You can select from the following options:

< *number* >	You can select a specific gauge number. When you select a gauge number, the **Wire Diameter** field is automatically updated.
USER	This option allows you to manually enter the **Wire Diameter**. This is useful when you want to enter a diameter that does not correspond to a particular wire gauge.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AUTO	This option sets the **Wire Diameter** to zero, and RMxprt automatically calculates the optimal value. The diameter information is then written to the output file when you analyze the design.
MIXED	This option allows you to define a conductor that is made of different size wires. For example, a single conductor may consist of five wires: three wires with a diameter of 0.21 mm and two with a diameter of 0.13 mm.

The gauge number is based on AWG settings. You can create your own wire table using **Machine > Wire**, and then you can select this wire table using the **Tools** > **Options** > **General Options** > **Machines** function.

d. When you are done setting the wire size, click **OK** to close the **Wire Size** window and return to the **Properties** window.

9. Enter the thickness of the wedge insulation in the **Wedge Thickness** field.
10. Enter the thickness of the insulation layer in the **Layer Insulation** field.
11. Enter the limited slot fill factor for the wire design in the **Limited Fill Factor** field. The slot fill factor is the ratio between the cross-sectional area of all conductors in one slot and the entire slot area.
12. Click **OK** to close the **Properties** window.

Defining Different Size Wires for a Universal Motor Stator Winding

To define different size wires:

1. In the **Wire Size** window, select **MIXED** from the **Gauge** drop-down menu.
2. Select either **Round** or **Rectangular** as the **Wire Type**.
3. Enter the appropriate wire data in the table:
    - For a round wire:
        - Enter the **Diameter** in the table.
        - Enter a **Number** in the table to specify how many of the conductor's wires have this diameter.
    - For a rectangular wire:
        - Enter the **Width** of the wire in the table.
        - Enter the **Thickness** of the wire in the table.
        - Enter the **Fillet** value in the table.
        - Enter a **Number** in the table to specify how many of the conductor's wires have this data.
4. Click **Add** to add the new wire data.
5. Repeat steps 3 and 4 for each size wire you want to add.
6. When you are finished defining the wires, click **OK** to close the **Wire Size** window and return to the RMxprt **Properties** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: For example, if one conductor is made up of five wires, and three of those wires have a diameter of 0.21 mm, and the other 2 have a diameter of 0.13 mm, then the mixed wire size table will have two lines. The first line will list Diameter = 0.21 and Number = 3. The second line will list Diameter = 0.13 and Number = 2. An equivalent wire diameter is displayed as Wire Size value in the Winding tab in the Properties window.

Stator Winding Data for Universal Motors

To access the stator winding data, double-click the **Machine** > **Stator** > **Winding** entry in the project tree.

The **Stator Winding Data Properties** window contains the following fields:

Insulation Thickness	The thickness of the insulation between the stator core and the field winding
End Adjustment	The end length adjustment of the stator coils, which is the distance one end of the conductor extends vertically beyond the end of the stator
Parallel Branches	The number of parallel branches in the stator winding
Turns per Pole	The number of turns per stator pole (0 for auto-design)
Number of Strands	The number of wires per conductor (0 for auto-design)
Wire Wrap	The thickness of the double-sided wire wrap (0 to automatically obtain this value from the wire library)
Wire Size	The diameter of the wire (0 for auto-design). Click the button to open the **Wire Size** window where you can specify units, wire type, diameter, and gauge.

Defining the Rotor Data for a Universal Motor

The rotor is equipped with slots containing copper conductors that are connected to the commutator. The commutator acts as a mechanical rectifier in the motor.

The rotor consists of copper bars in which current is induced by the magnetic fields produced by the stator windings.

To define general rotor data:

1. To open the **Rotor Data Properties** window, double-click the **Machine** > **Rotor** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Enter the stacking factor for the rotor core in the **Stacking Factor** field. This value relates to the effective magnetic length of the core, and ranges from 0 to 1. It is defined as the total length minus the total insulation from the laminations, divided by the total length. A value of 1 indicates that the rotor is not laminated.
3. Enter the number of slots in the rotor core in the **Number of Slots** field.
4. Select a **Slot Type**:
    a. Click the button.
    b. In the **Select Slot Type** window that appears, click a button to select the desired pole type (1, 2, 3, 4, 5, or 6). Though slots **3** and **4** are visually similar, they differ in how the edges are constructed. Slot **3** has a tapered edge leading from the slot opening to the main slot body. Slot **4** has a rounded edge at the same location, where the quantity **Hr1** defines the radius of the corner slot. TIP: When you run the mouse over each option, the diagram changes to show that pole type.
    c. Click **OK** to close the **Select Slot Type** window and return to the **Properties** window.

    Optionally, check **User Defined Slot** if you wish to define the slot dimensions using the **Slot Editor**.
5. Enter the outer diameter of the rotor core in the **Outer Diameter** field.
6. Enter the inner diameter of the rotor core in the **Inner Diameter** field.
7. Enter the length of the rotor core in the **Length** field.
8. Select a **Steel Type** for the rotor core:
    a. Click the button for **Steel Type**.
    b. In the **Select Definition** window that appears, select a steel type from the list, or define a new steel type.
    c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.
9. Enter the number of slots in the skew width in the **Skew Width** field.
10. Click **OK** to close the **Properties** window.

Rotor Data for Universal Motors

To access the general rotor data, double-click the **Machine** > **Rotor** entry in the project tree.

The **Rotor Data Properties** window contains the following fields:

Stacking Factor	The effective magnetic length of the rotor core
Number of Slots	The number of slots in the rotor core
Slot Type	The rotor core slot type. Click the button to open the **Select Slot Type** window and select from the following types: **1**, **2**, **3**, **4**, **5**, **6**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Outer Diameter	The outer diameter of the rotor core
Inner Diameter	The inner diameter of the rotor core
Length	The length of the rotor core
Steel Type	The steel type of the rotor core. Click the button to open the **Select Definition** window
Skew Width	The skew width measured in slot number

Defining the Rotor Slots for Universal Motors

To define the physical dimensions of the rotor slots:

1. To open the **Rotor Slot Data Properties** window, double-click the **Machine** > **Rotor** > **Slot** entry in the Project Manager tree. (You can also enter values in the **Properties** window without opening a separate window.)
2. Optionally, to automatically design the dimensions of slots **Hs2**, **Bs1**, and **Bs2**, select the **Auto Design** check box. Using this option causes the software to converge to a flux density value of 1.5 Tesla in the rotor teeth.
3. Enter the available slot dimensions. The following dimensions may be listed, depending on the **Slot Type** selected and depending on whether or not **Auto Design** is selected:
   - **Hs0** Always available
   - **Hs1** Always available
   - **Hs2** Available only when **Auto Design** is cleared. When **Auto Design** is selected, this slot dimension is determined automatically.
   - **Bs0** Always available
   - **Bs1** Available only when **Auto Design** is cleared. When **Auto Design** is selected, this slot dimension is determined automatically. When **Parallel Tooth** is selected, this slot dimension is determined based on the value entered in the **Tooth Width** field.
   - **Bs2** Available only when **Auto Design** is cleared. When **Auto Design** is selected, this slot dimension is determined automatically. When **Parallel Tooth** is selected, this slot dimension is determined based on the value entered in the **Tooth Width** field.
   - **Rs** Always available; Rs is added when the slot type is 3 or 4.
4. Click **OK** to close the **Properties** window.

Rotor Slot Data for Universal Motors

To access the rotor slot data, double-click the **Machine** > **Rotor** > **Slot** entry in the Project Manager tree.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Rotor Slot Data Properties** window contains the following fields:

Auto Design	Select or clear this to enable or disable auto-design of slots **Hs2**, **Bs1**, and **Bs2**. When this check box is selected, only two other fields appear in the window: **Hs0** and **Bs0**.
Parallel Tooth	Select this to design **Bs1** and **Bs2** based on the tooth width. When this check box is selected, the **Bs1** and **Bs2** fields are removed, and the **Tooth Width** field is added.
Tooth Width	The tooth width for the parallel tooth, on which **Bs1** and **Bs2** are designed
Hs0	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected)
Hs1	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected)
Hs2	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected)
Bs0	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected).
Bs1	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected)
Bs2	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected)
Rs	A slot dimension. (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected). Rs is added when the slot type is 3 or 4.

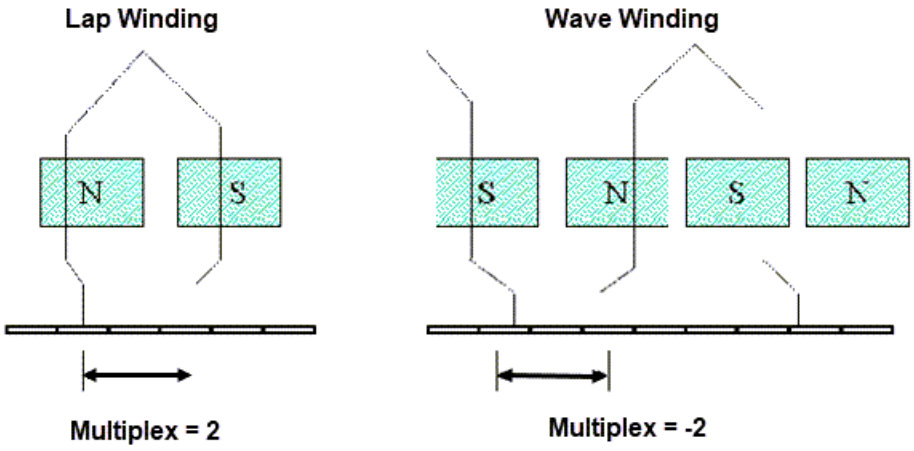
Defining the Rotor Windings and Conductors for a Universal Motor

To define the rotor windings, wires, and conductors:

1. To open the **Rotor Slot Winding Properties** window, double-click the **Machine** > **Rotor** > **Winding** entry in the Project Manager tree. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Click the **Winding** tab.
3. Select a **Winding Type**:
    a. Click the button for **Winding Type**.
    b. In the **Winding Type** window that appears, select from one of the following three types of winding:
        - **Lap**
        - **Wave**
        - **Frog Leg**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Enter the number of windings in the **Multiplex Number** field (1 for a single winding, 2 for double windings, 3 for triple windings). For a lap winding, the multiplex number is the number of commutators between the start and end of one winding, and the number of parallel branches is equal to the number of poles multiplied by the multiplex number. For a wave winding, the number of parallel branches equals the multiplex number multiplied by two.

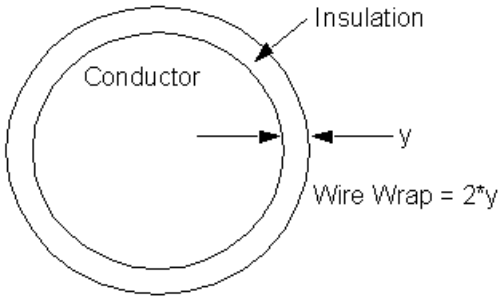


5. Enter the number of virtual slots per each real slot in the **Virtual Slots** field. The rotor is assumed to have two layers of conductors, an upper and a lower layer. Each layer of conductors can have a number of windings, which are referred to as virtual slots.

> **Note:** For example, the upper and lower layer can have two windings each, which would yield a virtual slot number of two; for a 12-slot machine, this would yield 24 commutation segments.

6. Enter the total number of conductors in each rotor slot in the **Conductors per Slot** field. This value is the number of turns per coil multiplied by the number of layers. This value is the total number of conductors in one real full rotor slot.
7. Enter the coil pitch, measured in number of slots, in the **Coil Pitch** field. The coil pitch is the number of slots separating one winding. For example, if a coil starts in slot 1 and ends in slot 6, it has a coil pitch of 5.
8. Enter the number of wires per conductor in the **Number of Strands** field. Enter 0 to have RMxprt auto-design this value.
9. Enter the thickness of the double-sided wire wrap in the **Wire Wrap** field. Enter 0 to automatically obtain this value from the wire library.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



10. Select the **Wire Size**:
    a. Click the button for **Wire Size**.
    b. In the **Wire Size** window that appears, select a value from the **Wire Diameter** drop-down menu.
    c. Select a wire gauge from the **Gauge** drop-down menu. You can select from the following options:

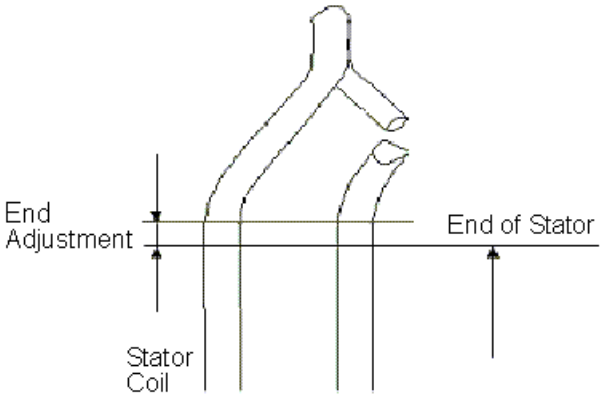
< *number* >	You can select a specific gauge number. When you select a gauge number, the **Wire Diameter** field is automatically updated.
USER	This option allows you to manually enter the **Wire Diameter**. This is useful when you want to enter a diameter that does not correspond to a particular wire gauge.
AUTO	This option sets the **Wire Diameter** to zero, and RMxprt automatically calculates the optimal value. The diameter information is then written to the output file when you analyze the design.
MIXED	This option allows you to define a conductor that is made of different size wires. For example, a single conductor may consist of five wires: three wires with a diameter of 0.21 mm and two wires with a diameter of 0.13 mm.

    The gauge number is based on AWG settings. You can create your own wire table using **Machine > Wire**, and then you can select this wire table using the **Tools** > **Options** > **General Options** > **Machines** option. When you are done setting the wire size, click **OK** to close the **Wire Size** window and return to the **Properties** window. Click the **End/Insulation** tab. Select or clear the **Input Half-turn Length** check box. Do one of the following:

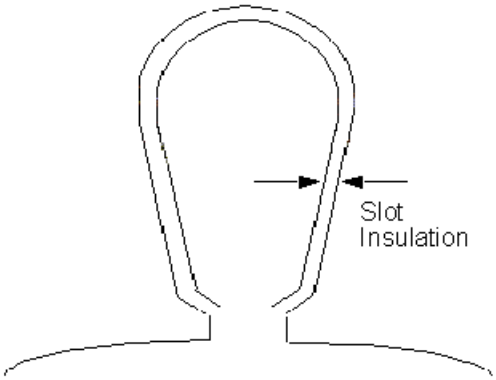
- If you selected **Input Half-turn Length**, then enter the half-turn length of the armature winding in the Half Turn Length field.
- If you cleared **Input Half-turn Length**, then enter the end length adjustment of the stator coils in the **End Adjustment** field. The end adjustment is the distance one

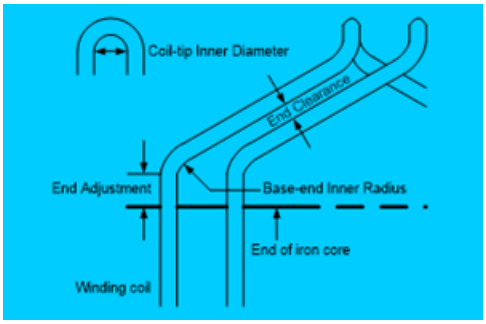
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

end of the conductor extends vertically beyond the end of the stator.

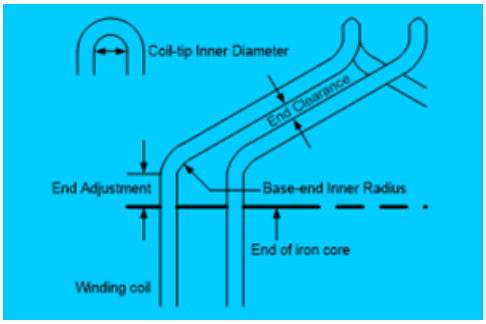


11. Enter the inner radius of the base corner in the **Base Inner Radius** field.
12. Enter the inner diameter of the coil tip in the **Tip Inner Diameter** field.
13. Enter the distance between two rotor coils in the **End Clearance** field.
14. Enter the thickness of the slot liner insulation in the **Slot Liner** field.





Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



15. Enter the thickness of the wedge insulation in the **Wedge Thickness** field.
16. Enter the thickness of the insulation layer in the **Layer Insulation** field.
17. Enter the limited slot fill factor for the wire design in the **Limited Fill Factor** field. The slot fill factor is the ratio between the cross-sectional area of all conductors in one slot and the entire slot area.
18. Select the type of equalizer connection from the **Equalizer Connection** drop-down menu. Select from **None**, **Half**, or **Full**.
19. Click **OK** to close the **Properties** window.

Defining Different Size Wires for a Universal Motor Rotor Winding

Use the **Gauge** option if you have a conductor that is made up different size wires.

To define different size wires:

1. In the **Wire Size** window, select **MIXED** from the **Gauge** drop-down menu.
2. Select either **Round** or **Rectangular** as the **Wire Type**.
3. Enter the appropriate wire data in the table:
    - For a round wire:
        - Enter the **Diameter** in the table.
        - Enter a **Number** in the table to specify how many of the conductor's wires have this diameter.
    - For a rectangular wire:
        - Enter the **Width** of the wire in the table.
        - Enter the **Thickness** of the wire in the table.
        - Enter the **Fillet** value in the table.
        - Enter a **Number** in the table to specify how many of the conductor's wires have this data.
4. Click **Add** to add the new wire data.
5. Repeat steps 3 and 4 for each size wire you want to add.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. When you are finished defining the wires, click **OK** to close the **Wire Size** window and return to the RMxprt **Properties** window.

> **Note:** For example, if one conductor is made up of five wires, and three of those wires have a diameter of 0.21 mm, and the other 2 have a diameter of 0.13 mm, then the mixed wire size table will have two lines. The first line will list Diameter = 0.21 and Number = 3. The second line will list Diameter = 0.13 and Number = 2. An equivalent wire diameter is displayed as Wire Size value in the Winding tab in the Properties window.

Rotor Winding Data for Universal Motors

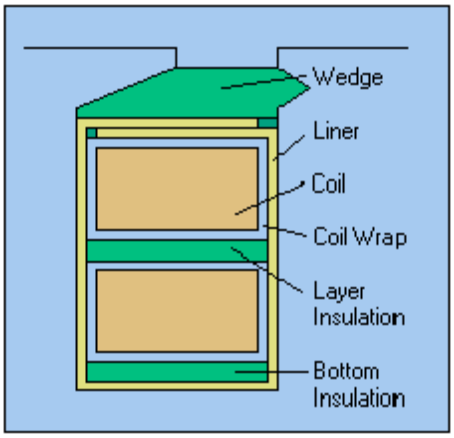
To access the stator winding data, double-click the **Machine** > **Rotor** > **Winding** entry in the Project Manager tree.

The **Rotor Winding Data Properties** window contains the following fields:

Winding tab	**Winding Type**	The type of rotor winding. Click the button to open the **Winding Type** window and choose from **Lap**, **Wave**, and **Frog Leg**.
	Multiplex Number	Single, double, or triple windings (**1**, **2**, or **3**)
	Virtual Slots	The number of virtual slots per real slot
	Conductors per Slot	The number of conductors per rotor slot (0 for auto-design)
	Coil Pitch	The coil pitch measured in number of slots
	Number of Strands	The number of wires per conductor (0 for auto-design)
	Wire Wrap	The thickness of the double-sided wire wrap (0 to automatically obtain this value from the wire library)
	Wire Size	The diameter of the wire (0 for auto-design). Click the button to open the **Wire Size** window where you can specify units, wire type, diameter, and gauge.
End/Insulation tab	**Input Half-turn Length**	Select or clear this check box to specify whether or not you want to

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	enter the half-turn length. When this check box is selected, the **Half Turn Length** field appears the next time you open the **Properties** window. When this check box is selected, the **End Adjustment** field appears instead.
Half Turn Length	The half-turn length of the armature winding
End Adjustment	The end length adjustment of the rotor coils
Base Inner Radius	The inner radius of the base corner
Tip Inner Diameter	The inner diameter of the coil tip
End Clearance	The end clearance between two adjacent coils
Slot Liner	The thickness of the slot liner insulation
Wedge Thickness	The thickness of the wedge insulation
Layer Insulation	The thickness of the insulation layer
Limited Fill Factor	The limited slot fill factor for the wire design. The slot fill factor is the ratio between the cross-sectional area of all conductors in one slot and the entire slot area.
Equalizer Connection	The connection type of the equalizer. Select from **None**, **Half**, or **Full**.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining the Commutator and Brush for a Universal Motor

The commutator allows current transfer between DC terminals or brushes and the rotor coils, providing the current to the system as a function of rotation. Due to the action of the commutator, the corresponding magnetic field has a fixed distribution with respect to the stator.

To define the commutator and brush pairs:

1. To open the **Commutator Data Properties** window, double-click the **Machine** > **Commutator** entry in the Project Manager tree. (You can also enter values in the **Properties** section of the Desktop without opening a separate window.)
2. Click the **Commutator** tab.
3. Select **Cylinder** or **Pancake Type** as the **Commutator Type**.

   > **Note:** When you place the mouse cursor over the commutator type, an outline of the commutator appears.

4. For **Cylinder** commutators, do the following:
   a. Enter the **Commutator Diameter**.
   b. Enter the **Commutator Length**.
5. For **Pancake** commutators, do the following:
   a. Enter the **Outer Diameter**.
   b. Enter the **Inner Diameter**.
6. Enter the thickness of the insulation between two consecutive commutator segments in the **Commutator Insulation** field.
7. Click the **Brush** tab.
8. Enter the **Brush Width**.
9. Enter the **Brush Length**.
10. Enter the number of brush pairs when using a wave armature winding in the **Brush Pairs** field.
11. Enter the angle of displacement from the neutral axis, in mechanical degrees, in the **Brush Displacement** field.

    > **Note:** The brush displacement is positive for the counter-clockwise direction. For example, if the rotor turns clockwise and the brush displacement is also clockwise, then the angle is negative; if the rotor turns clockwise but the brush displacement is counter-clockwise, then the angle is positive.

12. Enter the voltage drop across one brush pair in the Brush Drop field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13. Enter the mechanical pressure of the brushes as they press against the commutator in the **Brush Press** field.
14. Enter the **Frictional Coefficient** of the brush.

> **Note:** If the **Friction Loss** field is used in the **General** window, the **Brush Press** and **Frictional Coefficient** fields will be hidden in the **Commutator/Brush** window. These fields are shown only when the **Friction Loss** field in the **General** window is set to zero.

15. Click **OK** to close the **Properties** window.

Commutator and Brush Data for Universal Motors

To access the commutator and brush data, double-click the **Machine** > **Commutator** entry in the Project Manager tree.

The **Commutator Data Properties** window contains the following fields:

Commutator tab	**Commutator Type**	The type of commutator; click the button to open the **Select Commutator Type** window and select from **Cylinder** or **Pancake**.
	Commutator Diameter	For a **Cylinder** commutator type, defines the diameter of the commutator.
	Commutator Length	For a **Cylinder** commutator type, defines the length of the commutator.
	Outer Diameter	For a **Pancake** commutator type, defines the outer diameter of the commutator.
	Inner Diameter	For a **Pancake** commutator type, defines the inner diameter of the commutator.
	Commutator Insulation	Thickness of the insulation between the two commutator bars
Brush tab	**Brush Width**	Width of the brush
	Brush Length	Length of the brush
	Brush Pairs	Number of brush pairs
	Brush Displacement	Displacement of the brush from the neutral position, in mechanical degrees (positive for anti-rotating direction)
	Brush Drop	Voltage drop across a one-pair brush
	Brush Press	Brush press per unit area(Available only when **Frictional Loss** is set to zero for the machine)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Frictional Coefficient	Frictional coefficient of the brush (Available only when **Frictional Loss** is set to zero for the machine.)

Defining the Shaft Data for a Universal Motor

To define the shaft:

1. To open the **Shaft Data Properties** window, double-click the **Machine** > **Shaft** entry in the Project Manager tree. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Select or clear the **Magnetic Shaft** check box to specify whether or not the shaft is to be made of magnetic material.
3. Click **OK** to close the **Properties** window.

Shaft Data for Universal Motors

To access the shaft data, double-click the **Machine** > **Shaft** entry in the Project Manager tree.

The **Shaft Data Properties** window contains the following fields:

Magnetic Shaft	Select or clear this check box to indicate whether or not the shaft is made of magnetic material.

Setting Up Analysis Parameters for a Universal Motor

To define the solution data:

1. To open the **Solution Setup** window, right-click **Analysis** in the Project Manager tree, and click **Add Solution Setup**.
2. Click the **General** tab. The **Operation Type** is automatically set to **Motor** for this machine type.
3. Select the **Load Type** used in the motor from the following options:

Const Speed	The speed remains constant in the motor.
Const Power	The output power remains constant in the motor.
Const Torque	The torque remains constant regardless of the speed. In this case, $T_{load} = T_{rated}$, given by the output power divided by the given rated speed.
Linear Torque	The torque increases linearly with speed. In this case, $T_{load} = T_{rated} * (n/n_{rated})$ where T_{rated} is given by the output power divided by the given rated speed.
Fan Load	The load varies nonlinearly with speed. In this case, $T_{load} = T_{rated} * (n/n_{rated})^2$ where T_{rated} is given by the output power divided by the given rated speed.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Enter the output power developed at the shaft of the motor in the **Rated Output Power** field.
5. Enter the RMS line-to-line voltage in the **Rated Voltage** field.
6. Enter the desired output speed of the motor at the load point in the **Rated Speed** field.
7. Enter the temperature at which the system functions in the **Operating Temperature** field.
8. Click the **Universal Motor** tab.
9. Enter the **Frequency**, and select the units.
10. Click **OK** to close the **Solution Setup** window.

Solution Data for Universal Motors

To access the solution data, right-click **Analysis** in the project tree, and click **Add Solution Setup**. For this machine type, there is only one tab, the **General** tab.

The **Solution Setup** window contains the following fields:

Operation Type	**General** tab. The operation type is automatically set to **Motor** for this machine type.
Load Type	**General** tab. Select from **Const Speed**, **Const Power**, **Const Torque**, **Linear Torque**, and **Fan Load**. The default is **Const Power**.
Rated Output Power	**General** tab. Type a value for the rated output voltage, and select the units.
Rated Voltage	**General** tab. Type a value for the rated voltage, and select the units.
Rated Speed	**General** tab. Type a value for the rated speed, and select the units.
Operating Temperature	**General** tab. Type a value for the operating temperature, and select the units.
Frequency	**Universal Motor** tab. Enter a frequency in the **Frequency** field, and select the units.

General DC Machines

After you have selected **DC Machine** as your model type, enter the motor data to define the following:

- General data, such as the output power, rated voltage, speed, and machine type (motor or generator)
- Stator data, such as its associated pole dimensions, type of steel, and pole magnet specifications
- Stator field data, such as shoe and pole insulation, dimensions, and winding information
- Rotor data, such as the slot types and dimensions, rotor diameter, lamination, and wire specifications

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Commutator and brush data, such as the commutator type and dimensions and brush length
- Shaft data
- Solution data

By option you can insert or remove the following for a DC machine:

- Add a machine housing
- Compensating data, added under the stator
- Commutating data, added under the stator
- Vent data, added under the rotor
- Shunt data, added under the stator field
- Series data, added under the stator field

Analysis Approach for General DC Machines

For a Direct-Current (DC) Electric Machine Design, either a generator or motor, the rotor is equipped with a distributed winding – called armature winding – that is connected to a commutator revolving together with the rotor.

The stator is equipped with *p* pairs of poles, which are excited by *p* pairs of shunt and/or series windings. A shunt winding may be separately excited or self-excited. The separately excited shunt winding is excited by a separate DC voltage source. The self-excited shunt winding is excited by the terminal voltage of the armature winding and is connected in parallel with the armature winding. A series winding is connected in series with the armature winding. If both self-excited shunt and series windings are mounted on the stator poles, RMxprt assumes that the armature winding connects the series winding in series first, then connects the shunt winding in parallel.

A system of brushes is kept in permanent electrical contact with the commutator. When DC current is applied to the rotating armature winding via the brushes and commutator, a stationary magnetic field distribution is created with the axis electrically perpendicular to the axis of the field produced by the shunt and/or series windings. As a result, a torque is produced by the interaction of the fields produced by the armature and exciting currents. For these brush commutating machines, the commutator together with the brushes acts as a mechanical rectifier.

The field produced by the armature current is called armature reaction field. The armature reaction field causes poor commutating and poor voltage distribution along commutator bars. In order to improve commutating, commutating poles and winding can be equipped between two adjacent main poles and compensating winding can be equipped under main poles.

The performance of a DC machine is computed by DC analysis.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DC Machine Operating as a Motor

The voltage equation of a DC motor is

$$U = E + (U_b + R_1 \cdot I_a)$$

where, U_b is the voltage drop of one-pair brushes, R_1 is the total series resistance of the armature branch, E is the back emf as given below:

$$E = C_{Ef} \cdot \omega \cdot I_f + C_{Es} \cdot \omega \cdot I_a$$

where C_{Ef} and C_{Es}, which depend on the saturation of the magnetic field, are the back-emf coefficients in ohm.s/rad, is the rotor speed in mechanical rad/s, and I_f and I_a are the exciting currents of the shunt and series windings, respectively.

For a given speed, armature current can be computed based on the terminal voltage U, as shown below:

$$I_a = \frac{U - U_b - C_{Ef} \cdot \omega \cdot I_f}{R_1 + C_{Es} \cdot \omega}$$

The shaft torque is computed from

$$T_2 = (C_{Tf} \cdot I_f + C_{Ts} \cdot I_a) \cdot I_a - T_{fw}$$

where C_{Tf} and C_{Ts} are the torque coefficients in Nm/A^2, which are numerically the same as C_{Ef} and C_{Es}, respectively. T_{fw} is the frictional and wind torque. The output power (mechanical power) is

$$P_2 = T_2 \cdot \omega$$

The input power (electrical power) is

$$P_1 = P_2 + (P_{fw} + P_{Cua} + P_b + P_{Fe})$$

where P_{fw}, P_{Cua}, P_b, and P_{Fe} are the frictional and wind loss, armature branch copper loss, brush drop loss, iron-core loss and shunt winding copper loss, respectively. The efficiency is

$$\eta = \frac{P_2}{P_1} \times 100\%$$

DC Machine Operating as a Generator

For a DC generator, the voltage equation is

$$U = E - (U_b + R_1 \cdot I_a)$$

$$E = C_{Ef} \cdot \omega \cdot I_f + C_{Es} \cdot \omega \cdot I_a$$

The performance is analyzed as follows

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$I_a = -\frac{U+U_b-C_{Ef}\cdot\omega\cdot I_f}{R_1-C_{Es}\cdot\omega}$$

$$T_1 = ((C_{Tf}\cdot I_f + C_{Ts}\cdot I_a)\cdot I_a) + T_{fw}$$

$$P_1 = T_1\cdot\omega$$

$$P_2 = P_1 - (P_{fw} + P_{Cua} + P_b + P_{Fe})$$

$$\eta = \frac{P_2}{P_1}\times 100\%$$

Defining a General DC Machine

The general procedure for defining a general DC machine is as follows:

1. Insert a DC machine into a new or existing project.
2. Double-click the **Machine** entry in the project tree to define the general data.
3. Double-click the **Machine** > **Stator** entry in the project tree to define the stator geometry.
4. Double-click the **Machine** > **Stator** > **Pole** entry in the project tree to define the stator pole dimensions.
5. Double-click the **Machine** > **Stator** > **Field** entry in the project tree to define the stator windings, conductors, and insulation data.
6. Double-click the **Machine** > **Rotor** entry in the project tree to define the general rotor geometry, the pole data, and the insulation data.
7. Double-click the **Machine** > **Rotor** > **Slot** entry in the project tree to define the rotor slot dimensions.
8. Double-click the **Machine** > **Rotor** > **Winding** entry in the project tree to define the rotor conductors and windings.
9. Double-click the **Machine** > **Commutator** entry in the project tree to define the commutator and brush data.
10. Double-click the **Machine** > **Shaft** entry in the project tree to define the magnetism of the shaft.
11. Optionally, right-click the **Machine** entry in the project tree and click **Insert Housing** to add a machine housing.
12. Right-click **Analysis** in the project tree, and click **Add Solution Setup** to define the solution data.
13. Choose **File > Save** to save the project.
14. Choose **RMxprt** > **Analyze** to analyze the design.

> **Note:** When you place the cursor over an entry field, a brief description of that field appears in the status bar at the bottom of the **RMxprt** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Once analyzed, the model can be viewed in the Maxwell 2D Modeler, or it can be used to create a new Maxwell 2D project, and a new Maxwell 3D design.

Refer to the *DC Machine* application note, on the technical support page of the Ansys web site, for a specific example of a problem using a DC machine.

Defining the General Data for a General DC Machine

Use the **General** window to define the basic parameters of the DC motor, such as the power settings, speed, and rated voltage.

To define the general data:

1. To open the **General Data Properties** window, double-click the **Machine** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the number of poles for the machine in the **Number of Poles** field. This value is the total number of poles in the stator (or the number of pole pairs multiplied by two).
3. Enter the energy loss due to friction at the given speed in the **Frictional Loss** field.
4. Enter the wind loss due to air resistance measured at the reference speed in the **Wind Loss** field.
5. Enter the given speed in the **Reference Speed** field.
6. Click **OK** to close the **Properties** window.

General Data for General DC Machines

To access the general data, double-click the **Machine** entry in the project tree.

The **General Data Properties** window for a three-phase induction motor contains the following fields:

Machine Type	The machine type you selected when inserting a new RMxprt design (DC Machine)
Number of Poles	The number of poles the machine contains
Frictional Loss	The frictional energy loss (due to friction) measured at the reference speed
Wind Loss	The wind loss (due to air resistance) measured at the reference speed
Reference Speed	The given speed of reference

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining the Stator Data for a General DC Machine

Use the **Stator Properties** windows to define the stator dimensions, slots, windings, and conductors.

The stator is the outer lamination stack where the polyphase voltage windings reside.

To define the general stator data:

1. To open the **Stator Data Properties** window, double-click the **Machine** > **Stator** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the maximum diameter for a polygon-type frame in the **Frame Outer Diameter** field.
3. Enter the minimum outer width for a polygon-type frame in the **Frame Overall Width** field.
4. Enter the **Frame Thickness**.
5. Enter the **Frame Length**.
6. Select a steel type for the frame:
    a. Click the button for **Frame Material**.
    b. In the **Select Definition** window that appears, select a steel type from the list, or define a new steel type.
    c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.
7. Select a **Pole Type**:
    a. Click the button.
    b. In the **Select Pole Type** window that appears, click a button to specify the desired field type (either 1 or 2).
    c. Click **OK** to close the **Select Pole Type** window and return to the Properties window.
8. Enter the length of the stator main pole in the **Pole Length** field.
9. Enter the effective magnetic length for the stator main pole in the **Pole Stacking Factor** field.
10. Select a steel type for the stator main pole:
    a. Click the button for **Pole Material**.
    b. In the **Select Definition** window that appears, select a steel type from the list, or define a new steel type.
    c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.
11. Enter the thickness of the pole press boards in the **Press Board Thickness** field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12. If the pole press board is made of magnetic material, then select the **Magnetic Press Board** check box.
13. Click **OK** to close the **Properties** window.

Stator Data for General DC Machines

To access the general stator data, double-click the **Machine** > **Stator** entry in the project tree.

The **Stator Data Properties** window contains the following fields:

Frame Outer Diameter	Maximum diameter for a polygon-type frame
Frame Overall Width	Minimum outer width for a polygon-type frame
Frame Thickness	Thickness of the frame
Frame Length	Length of the frame
Frame Material	Steel type of the frame; click the button to open the **Select Definition** window.
Pole Type	Pole type of the stator; click the button to open the **Select Pole Type** window and select from the following two types: **1** and **2**.
Pole Length	Length of the stator main pole
Pole Stacking Factor	Stacking factor of the stator main pole
Pole Material	Steel type of the stator main pole. Click the button to open the **Select Definition** window
Press Board Thickness	Thickness of the pole press boards
Magnetic Press Board	Designates if the pole press board is made of magnetic material

Defining the Stator Pole for a General DC Machine

The rotor pole drives the electromagnetic field which is coupled with the stator windings. Use the **Stator Pole Data Properties** window to define the stator pole.

To define the rotor pole:

1. To open the **Stator Pole Data Properties** window, double-click the **Machine** > **Stator** > **Pole** entry in the Project Manager tree. (You can also enter values in the **Properties** pane

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of the desktop without opening a separate window.)

> **Note:** For a two-pole machine, a pole embrace of 0.75 yields a magnet with a span of 135 degrees (based on 0.75*180 degrees).

2. Enter the inner diameter at the pole center in the Dmin field.
3. Enter the diameter at the pole tip in the Dmax field.
4. Enter the width of the pole arc with a uniform air gap in the Bp0 field. For an eccentric air gap, enter 0.
5. Enter the width of the pole tip in the Bp1 field.
6. Enter the maximum width of the pole shoe in the Bp2 field. This field is only available for a Pole Type of 1.
7. Enter the minimum width of the pole shoe in the Bp3 field. This field is only available for a Pole Type of 1.
8. Enter the size of the pole shoe fillet in the Rp0 field. This field is only available for a Pole Type of 2.
9. Enter the fillet between the pole shoe and the pole body in the Rp1 field. This field is only available for a Pole Type of 2.
10. Enter the pole shoe height in the Hp field.
11. Enter the pole body width in the Bm field.
12. Click **OK** to close the **Properties** window.

Stator Pole Data for General DC Machines

To access the stator pole data, double-click the **Machine** > **Stator** > **Pole** entry in the Project Manager tree.

The **Rotor Pole Data Properties** window contains the following fields:

Dmin	Inner diameter at the pole center
Dmax	Diameter at the pole tip
Bp0	Width of the pole arc with a uniform air gap (0 for an eccentric air gap).
Bp1	The width of the pole tip
Bp2	Maximum width of the pole shoe; for pole type 1.
Bp3	Minimum width of the pole shoe; for pole type 1.
Rp0	Pole shoe fillet; for pole type 2.
Rp1	Fillet between the pole shoe and the pole body; for pole type 2.
Hp	Height of the pole shoe
Bm	Width of the pole body

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining the Stator Field Data for a General DC Machine

To define the stator windings and insulation data:

1. To open the **Stator Field Properties** window, double-click the **Machine** > **Stator** > **Field** entry in the Project Manager tree. (You can also enter values in the **Properties** pane of the desktop without opening a separate window.)
2. Enter the thickness of the insulation under the pole shoe in the **Shoe Insulation** field.
3. Enter the thickness of the insulation at the pole body side in the **Pole Insulation** field.
4. Enter the minimum gap in the **Winding Clearance** field. The winding clearance is one of the following: the minimum gap between two field windings, or the minimum gap between a field winding and a commutating winding.
5. Enter the thickness of the insulation between the shunt winding and the series winding in the **Winding Insulation** field.
6. Select the type of exciting of the series winding to the shunt winding from the **Compound Exciting Mode** drop-down menu. The options are **Cumulative** and **Differential**.
7. Click **OK** to close the **Properties** window.

Stator Field Data for General DC Machines

To access the stator field data, double-click the **Machine** > **Stator** > **Field** entry in the Project Manager tree.

The **Stator Field Data Properties** window contains the following fields:

Shoe Insulation	Thickness of the insulation under the pole shoe
Pole Insulation	Thickness of the insulation at the pole body side
Winding Clearance	Minimum air gap between two field windings, or the minimum gap between a field winding and a commutating winding
Winding Insulation	Thickness of the insulation between the shunt winding and the series winding
Compound Exciting Mode	Cumulative exciting or differential exciting of the series winding to the shunt winding. Select **Cumulative** or **Differential** from the drop-down menu.

Shunt Data for General DC Machines

By option you can insert or remove a shunt from a General DC Machine. If you insert a shunt, it appears in the Project Manager tree under the stator field data.

To insert a shunt:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Right-click on the **Field** icon under the stator in the project tree to display the drop-down menu.
2. Click **Insert Shunt**.

The **Shunt** icon appears under the **Field** icon.

To remove an existing shunt:

1. Right-click on the **Field** icon under the Stator in the Project Manager tree to display the drop-down menu.
2. Click **Remove Shunt**.

The shut is removed from the Project Manager tree.

The Shunt data for a General DC Machine contains the following fields.

Winding type	Specified as **Round**, **Cylinder** coil, or **Edgewise** coil, by clicking the button to display the Winding Type selection window.
Parallel branches	Number of parallel branches
Conductors per pole	Number of conductors per pole; 0 for auto-design. Odd number of strands for the case where the input and output leads are on different sides.
Number of strands	Number of strands (number of wires per conductor); 0 for auto-design.
Wire wrap	Double side wire wrap thickness. 0 for auto-pickup in the wire library.
Wire size	Click the button to display the Wire Size selection window.
Axial Clearance	Axial gap between field winding and pole body on the inner coil
Limited cross width	Limited cross-section width for winding design or arrangement; 0 for available maximum area.
Limited cross height	Limited cross-section height for winding design or arrangement; 0 for available maximum area.
Winding fillet	Defines the size of the winding fillet

Series Data for General DC Machines

By option, you can insert or remove a series from a General DC Machine. If you insert a series, it appears in the Project Manager tree under the stator field data.

To insert a series:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Right-click on the **Field** icon under the stator in the Project Manager tree to display the drop-down menu.
2. Click **Insert Series**.

The Series icon appears under the field icon.

To remove an existing series:

1. Right-click on the Field icon under the stator in the Project Manager tree to display the drop-down menu.
2. Click **Remove Series**.

The series is removed from the Project Manager tree.

The Series data for a General DC Machine contains the following fields.

Winding type	Specified as Round, Cylinder coil, or Edgewise coil, by clicking the button to display the Winding Type selection window.
Parallel branches	Number of parallel branches
Conductors per pole	Number of conductors per pole; 0 for auto-design. Odd number of strands for the case where the input and output leads are on different sides.
Number of strands	Number of strands (number of wires per conductor); 0 for auto-design.
Wire wrap	Double side wire wrap thickness; 0 for auto-pickup in the wire library.
Wire size.	Click the button to display the Wire Size selection window.
Axial Clearance	Axial gap between field winding and pole body on the inner coil
Limited cross width	Limited cross-section width for winding design or arrangement; 0 for available maximum area.
Limited cross height	Limited cross-section height for winding design or arrangement; 0 for available maximum area.
Winding fillet	

Compensating Data for General DC Machines

By option, you can insert or remove Compensating for a General DC Machine.

To insert compensating:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Right-click on the Stator icon to display the pop-up menu.
2. Click **Insert Compensating**.

To remove an existing Compensating:

1. Right-click on the Stator icon to display the pop-up menu.
2. Click **Remove Compensating**.

To access the data for compensating inserted to a General DC Machine, double-click on the **Machine** > **Rotor** > **Compensating** item in the project tree.

The **Compensating** properties window contains the following fields.

Slots per pole	Number of slots per pole for the compensating winding
Bc0	Opening width of the compensating slots
Hc0	Opening height of the compensating slots
Bc2	Width of the compensating slots
Hc2	Height of the compensating slots
Parallel branches	Number of parallel branches.
Conductors per slot	Number of conductors per slot for the compensating windings
Number of strands	Number of strands (number of wires per conductor), 0 for auto-design.
Wire wrap	Double-side wire wrap thickness, 0 for auto pickup in the wire library
Rectangle wire	Whether to use round (the default) or rectangle wire
Wire size	Click the button to display the Wire Size window to specify the wire diameter and gauge.
Slot liner	Insulation slot liner thickness
End adjustment	One side end length adjustment of a conductor

Commutating Data for General DC Machines

Commutating must be inserted under the stator by right-clicking on the stator icon to display the pop-up menu, and click **Insert Commutating** command. This command also inserts an icon in the project tree for an associated winding.

To remove an existing Commutating (and associated winding), right-click on the stator icon to display the pop-up menu and click **Remove Commutating**. This removes the commutating and the associated winding.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** These functions are distinct from the general Commutator data associated with rotor.

Pole width	Width of the commutating poles
Pole height	Height of the commutating poles
Pole length	Length of the commutating poles
Shoe width	Shoe width of the commutating poles
Shoe height	Shoe height of the commutating poles
Second air gap	Length of the second air gap between the commutating pole and the frame.
Pole stacking factor	Stacking factor for the commutating poles
Pole material	Steel type of the commutating poles. Click the button to display the Select Definition window
Pole insulation	Thickness of insulation on the pole body side

Winding Data for Commutating

If you have inserted commutating for a General DC machine, an additional winding icon appears in the project tree for the associated winding.

Winding type	Specified as **Round**, **Cylinder** coil, or **Edgewise** coil, by clicking the button to display the Winding Type selection window.
Parallel branches	Number of parallel branches
Conductors per pole	Number of conductors per pole; 0 for auto-design. Odd number of strands for the case where the input and output leads are on different sides.
Number of strands	Number of strands (number of wires per conductor); 0 for auto-design.
Wire wrap	Double side wire wrap thickness; 0 for auto-pickup in the wire library.
Wire size…	Click the button to display the Wire Size selection window.
Axial Clearance	Axial gap between field winding and pole body on the inner coil
Limited cross width	Limited cross-section width for winding design or arrangement; 0 for available maximum area
Limited cross	Limited cross-section height for winding design or arrangement; 0 for available maximum area

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

height	
Winding fillet	Defines the size of the winding fillet

Defining the Rotor Data for a General DC Machine

The rotor is equipped with slots containing copper conductors that are connected to the commutator. The commutator acts as a mechanical rectifier in the motor.

The rotor consists of copper bars in which current is induced by the magnetic fields produced by the stator windings. In the project tree, double-click **Machine** > **Rotor** and **Machine** > **Rotor** > **Pole** to define the rotor and the pole.

To define general stator data:

1. To open the **Rotor Data Properties** window, double-click the **Machine** > **Rotor** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the stacking factor for the rotor core in the **Stacking Factor** field. This value relates to the effective magnetic length of the core, and ranges from 0 to 1. It is defined as the total length minus the total insulation from the laminations, divided by the total length. A value of 1 indicates that the rotor is not laminated.
3. Enter the number of slots in the rotor core in the **Number of Slots** field.
4. Select a **Slot Type**:
   a. Click the button.
   b. In the **Select Slot Type** window that appears, click a button to select the desired slot type (1, 2, 3, 4, 5, or 6). Though slots **3** and **4** are visually similar, they differ in how the edges are constructed. Slot **3** has a tapered edge leading from the slot opening to the main slot body. Slot **4** has a rounded edge at the same location, where the quantity **Hr1** defines the radius of the corner slot. TIP: When you run the mouse over each option, the diagram changes to show that pole type.
   c. Click **OK** to close the **Select Slot Type** window and return to the **Properties** window.

   Optionally, check **User Defined Slot** if you wish to define the slot dimensions using the **Slot Editor**.
5. Enter the number of lamination sectors in the **Lamination Sectors** field.
6. Enter the outer diameter of the rotor core in the **Outer Diameter** field.
7. Enter the inner diameter of the rotor core in the **Inner Diameter** field.
8. Enter the length of the rotor core in the **Length** field.
9. Select a **Steel Type** for the rotor core:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Click the button for **Steel Type**.
b. In the **Select Definition** window that appears, select a steel type from the list, or define a new steel type.
c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.

10. Enter the thickness of the pole press boards in the **Press Board Thickness** field.
11. Enter the number of slots in the skew width in the **Skew Width** field.
12. Click **OK** to close the **Properties** window.

Rotor Data for General DC Machines

To access the general rotor data, double-click the **Machine** > **Rotor** entry in the Project Manager tree.

The **Rotor Data Properties** window contains the following fields:

Stacking Factor	Effective magnetic length of the rotor core
Number of Slots	Number of slots the rotor core contains
Slot Type	Type of slots in the rotor core. Click the button to open the **Select Slot Type** window.
Lamination Sectors	Number of lamination sectors
Outer Diameter	Outer diameter of the rotor core
Inner Diameter	Inner diameter of the rotor core
Length	Length of the rotor core
Steel Type	Steel type of the rotor core. Click the button to open the **Select Definition** window.
Press Board Thickness	Thickness of the pole press boards
Skew Width	Skew width measured in slot number

Defining the Rotor Slots for a General DC Machine

To define the physical dimensions of the rotor slots:

1. To open the **Rotor Slot Data Properties** window, double-click the **Machine** > **Rotor** > **Slot** entry in the Project Manager tree. (You can also enter values in the **Properties** pane of the desktop without opening a separate window.)
2. Optionally, to automatically design the dimensions of slots **Hs2**, **Bs1**, and **Bs2**, select the **Auto Design** check box. Using this option causes the software to converge to a flux density value of 1.5 Tesla in the rotor teeth.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Enter the available slot dimensions. The following dimensions may be listed, depending on the **Slot Type** selected and depending on whether or not **Auto Design** is selected:
   - **Hs0** Always available
   - **Hs1** Always available
   - **Hs2** Available only when **Auto Design** is cleared. When **Auto Design** is selected, this slot dimension is determined automatically.
   - **Bs0** Always available
   - **Bs1** Available only when **Auto Design** is cleared. When **Auto Design** is selected, this slot dimension is determined automatically. When **Parallel Tooth** is selected, this slot dimension is determined based on the value entered in the **Tooth Width** field.
   - **Bs2** Available only when **Auto Design** is cleared. When **Auto Design** is selected, this slot dimension is determined automatically. When **Parallel Tooth** is selected, this slot dimension is determined based on the value entered in the **Tooth Width** field.
   - **Rs** Always available
4. Click **OK** to close the **Properties** window.

Rotor Slot Data for General DC Machines

To access the rotor slot data, double-click the **Machine** > **Rotor** > **Slot** entry in the Project Manager tree.

The **Rotor Slot Data Properties** window contains the following fields:

Auto Design	Select or clear this to enable or disable auto-design of slots **Hs2**, **Bs1**, and **Bs2**. When this check box is selected, only two other fields appear in the window: **Hs0** and **Bs0**.
Parallel Tooth	Select this to design **Bs1** and **Bs2** based on the tooth width. When this check box is selected, the **Bs1** and **Bs2** fields are removed, and the **Tooth Width** field is added.
Tooth Width	The tooth width for the parallel tooth, on which **Bs1** and **Bs2** are designed
Hs0	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected)
Hs1	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected)
Hs2	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected)
Bs0	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected)
Bs1	A slot dimension (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

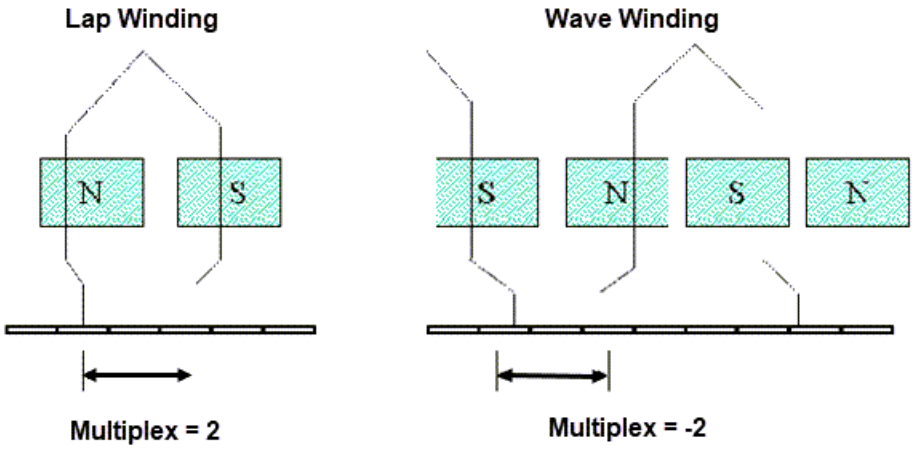
Bs2 A slot dimension (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected)

Rs A slot dimension. (see the diagram shown in the modeling window when **Machine** > **Rotor** > **Slot** is selected). Rs is added when the slot type is 3 or 4.

Defining the Rotor Windings and Conductors for a General DC Machine

To define the rotor windings, wires, and conductors:

1. To open the **Rotor Slot Winding Properties** window, double-click the **Machine** > **Rotor** > **Winding** entry in the Project Manager tree. (You can also enter values in the **Properties** pane of the Desktop without opening a separate window.)
2. Click the **Winding** tab.
3. Select a **Winding Type**:
    a. Click the button for **Winding Type**.
    b. In the **Winding Type** window that appears, select from one of the following three types of winding:
        - **Lap**
        - **Wave**
        - **Frog Leg**
4. Enter the number of windings in the **Multiplex Number** field (1 for a single winding, 2 for double windings, 3 for triple windings). For a lap winding, the multiplex number is the number of commutators between the start and end of one winding, and the number of parallel branches is equal to the number of poles multiplied by the multiplex number. For a wave winding, the number of parallel branches equals the multiplex number multiplied by two.

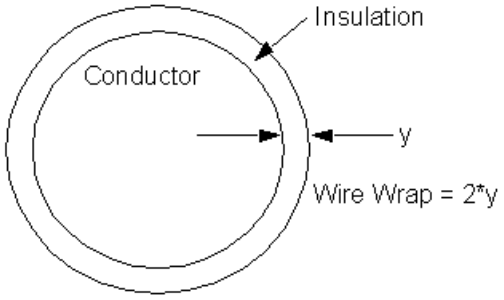


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Enter the number of virtual slots per each real slot in the **Virtual Slots** field. The rotor is assumed to have two layers of conductors, an upper and a lower layer. Each layer of conductors can have a number of windings, which are referred to as virtual slots.

   > **Note:** For example, the upper and lower layer can have two windings each, which would yield a virtual slot number of two; for a 12-slot machine, this would yield 24 commutation segments.

6. Enter the total number of conductors in each rotor slot in the **Conductors per Slot** field. This value is the number of turns per coil multiplied by the number of layers. This value is the total number of conductors in one real full rotor slot.
7. Enter the coil pitch, measured in number of slots, in the **Coil Pitch** field. The coil pitch is the number of slots separating one winding. For example, if a coil starts in slot 1 and ends in slot 6, it has a coil pitch of 5.
8. Enter the number of wires per conductor in the **Number of Strands** field. Enter 0 to have RMxprt auto-design this value.
9. Enter the thickness of the double-sided wire wrap in the **Wire Wrap** field. Enter 0 to automatically obtain this value from the wire library.



10. Select the **Wire Size**:
    a. Click the button for **Wire Size**.
    b. In the **Wire Size** window that appears, select a value from the **Wire Diameter** drop-down list.
    c. Select a wire gauge from the **Gauge** drop-down list. You can select from the following options:

< ***number*** **>**	You can select a specific gauge number. When you select a gauge number, the **Wire Diameter** field is automatically updated.
USER	This option allows you to manually enter the **Wire Diameter**. This is useful when you want to enter a diameter that does not correspond to a particular wire gauge.

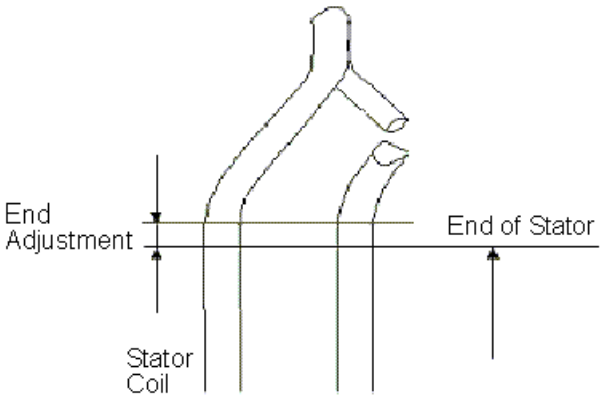
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AUTO	This option sets the **Wire Diameter** to zero, and RMxprt automatically calculates the optimal value. The diameter information is then written to the output file when you analyze the design.
MIXED	This option allows you to define a conductor that is made of different size wires. For example, a single conductor may consist of five wires: three wires with a diameter of 0.21 mm and two wires with a diameter of 0.13 mm.

The gauge number is based on AWG settings. You can create your own wire table using **Machine > Wire**, and then you can select this wire table using the **Tools** > **Options** > **General Options** > **Machines** function.

d. When you are done setting the wire size, click **OK** to close the **Wire Size** window and return to the **Properties** window.

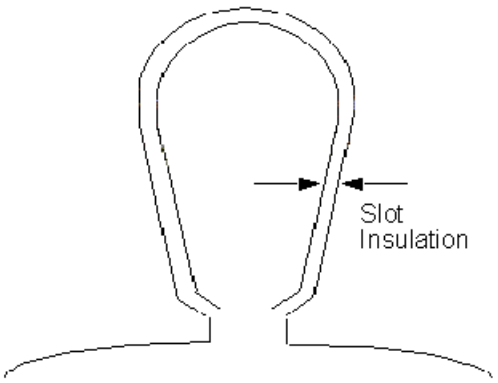
11. Click the **End/Insulation** tab.
12. Select or clear the **Input Half-turn Length** check box.
13. Do one of the following:
    - If you selected **Input Half-turn Length**, then enter the half-turn length of the armature winding in the Half Turn Length field.
    - If you cleared **Input Half-turn Length**, then enter the end length adjustment of the stator coils in the **End Adjustment** field. The end adjustment is the distance one end of the conductor extends vertically beyond the end of the stator.



14. Enter the inner radius of the base corner in the **Base Inner Radius** field.
15. Enter the inner diameter of the coil tip in the **Tip Inner Diameter** field.
16. Enter the distance between two rotor coils in the **End Clearance** field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

17. Enter the thickness of the slot liner insulation in the **Slot Liner** field.



18. Enter the thickness of the wedge insulation in the **Wedge Thickness** field.
19. Enter the thickness of the insulation layer in the **Layer Insulation** field.
20. Enter the limited slot fill factor for the wire design in the **Limited Fill Factor** field. The slot fill factor is the ratio between the cross-sectional area of all conductors in one slot and the entire slot area.
21. Select the type of equalizer connection from the **Equalizer Connection** drop-down menu. Select from **None**, **Half**, or **Full**.
22. Click **OK** to close the **Properties** window.

Defining Different Size Wires for a General DC Machine Rotor Winding

Use the **Gauge** option if you have a conductor that is made up different size wires.

To define different size wires:

1. In the **Wire Size** window, select **MIXED** from the **Gauge** drop-down list.
2. Select either **Round** or **Rectangular** as the **Wire Type**.
3. Enter the appropriate wire data in the table:
    - For a round wire:
        - Enter the **Diameter** in the table.
        - Enter a **Number** in the table to specify how many of the conductor's wires have this diameter.
    - For a rectangular wire:
        - Enter the **Width** of the wire in the table.
        - Enter the **Thickness** of the wire in the table.
        - Enter the **Fillet** value in the table.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Enter a **Number** in the table to specify how many of the conductor's wires have this data.

4. Click **Add** to add the new wire data.
5. Repeat steps 3 and 4 for each size wire you want to add.
6. When you are finished defining the wires, click **OK** to close the **Wire Size** window and return to the RMxprt **Properties** window.

> **Note:** For example, if one conductor is made up of five wires, and three of those wires have a diameter of 0.21 mm, and the other 2 have a diameter of 0.13 mm, then the mixed wire size table will have two lines. The first line will list Diameter = 0.21 and Number = 3. The second line will list Diameter = 0.13 and Number = 2. An equivalent wire diameter is displayed as Wire Size value in the Winding tab in the Properties window.

Rotor Winding Data for General DC Machines

To access the rotor winding data, double-click the **Machine** > **Rotor** > **Winding** entry in the Project Manager tree.

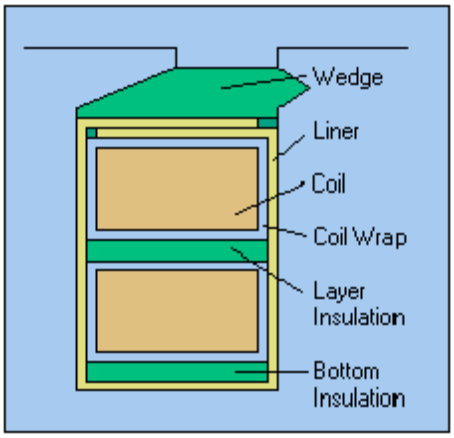
The **Rotor Winding Data Properties** window contains the following fields:

Winding tab	**Winding Type**	Type of rotor winding. Click the button to open the **Winding Type** window and choose from **Lap**, **Wave**, and **Frog Leg**.
	Multiplex Number	Single, double, or triple windings (**1**, **2**, or **3**)
	Virtual Slots	Number of virtual slots per real slot
	Conductors per Slot	Number of conductors per rotor slot (0 for auto-design)
	Coil Pitch	Coil pitch measured in number of slots
	Number of Strands	Number of wires per conductor (0 for auto-design)
	Wire Wrap	Thickness of the double-sided wire wrap (0 to automatically obtain this value from the wire library)
	Wire Size	Diameter of the wire (0 for auto-design). Click the button to open the **Wire Size** window where you can specify units, wire type, diameter,

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		and gauge.
End/Insulation tab	**Input Half-turn Length**	Select or clear this check box to specify whether or not you want to enter the half-turn length. When this check box is selected, the **Half Turn Length** field appears the next time you open the **Properties** window. When this check box is selected, the **End Adjustment** field appears instead.
	Half Turn Length	Half-turn length of the armature winding
	End Adjustment	End length adjustment of the rotor coils
	Base Inner Radius	Inner radius of the base corner
	Tip Inner Diameter	Inner diameter of the coil tip
	End Clearance	End clearance between two adjacent coils
	Slot Liner	Thickness of the slot liner insulation
	Wedge Thickness	Thickness of the wedge insulation
	Layer Insulation	Thickness of the insulation layer
	Limited Fill Factor	Limited slot fill factor for the wire design. The slot fill factor is the ratio between the cross-sectional area of all conductors in one slot and the entire slot area.
	Equalizer Connection	Connection type of the equalizer. Select from **None**, **Half**, or **Full**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Vent Data for General DC Machines

By option, you can insert or remove Vent data for general DC machines. If you have inserted a Vent, the icon appears under the rotor winding in the project tree.

To insert a vent:

1. Right-click on the rotor icon to display the pop-up menu.
2. Click **Insert Vent**.

To remove an existing vent:

1. Right-click on the Stator icon to display the pop-up menu.
2. Click **Remove Vent**.

The **Vent Data Properties** window contains the following fields.

Vent Ducts	Number of radial vent ducts
Duct Width	Width of radial vent ducts
Magnetic Spacer Width	Width of magnetic spacer which hold vent ducts. 0 for non-magnetic spacer.
Duct Pitch	Vent ducts
Holes per Row	Number of axial vent holes per row
Inner Hole Diameter	Diameter of vent holes in inner row
Outer Hole Diameter	Diameter of vent holes in outer row
Inner Hole Location	Center-to-center diameter of inner row hole vents
Outer Hole Location	Center-to-center diameter of outer row hole vents

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Banding Slots	Number of axial banding slots to tight the rotor winding
Width of Banding Slots	Width of axial banding slots
Depth of Banding Slots	Depth of axial banding slots

Defining the Commutator and Brush for a General DC Machine

The commutator allows current transfer between DC terminals or brushes and the rotor coils, providing the current to the system as a function of rotation. Due to the action of the commutator, the corresponding magnetic field has a fixed distribution with respect to the stator.

To define the commutator and brush pairs:

1. To open the **Commutator Data Properties** window, double-click the **Machine** > **Commutator** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Click the **Commutator** tab.
3. Select **Cylinder** or **Pancake Type** as the **Commutator Type**.

   > **Note:** When you place the mouse cursor over the commutator type, an outline of the commutator appears.

4. For **Cylinder** commutators, do the following:
   a. Enter the **Commutator Diameter**.
   b. Enter the **Commutator Length**.
5. For **Pancake** commutators, do the following:
   a. Enter the **Outer Diameter**.
   b. Enter the **Inner Diameter**.
6. Enter the thickness of the insulation between two consecutive commutator segments in the **Commutator Insulation** field.
7. Click the **Brush** tab.
8. Enter the **Brush Width**.
9. Enter the **Brush Length**.
10. Enter the number of brush pairs when using a wave armature winding in the **Brush Pairs** field.
11. Enter the angle of displacement from the neutral axis, in mechanical degrees, in the **Brush Displacement** field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** The brush displacement is positive for the counter-clockwise direction. For example, if the rotor turns clockwise and the brush displacement is also clockwise, then the angle is negative; if the rotor turns clockwise but the brush displacement is counter-clockwise, then the angle is positive.

12. Enter the voltage drop across one brush pair in the **Brush Drop** field.
13. Enter the mechanical pressure of the brushes as they press against the commutator in the **Brush Press** field.
14. Enter the **Frictional Coefficient** of the brush.

> **Note:** If the **Friction Loss** field is used in the **General** window, the **Brush Press** and **Frictional Coefficient** fields will be hidden in the **Commutator/Brush** window. These fields are shown only when the **Friction Loss** field in the **General** window is set to zero.

15. Click **OK** to close the **Properties** window.

Commutator and Brush Data for General DC Machines

To access the commutator and brush data, double-click the **Machine** > **Commutator** entry in the project tree.

The **Commutator Data Properties** window contains the following fields:

Commutator tab	**Commutator Type**	The type of commutator. Click the button to open the **Select Commutator Type** window and select from **Cylinder** or **Pancake**.
	Commutator Diameter	For a **Cylinder** commutator type, the diameter of the commutator
	Commutator Length	For a **Cylinder** commutator type, the length of the commutator
	Outer Diameter	For a **Pancake** commutator type, the outer diameter of the commutator
	Inner Diameter	For a **Pancake** commutator type, the inner diameter of the commutator
	Commutator Insulation	The thickness of the insulation between the two commutator bars
Brush tab	**Brush Width**	The width of the brush

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Brush Length	The length of the brush
Brush Pairs	The number of brush pairs
Brush Displacement	The displacement of the brush from the neutral position, in mechanical degrees (positive for anti-rotating direction).
Brush Drop	The voltage drop across a one-pair brush
Brush Press	The brush press per unit area. (Available only when **Frictional Loss** is set to zero for the machine.)
Frictional Coefficient	The frictional coefficient of the brush. (Available only when **Frictional Loss** is set to zero for the machine.)

Defining the Shaft Data for a General DC Machine

To define the shaft:

1. To open the **Shaft Data Properties** window, double-click the **Machine** > **Shaft** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Select or clear the **Magnetic Shaft** check box to specify whether or not the shaft is to be made of magnetic material.
3. Select or clear the **No Fan** check box to specify whether or not the machine contains a ventilation fan.
4. If you cleared the **No Fan** check box, then do the following:
    a. Enter the outer diameter of the ventilation fan in the **Fan Diameter** field.
    b. Enter the width of the fan blades in the **Blade Width** field.
5. Click **OK** to close the **Properties** window.

Access Shaft Data for General DC Machines

To access the shaft data, double-click the **Machine** > **Shaft** entry in the project tree.

The **Shaft Data Properties** window contains the following fields:

Magnetic Shaft	Select or clear this check box to indicate whether or not the shaft is made of magnetic material. When selected, the shaft is magnetic. Default is unchecked for PM_INTERIOR rotor type.
Frictional Loss	The frictional loss measured at the **Reference Speed**
Windage Loss or Power	The **Windage Loss** (or **Power** for wind power generators) measured at the **Reference Speed**.
Reference Speed	The speed at which the friction and windage losses are measured. Default is 3600 rpm for PM_INTERIOR and AXIAL_CAGE rotor types.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Up Analysis Parameters for a General DC Machine

To define the solution data:

1. To open the **Solution Setup** window, right-click **Analysis** in the project tree, and click **Add Solution Setup**.
2. Click the **General** tab. The **Operation Type** is automatically set to **Motor** for this machine type.
3. Select the **Load Type** used in the motor from the following options:

Const Speed	The speed remains constant in the motor.
Const Power	The output power remains constant in the motor.
Const Torque	The torque remains constant regardless of the speed. In this case, $T_{load} = T_{rated}$, given by the output power divided by the given rated speed.
Linear Torque	The torque increases linearly with speed. In this case, $T_{load} = T_{rated} * (n/n_{rated})$ where T_{rated} is given by the output power divided by the given rated speed.
Fan Load	The load varies nonlinearly with speed. In this case, $T_{load} = T_{rated} * (n/n_{rated})^2$ where T_{rated} is given by the output power divided by the given rated speed.

4. Enter the output power in the **Rated Output Power** field.
5. Enter the applied or output rated DC voltage in the **Rated Voltage** field.
6. Enter the given rated speed in the **Rated Speed** field.
7. Enter the temperature at which the system functions in the **Operating Temperature** field.
8. Click the **DC Machine** tab.
9. Select one of the following from the **Field Exciting Type** drop-down menu:
    - **Separately Excited**
    - **Self Excited**
10. Enter the **Exciting Voltage,** and select the units.
11. Enter the **Series Resistance**, and select the units.
12. To automatically obtained the Exciting Voltage and Series Resistance via the **Rated Speed**, rather than entering their values, then select the **Determined by Rated Speed** check box.
13. Click **OK** to close the **Solution Setup** window.

Solution Data for General DC Machines

To access the solution data, right-click **Analysis** in the project tree, and click **Add Solution Setup**. For this machine type, there is only one tab, the **General** tab.

The **Solution Setup** window contains the following fields:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Operation Type	**General** tab. Select **Motor** or **Generator** from the drop-down menu.
Load Type	On the **General** tab. For a motor, select from **Const Speed**, **Const Power**, **Const Torque**, **Linear Torque**, and **Fan Load**. The default is **Const Power**. For a generator, select from **Infinite Bus** and **Independent Generator**.
Rated Output Power	**General** tab. Type a value for the rated output voltage, and select the units.
Rated Voltage	**General** tab. Type a value for the rated voltage, and select the units.
Rated Speed	**General** tab. Type a value for the rated speed, and select the units.
Operating Temperature	**General** tab. Type a value for the operating temperature, and select the units.
Field Exciting Type	**DC Machine** tab. Select **Separately Excited** or **Self Excited** from the drop-down menu.
Determined by Rated Speed	Select this check box to automatically calculate the **Exciting Voltage** and the **Series Resistance** from the **Rated Speed**, rather than entering the values.
Exciting Voltage	Enter a voltage value in the field, and select the units from the drop-down menu.
Series Resistance	Enter a resistance value in the field, and select the units from the drop-down menu.

Claw-Pole Alternators

After you have selected **Claw-Pole Alternators** as your model type, enter the motor data to define the following:

- General data, such as the output power, rated voltage, and speed
- Stator data
- Stator slot data
- Stator winding data
- Rotor data, such as the slot types and dimensions, rotor diameter, and lamination
- Rotor pole data
- Shaft data
- Optional machine housing
- Solution data

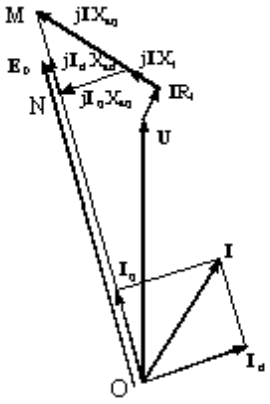
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Analysis Approach for Claw-Pole Alternators

Claw-pole alternators (or claw-pole synchronous generators) are widely used in auto industry. They receive mechanical energy at the shaft and transform it into electrical energy.

The stator of a claw-pole alternator is equipped with a polyphase winding. The rotor is comprised of claw poles with the same pole number as the stator winding. The claw poles of the rotor are magnetized by a cylinder winding and/or a cylinder permanent magnet. The spinning rotor creates a rotating magnetic field in the air gap, which produces induced voltage in the stator winding.

The performance of a claw-pole alternator is analyzed based on the frequency-domain phasor diagram, as shown in the figure below.



If a claw-pole alternator is equipped with a permanent magnet, the d-axis armature reactance X_{ad} and q-axis armature reactance X_{aq} are about constant. Otherwise, X_{ad} is a linearized nonlinear parameter, and X_{aq} is a linear parameter. The d-axis synchronous reactance X_d and q-axis synchronous reactance X_q are calculated directly from the following:

$X_d = X_1 + X_{ad}$

$X_q = X_1 + X_{aq}$

Rotor Equipped with an Excitation Winding

If the rotor is equipped with an excitation winding, the exciting current can be adjusted, and the d- and the q-axis currents are obtained based on the following process.

Take the input voltage *U* as the reference phasor, let the power factor angle be *f*, then the current phasor is

The phasor represented by **OM** can be expressed as

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\mathbf{OM} = \mathbf{U} + \mathbf{I}(R_1 + jX_q)$$

The phasor represented by **OM** can be used to determine the direction of E_0.

Let θ denote the power angle (the angle that *U* lags E_0), then the angle that *I* lags E_0 is

$$\Psi = \varphi + \theta$$

The d- and the q-axis currents are obtained as follows:

$$I_d = I \sin\psi$$

$$I_q = I \cos\psi$$

In the phasor diagrams, the phasor length ON represents the d-axis back emf due to the d-axis resultant flux linkage and is used to determine the d-axis field saturation. From the no-load characteristic curve of the magnetic circuit, E_0, X_{ad} and the excitation current If can be determined based on the frozen method.

Rotor Equipped with a Permanent Magnet Only

If the rotor is equipped with a permanent magnet only, the field excitation can not be adjusted, and the d- and the q-axis currents are obtained based on the following process.

For a given power angle θ (the angle that *U* lags E_0), we have

$$I_d X_d + I_q R_1 = -(U\cos\theta - E_0)$$

$$-I_d R_1 + I_q X_q = U\sin\theta$$

Solving for I_d and I_q yields

$$I_d = -\frac{X_q(U\cos\theta - E_0) - R_1 U\sin\theta}{R_1^2 + X_d X_q}$$

$$I_q = \frac{R_1(U\cos\theta - E_0) - X_d U\sin\theta}{R_1^2 + X_d X_q}$$

Let the angle that *I* lags E_0 be ω, we have

$$\Psi = \tan h\ \frac{I_d}{I_q}$$

The power factor angle f (the angle that *I* lags *U*) is

$$\varphi = \Psi - \theta$$

Power and Efficiency

The output electric power is

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$P_2 = 3UIcos\varphi$$

The input mechanical power is

$$P_1 = P_2 + P_{fw} + P_{Cua} + P_{Fe} + P_{Cuf}$$

where P_{fw}, P_{Cua}, P_{Fe}, , and P_{Cuf} are the frictional and wind, the armature copper, the iron core, the excitation winding copper (if an excitation winding is equipped) losses, respectively.

The input mechanical torque is

$$T_1 = \frac{P_1}{\omega}$$

where ω denotes the synchronous speed in rad/s.

The efficiency of the generator is

$$\eta = \frac{P_2}{P_1} \times 100\%$$

Defining a Claw-Pole Alternator

The general procedure for defining a claw-pole alternator is as follows:

1. Create the alternator project.
2. Double-click the **Machine** entry in the project tree to define the general data.
3. Double-click the **Machine** > **Stator** entry in the project tree to define the stator geometry.
4. Double-click the **Machine** > **Stator** > **Slot** entry in the project tree to define the stator slot dimensions.
5. Double-click the **Machine** > **Stator** > **Winding** entry in the project tree to define the stator windings and conductors.
6. Double-click the **Machine** > **Rotor** entry in the project tree to define the rotor geometry.
7. Double-click the **Machine** > **Rotor** > **Pole** entry in the project tree to define the pole, embrace, offset, and air gap data for the rotor pole.
8. Double-click the **Machine** > **Shaft** entry in the project tree to define the magnetism of the shaft.
9. Optionally, right-click the **Machine** entry in the project tree and click **Insert Housing** to add a machine housing.
10. Right-click **Analysis** in the project tree, and click **Add Solution Setup** to define the solution data.
11. Choose **File** > **Save** to save the project.
12. Choose **RMxprt** > **Analyze** to analyze the design.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** When you place the cursor over an entry field in the data windows, a brief description of that field appears in the status bar at the bottom of the **RMxprt** window.

Once analyzed, the model can be viewed in the Maxwell 2D Modeler, or it can be used to create a new Maxwell 2D project, and new Maxwell 3D design.

Refer to the *Claw-Pole Alternator Problem* application note, on the technical support page of the Ansys web site, for a specific example of a permanent-magnet DC motor problem.

Defining the General Data for a Claw-Pole Alternator

Use the **General** window to define the basic parameters of the alternator, such as the power settings, speed, and rated voltage.

To define the general data:

1. To open the **General Data Properties** window, double-click the **Machine** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the number of poles for the machine in the **Number of Poles** field. This value is the total number of poles in the stator (or the number of pole pairs multiplied by two).
3. Enter or select the **Number of Phases** (2, 3, or 4).
4. Enter the energy loss due to friction at the given speed in the **Frictional Loss** field.
5. Enter the wind loss due to air resistance measured at the reference speed in the **Wind Loss** field.
6. Enter the given speed in the **Reference Speed** field.
7. Click **OK** to close the **Properties** window.

General Data for Claw-Pole Alternators

To access the general data, double-click the **Machine** entry in the project tree.

The **General Data Properties** window for a three-phase induction motor contains the following fields:

Machine Type	Machine type you selected when inserting a new RMxprt design (Claw-Pole Synchronous Machine)
Frictional Loss	Frictional energy loss (due to friction and air resistance) measured at the reference speed
Number of Poles	Number of poles the machine contains
Number of Phases	Number of phases

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Wind Loss	Wind loss measured at the reference speed
Reference Speed	Given speed of reference

Defining the Stator Data for a Claw-Pole Alternator

Use the **Stator Properties** windows to define the stator dimensions, slots, windings, and conductors.

The stator is the outer lamination stack where the polyphase voltage windings reside.

To define the general stator data:

1. To open the **Stator Data Properties** window, double-click the **Machine** > **Stator** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the **Outer Diameter** of the stator.
3. Enter the **Inner Diameter** of the stator.
4. Enter the length of the stator core in the **Length** field.
5. Enter the stacking factor for the stator core in the **Stacking Factor** field. This value is a ratio of he effective magnetic length of the core, and ranges from 0 to 1. The stacking factor is defined as the total length minus the total insulation from the laminations, divided by the total length. A value of 1 indicates that the rotor is not laminated.
6. Select a **Steel Type** for the stator core:
    a. Click the button for **Steel Type**.
    b. In the **Select Definition** window that appears, select a steel type from the list, or define a new steel type.
    c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.
7. Enter the **Number of Slots** in the stator.
8. Select the **Slot Type**:
    a. Click the button for the **Slot Type**.
    b. In the **Select Slot Type** window that appears, select a slot type (available types include 1 through 4).

    > **Note:** When you place the mouse cursor over the slot type, a schematic of the selected type appears, displaying the slot dimension variables.

    Optionally, check **User Defined Slot** if you wish to define the slot dimensions using the **Slot Editor**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Click **OK** to close the **Select Slot Type** window and return to the **Properties** window.

9. Enter the skew width, measured in slot number, in the **Skew Width** field.
10. Click **OK** to close the **Properties** window.

Stator Data for Claw-Pole Alternators

To access the general stator data, double-click the **Machine** > **Stator** entry in the project tree.

The **Stator Data Properties** window contains the following fields:

Outer Diameter	Outer diameter of the stator core
Inner Diameter	Inner diameter of the stator core
Length	Length of the stator core
Stacking Factor	Stacking factor of the stator core
Steel Type	Steel type of the stator core; click the button to open the **Select Definition** window.
Number of Slots	Number of slots the stator core contains
Slot Type	Type of slots in the stator core; click the button to open the **Select Slot Type** window.
Skew Width	Skew width measured in slot number

Defining the Stator Slot Data for a Claw-Pole Alternator

To define the stator slots:

1. To open the **Stator Slot Data Properties** window, double-click the **Machine** > **Stator** > **Slot** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Optionally, to automatically design the dimensions of slots **Hs2**, **Bs1**, and **Bs2**, select the **Auto Design** check box.
3. Optionally, to design dimensions of slots **Bs1** and **Bs2** based on the stator tooth width, select the **Parallel Tooth** check box, and enter a value in the **Tooth Width** field.
4. Enter the available slot dimensions:

   Hs0 Always available

   Hs2 Available only when **Auto Design** is cleared. When **Auto Design** is selected, this slot dimension is determined automatically.

   Bs0 Always available

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Bs1 Available only when **Auto Design** and **Parallel Tooth** are both cleared. When **Auto Design** is selected, this slot dimension is determined automatically. When **Parallel Tooth** is selected, this slot dimension is determined based on the value entered in the **Tooth Width** field.

Bs2 Available only when **Auto Design** and **Parallel Tooth** are both cleared. When **Auto Design** is selected, this slot dimension is determined automatically. When **Parallel Tooth** is selected, this slot dimension is determined based on the value entered in the **Tooth Width** field.

Rs Rs is added when the slot type is 3 or 4.

5. Click **OK** to close the **Properties** window.

Stator Slot Data for Claw-Pole Alternators

To access the stator slot data, double-click the **Machine** > **Stator** > **Winding** entry in the Project Manager tree.

The **Stator Slot Data Properties** window contains the following fields:

Auto Design Select or clear this to enable or disable auto-design of slots **Hs2**, **Bs1**, and **Bs2**. When this check box is selected, only two other fields appear in the window: **Hs0** and **Bs0**.

Parallel Tooth Select this to design **Bs1** and **Bs2** based on the tooth width. When this check box is selected, the **Bs1** and **Bs2** fields are removed, and the **Tooth Width** field is added.

Tooth Width The tooth width for the parallel tooth, on which **Bs1** and **Bs2** are designed.

Hs0 A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Winding** is selected)

Hs1 A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Winding** is selected)

Hs2 A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Winding** is selected)

Bs0 A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Winding** is selected)

Bs1 A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Winding** is selected)

Bs2 A slot dimension (see the diagram shown in the modeling window when **Machine** > **Stator** > **Winding** is selected)

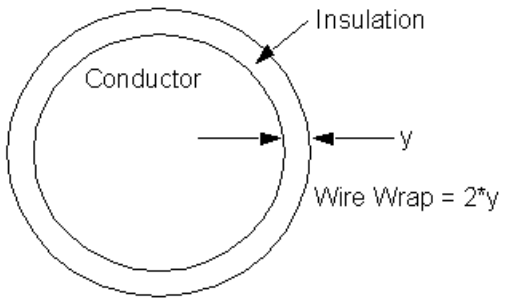
Rs A slot dimension. (see the diagram shown in the modeling window when **Machine** > **Stator** > **Winding** is selected). Rs is added when the slot type is 3 or 4.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining the Stator Winding Data for a Claw-Pole Alternator

To define the stator windings and conductors:

1. To open the **Stator Slot Winding Properties** window, double-click the **Machine** > **Stator** > **Winding** entry in the Project Manager tree. (You can also enter values in the **Properties** pane in the Desktop without opening a separate window.)
2. Click the **Winding** tab.
3. Enter the number of layers in the stator winding in the **Winding Layers** field.
4. Select a **Winding Type**:
    a. Click the button for **Winding Type**.
    b. In the **Winding Type** window that appears, select from one of the following three types of winding:
        - **Whole Coiled**
        - **Half Coiled**
        - **Editor**
5. Select or enter the number of parallel branches in one phase of the winding in the **Parallel Branches** field.
6. Enter the total number of conductors in each stator slot in the **Conductors per Slot** field. This value is the number of turns per coil multiplied by the number of layers.
7. Enter the coil pitch, measured in number of slots, in the **Coil Pitch** field. The coil pitch is the number of slots separating one winding. For example, if a coil starts in slot 1 and ends in slot 6, it has a coil pitch of 5.
8. Enter the number of wires per conductor in the **Number of Strands** field. Enter 0 to have RMxprt auto-design this value.
9. Enter the thickness of the double-sided wire wrap in the **Wire Wrap** field. Enter 0 to automatically obtain this value from the wire library.



10. Select the **Wire Size**:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

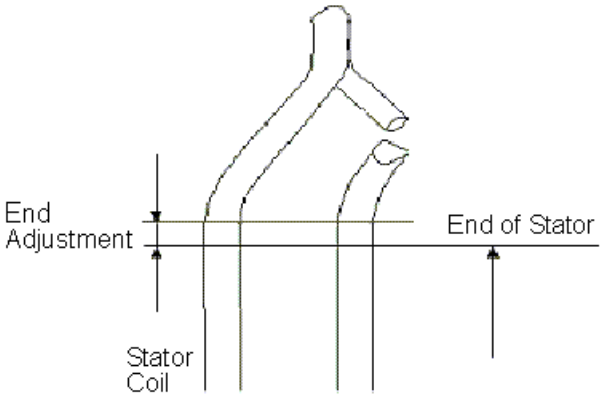
a. Click the button for **Wire Size**.
b. In the **Wire Size** window that appears, select a value from the **Wire Diameter** drop-down list.
c. Select a wire gauge from the **Gauge** drop-down list. You can select from the following options:

< ***number*** **>**	You can select a specific gauge number. When you select a gauge number, the **Wire Diameter** field is automatically updated.
USER	This option allows you to manually enter the **Wire Diameter**. This is useful when you want to enter a diameter that does not correspond to a particular wire gauge.
AUTO	This option sets the **Wire Diameter** to zero, and RMxprt automatically calculates the optimal value. The diameter information is then written to the output file when you analyze the design.
MIXED	This option allows you to define a conductor that is made of different size wires. For example, a single conductor may consist of five wires: three wires with a diameter of 0.21 mm and two wires with a diameter of 0.13 mm.

The gauge number is based on AWG settings. You can create your own wire table using **Machine > Wire**, and then you can select this wire table using **Tools** > **Options** > **General Options** > **Machines**.

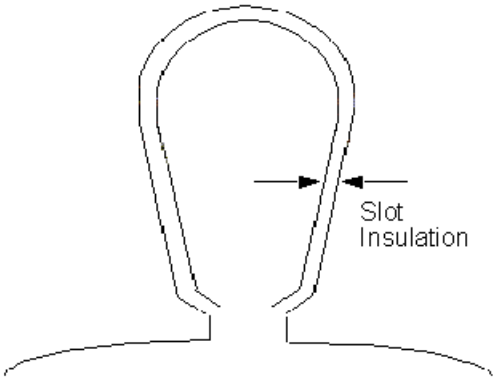
When you are done setting the wire size, click **OK** to close the **Wire Size** window and return to the **Properties** window. Click the **End/Insulation** tab. Select or clear the **Input Half-turn Length** check box. Do one of the following:

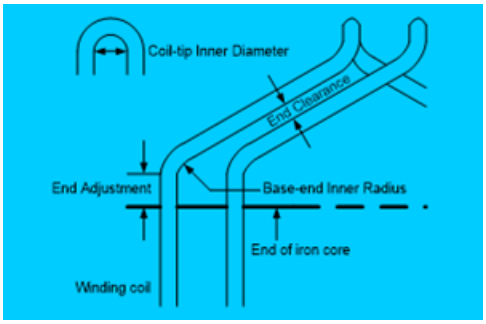
- If you selected **Input Half-turn Length**, then enter the half-turn length of the armature winding in the Half Turn Length field.
- If you cleared **Input Half-turn Length**, then enter the end length adjustment of the stator coils in the **End Adjustment** field. The end adjustment is the distance one end of the conductor extends vertically beyond the end of the stator.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11. Enter the inner radius of the base corner in the **Base Inner Radius** field.
12. Enter the inner diameter of the coil tip in the **Tip Inner Diameter** field.
13. Enter the distance between two stator coils in the **End Clearance** field.
14. Enter the thickness of the slot liner insulation in the **Slot Liner** field.





15. Enter the thickness of the wedge insulation in the **Wedge Thickness** field.
16. Enter the thickness of the insulation layer in the **Layer Insulation** field.
17. Enter the limited slot fill factor for the wire design in the **Limited Fill Factor** field. The slot fill factor is the ratio between the cross-sectional area of all conductors in one slot and the entire slot area.
18. Click **OK** to close the **Properties** window.

Winding Editor for a Claw-Pole Alternator

For a claw-pole alternator, you may want to specify a different number of conductors for each stator slot. The **Winding Editor** makes this possible by enabling you to specify the number of turns for each coil.

To specify the number of turns for each coil:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Machine** > **Winding** > **Edit Layout.**
2. In the **Winding Editor** window that appears, from the table in the upper left, set which phase you want for each coil and which slot is the “in” and “out” slot for the current in each coil.
3. If you are working on a quarter or half model, you may want to specify a multiplier by clicking the **Periodic Multiplier** check box and specifying a value.
4. Select or deselect the **Constant Turns** or **Constant Pitch** check boxes, depending on whether you want to be able to change these setting in the table above. When these options are selected, you cannot change the turns or pitch.
5. When you are satisfied with the coil settings, click **OK** to close the **Winding Editor** window.

Defining Different Size Wires for a Claw-Pole Alternator

Use the **Gauge** option in the Wire Size window if you have a conductor that is made up different size wires.

To define different size wires:

1. In the **Wire Size** window, select **MIXED** from the **Gauge** drop-down list.
2. Select either **Round** or **Rectangular** as the **Wire Type**.
3. Enter the appropriate wire data in the table:
    - For a round wire:
        - Enter the **Diameter** in the table.
        - Enter a **Number** in the table to specify how many of the conductor’s wires have this diameter.
    - For a rectangular wire:
        - Enter the **Width** of the wire in the table.
        - Enter the **Thickness** of the wire in the table.
        - Enter the **Fillet** value in the table.
        - Enter a **Number** in the table to specify how many of the conductor’s wires have this data.
4. Click **Add** to add the new wire data.
5. Repeat steps 3 and 4 for each size wire you want to add.
6. When you are finished defining the wires, click **OK** to close the **Wire Size** window and return to the RMxprt **Properties** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: For example, if one conductor is made up of five wires, and three of those wires have a diameter of 0.21 mm, and the other 2 have a diameter of 0.13 mm, then the mixed wire size table will have two lines. The first line will list Diameter = 0.21 and Number = 3. The second line will list Diameter = 0.13 and Number = 2. An equivalent wire diameter is displayed as Wire Size value in the Winding tab in the Properties window.

Stator Winding Data for Claw-Pole Alternators

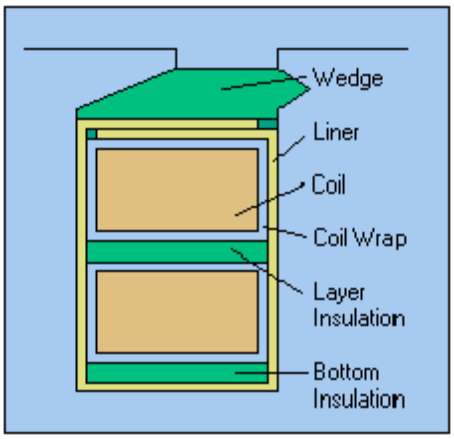
To access the stator winding data, double-click the **Machine** > **Stator** > **Winding** entry in the Project Manager tree.

The **Stator Winding Data Properties** window contains the following fields:

Winding tab	**Winding Layers**	The number of winding layers.
	Winding Type	The type of stator winding. Click the button to open the **Winding Type** window and choose from **Whole Coiled**, **Half Coiled**, and **Editor**.
	Parallel Branches	The number of parallel branches in the stator winding.
	Conductors per Slot	The number of conductors per stator slot (0 for auto-design).
	Coil Pitch	The coil pitch measured in number of slots.
	Number of Strands	The number of wires per conductor (0 for auto-design).
	Wire Wrap	The thickness of the double-sided wire wrap (0 to automatically obtain this value from the wire library).
	Wire Size	The diameter of the wire (0 for auto-design). Click the button to open the **Wire Size** window where you can specify units, wire type, diameter, and gauge.
End/Insulation tab	**Input Half-turn Length**	Select or clear this check box to specify whether or not you want to enter the half-turn length. When this check box is selected, the **Half Turn Length** field appears the next time you open the **Properties** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	When this check box is selected, the **End Adjustment** field appears instead.
Half Turn Length	The half-turn length of the armature winding.
End Adjustment	The end length adjustment of the stator coils, which is the distance one end of the conductor extends vertically beyond the end of the stator.
Base Inner Radius	The inner radius of the base corner.
Tip Inner Diameter	The inner diameter of the coil tip.
End Clearance	The end clearance between two adjacent coils.
Slot Liner	The thickness of the slot liner insulation.
Wedge Thickness	The thickness of the wedge insulation.
Layer Insulation	The thickness of the insulation layer.
Limited Fill Factor	The limited slot fill factor for the wire design. The slot fill factor is the ratio between the cross-sectional area of all conductors in one slot and the entire slot area.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining the Rotor Data for a Claw-Pole Alternator

The rotor is equipped with slots containing copper conductors that are connected to the commutator. The commutator acts as a mechanical rectifier in the motor.

The rotor consists of copper bars in which current is induced by the magnetic fields produced by the stator windings. In the project tree, double-click **Machine** > **Rotor** and **Machine** > **Rotor** > **Pole** to define the rotor and the pole.

To define general rotor data:

1. To open the **Rotor Data Properties** window, double-click the **Machine** > **Rotor** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the outer diameter of the rotor in the **Outer Diameter** field.
3. Enter the inner diameter of the rotor in the **Inner Diameter** field.
4. Enter the length of the rotor core in the **Length** field.
5. Select a **Steel Type** for the rotor core:
    a. Click the button for **Steel Type**.
    b. In the **Select Definition** window that appears, select a steel type from the list, or define a new steel type.
    c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.
6. Enter the diameter of the rotor yoke in the **Yoke Diameter** field.
7. Click **OK** to close the **Properties** window.

Rotor Data for Claw-Pole Alternators

To access the general rotor data, double-click the **Machine** > **Rotor** entry in the project tree.

The **Rotor Data Properties** window contains the following fields:

Outer Diameter	Outer diameter of the rotor core
Inner Diameter	Inner diameter of the rotor core
Length	Length of the rotor core
Steel Type	Steel type of the rotor core; click the button to open the **Select Definition** window.
Yoke Diameter	Diameter of the rotor yoke

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining the Rotor Pole for a Claw-Pole Alternator

The rotor pole drives the electromagnetic field which is coupled with the stator windings. Use the **Rotor Pole Data Properties** window to define the rotor pole.

> **Note:** Some of the fields in the **Rotor Pole** window change, or are inactive, depending on the **Rotor Type** you select.

To define the rotor pole:

1. To open the **Rotor Pole Data Properties** window, double-click the **Machine** > **Rotor** > **Pole** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the pole embrace at the pole tip in the **Tip Embrace** field. This value must be between 0 and 1, exclusive.
3. Enter the pole embrace at the pole root in the **Root Embrace** field. This value must be between 0 and 2, exclusive.
4. Enter the pole thickness at the pole tip in the **Tip Thickness** field.
5. Enter the pole thickness at the pole root in the **Root Thickness** field.
6. Enter the **Pole Length**.
7. Enter the **Slot Depth**.
8. Enter the **Shoe Thickness**.
9. Select the type of magnet to use in the rotor pole from the **Magnet Type** drop-down list.
10. If a magnet is being used, enter its length in the **Magnet Length** field.
11. Enter the width of the second air gap in the **Second Air Gap** field.
12. Click **OK** to close the **Properties** window.

Rotor Pole Data for Claw-Pole Alternators

To access the pole rotor data, double-click the **Machine** > **Rotor** > **Pole** entry in the project tree.

The **Rotor Pole Data Properties** window contains the following fields:

Tip Embrace	Pole embrace at the pole tip; must be > 0 and < 1
Root Embrace	Pole embrace at the pole root; must be > 0 and < 2
Tip Thickness	Pole thickness at the pole tip
Root Thickness	Pole thickness at the pole root

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pole Length	Length of the pole
Slot Depth	Slot depth
Shoe Thickness	Shoe thickness
Magnet Type	Type of magnet; click the button to open the **Select Definition** window. For all pole types.
Magnet Length	Length of the magnet (if a magnet is used)
Second Air Gap	Width of the second air gap

Defining the Shaft Data for a Claw-Pole Alternator

To define the shaft:

1. To open the **Shaft Data Properties** window, double-click the **Machine** > **Shaft** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Select or clear the **Magnetic Shaft** check box to specify whether or not the shaft is to be made of magnetic material.
3. Click **OK** to close the **Properties** window.

Shaft Data for Claw-Pole Alternators

To access the shaft data, double-click the **Machine** > **Shaft** entry in the project tree.

The **Shaft Data Properties** window contains the following fields:

Magnetic Shaft	Select or clear this check box to indicate whether or not the shaft is made of magnetic material.

Setting Up Analysis Parameters for a Claw-Pole Alternator

To define the solution data:

1. To open the Solution Setup window, right-click **Analysis** in the project tree, and click **Add Solution Setup**.
2. Click the **General** tab. The **Operation Type** is automatically set to **General** for this machine type.
3. Select the **Load Type** used in the machine from the following options:

Infinite Bus	When the generator is connected to an infinite bus of the power system, the terminal voltages remain constant and the output power is determined by the output currents.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Independent Generator	The generator supplies electrical power to a load inductor. For an AC generator, the resistance and inductance of the load inductor are determined by the rated output power, the rated voltage, and the rated power factor. For a DC generator, the resistance of the load resistor is determined by the rated output power and the rated voltage.

4. Enter the output power developed at the shaft of the generator in the **Rated Output Power** field.
5. Enter the RMS line-to-line voltage in the **Rated Voltage** field.
6. Enter the desired output speed of the alternator at the load point in the **Rated Speed** field.
7. Enter the temperature at which the system functions in the **Operating Temperature** field.
8. Click the **Claw-Pole Synchronous Machine** tab.
9. Enter a value in the **Rated Power Factor** field.
10. To enter an **Input Exciting Current**, select the check box, enter a value, and select the units.
11. Click **OK** to close the **Solution Setup** window.

Solution Data for Claw-Pole Alternators

To access the solution data, right-click **Analysis** in the project tree, and click **Add Solution Setup**. For this machine type, there is only one tab, the **General** tab.

The **Solution Setup** window contains the following fields:

General tab

Operation Type	Select **Motor** or **Generator** from the drop-down menu. **Generator** is automatically selected for this machine type
Load Type	Select from **Infinite Bus** and **Independent Generator**.
Rated Output Power	Type a value for the rated output voltage, and select the units.
Rated Voltage	Type a value for the rated voltage, and select the units.
Rated Speed	Type a value for the rated speed, and select the units.
Operating Temperature	Type a value for the operating temperature, and select the units.

Claw-Pole Synchronous Machine tab.

Rated Power Factor	Type a value in the field.
Input Exciting Current	Select this check box, enter a value, and select the units. If this check box is cleared, the value will be calculated automatically rather than entered.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Three-Phase Non-Salient Synchronous Machines (NSSM)

After you have selected Three-Phase Non-Salient Synchronous Machine as your model type, enter the data to define the following:

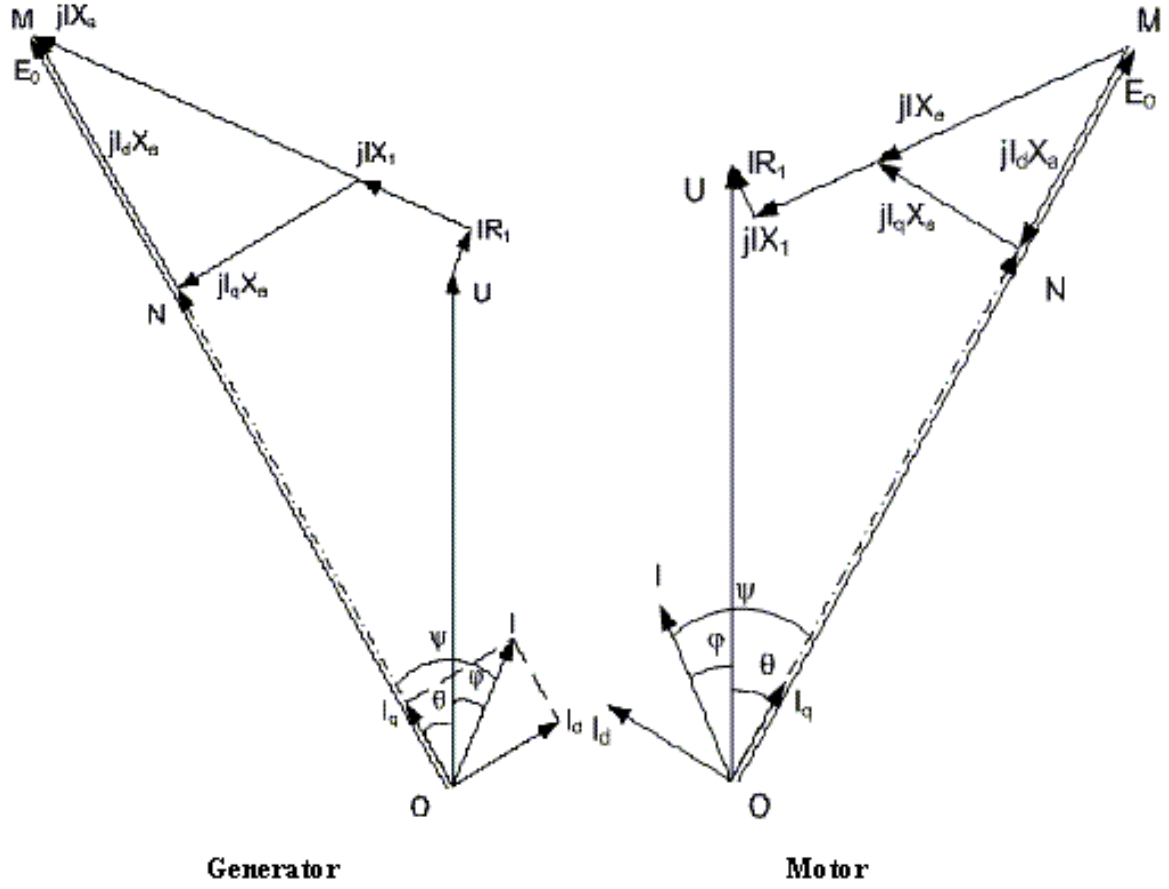
- General data, such as the number of poles, frictional loss, and reference speed
- Stator pole and winding data, such as its associated pole dimensions, type of steel, and wire definitions
- Rotor data, such as the slot types and dimensions, rotor diameter, laminations, and windings and conductors
- Commutator and brush data, such as the commutator dimensions and brush length
- Shaft data
- Optional machine housing
- Solution data

Analysis Approach for Three-Phase Non-Salient Synchronous Machines

The three-phase non-salient-pole synchronous electric machine has two types: the generator and the motor. Their basic structures are the same. The three-phase non-salient-pole synchronous generators are the main source of the electrical energy for industrial, commercial, and private use. They receive the mechanical energy at the shaft and transform it into the electrical energy. The rotor is equipped with a non-salient-pole winding excited by a DC source. The stator is equipped with a three-phase winding that has a sinusoidal spatial distribution. The spinning rotor produces a rotating magnetic field in the air gap of the machine. The frequency of the voltage induced in the stator is given by $f=(pn)/60$, where p is the number of pairs of poles, and n is the mechanical speed of the rotor in rpm, which is called the synchronous speed. The machine is capable of producing both the active and the reactive power as required by the load connected at the stator terminal.

Usually the frequency-domain phasor diagram is adopted to analyze the characteristics. The phasor diagrams for a generator and a motor are shown.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the figure, R_1, X_1, and X_a are the armature resistance, the armature leakage reactance, and the armature reactance, respectively. In a non-salient-pole synchronous machine, $X_{ad} \cong X_{aq}$ and they are both expressed by X_a. Taking the input voltage **U** as the reference phasor, for a given current:

$$I = I\angle -\varphi$$

where φ is the angle **I** lags **U** , which is called the power factor angle.

The internal back EMF induced by the resultant air gap field considering the effects of armature reaction $\mathbf{E_i}$ can be derived from

$$E_i = \begin{cases} U + (R_1 + jX_1) \cdot I & \text{for generator} \\ U - (R_1 + jX_1) \cdot I & \text{for motor} \end{cases}$$

Based on $\mathbf{E_i}$, the resultant air gap flux considering the effects of armature reaction can be computed, and therefore, the magnetic circuit can be solved. With solved magnetic saturation factor, saturated X_a is derived, and therefore, the no-load induced voltage $\mathbf{E_0}$ with the same magnetic saturation (frozen magnetic circuit) can be calculated from

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$E_0 = \begin{cases} E_i + (jX_a) \cdot I & \text{for generator} \\ E_i - (jX_a) \cdot I & \text{for motor} \end{cases}$$

Let the angle **U** lags $\mathbf{E_0}$ be θ, which is called the power angle for the generator or the torque angle for the motor, then the angle **I** lags $\mathbf{E_0}$ is

$$\psi = \varphi + \theta$$

The d- and the q-axis currents can be obtained respectively as follows:

$$I = \begin{bmatrix} I_d \\ I_q \end{bmatrix} = I \begin{bmatrix} \sin\psi \\ \cos\psi \end{bmatrix}$$

Based on the magnetic circuit solution and $\mathbf{E_0}$, X_a and the excitation current I_f can be determined based on the frozen method.

For the generator:

The output power (electric power) is directly computed from the voltage and the current as:

$$P_2 = 3UIcos\varphi$$

The input power (mechanical power) is defined as

$$P_1 = P_2 + P_{fw} + P_{Cua} + P_{Fe} + P_{add} + P_{cuf} + P_{ex}$$

where P_{fw}, P_{Cua}, P_{Fe}, P_{add}, P_{cuf} and P_{ex} are the frictional and wind loss, the armature copper loss, the iron-core loss, the additional loss, the field winding copper loss, and the exciter loss, respectively.

The input mechanical shaft torque is

$$T_1 = \frac{P_1}{\omega}$$

where ω denotes the synchronous speed in rad/s.

For the motor:

The input power (electric power) is directly computed from the voltage and the current as

$$P_1 = 3UIcos\ \varphi$$

The output power (mechanical power) is defined as

$$P_2 = P_1 - (P_{fw} + P_{Cua} + P_{Fe} + P_{add} + P_{cuf} + P_{ex})$$

where P_{fw}, P_{Cua}, P_{Fe}, P_{add}, P_{cuf} and P_{ex} are the frictional and wind loss, the armature copper loss, the iron-core loss, the additional loss, the field winding copper loss, and the exciter loss, respectively.

The output mechanical shaft torque is

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$T_2 = \frac{P_2}{\omega}$$

The efficiency is computed for both the generator and the motor by

$$\eta = \frac{P_2}{P_1} \times 100\%$$

Defining Three-Phase Non-Salient Synchronous Machines

The general procedure for defining a three-phase non-salient synchronous machine is as follows:

1. Create the non-salient synchronous machine project.
2. After you have selected **Three-Phase Non-Salient Synchronous Machine** as your model type, you must define the following:

- General data, such as number of poles, losses, and reference speed
- Stator data, such as dimensions, slot type, skew, and laminations
- Define the Stator slot dimensions
- Winding data, such as the parallel branches, conductors, and wire dimensions and insulation
- Rotor data, such as the rotor dimensions, lamination and slot type
- Define the Rotor slot data
- Define the Shaft Data
- Solution data, such as specifying motor or generator application, and rated output voltage and frequency.

You may also use the following options:

- Add a machine housing
- Add a damper to or remove an existing damper from the rotor
- Add vents to and remove existing vents from the stator

Defining the General Data for a Three-Phase NSSM

To access the general data, double-click the **Machine** entry in the project tree.

The **Properties** window for a three-phase non-salient synchronous machine contains the following fields to be entered:

Machine Type	The machine type you selected when inserting a new RMxprt design (Three Phase Non-Salient Synchronous Machine)
Number of Poles	The number of poles the machine contains. This value is the total number of poles in the stator (or the number of pole pairs multiplied by two).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Frictional Loss	The frictional energy loss (due to friction) measured at the reference speed
Windage Loss	The windage loss (due to air resistance) measured at the reference speed
Reference Speed	The given speed of reference

Defining the Stator for Three-Phase NSSM

The stator is the outer lamination stack where the three-phase windings reside.

Double-click the icon **Machine** > **Stator** in the project tree to display the **Properties** window.

The **Properties** window contains the following fields:

Outer Diameter	Outer diameter of the stator
Inner Diameter	Inner diameter of the stator
Length	Length of the stator core
Stacking Factor	Stacking factor of the stator core
Steel Type	Steel type of the stator core; click the button to open the **Select Definition** window.
Number of Slots	Number of slots the stator core contains
Slot Type	Type of slots in the stator core; click the button to open the **Select Slot Type** window.
Lamination Sectors	Number of lamination sectors
Pressboard Thickness	Magnetic press board thickness (enter 0 for a non-magnetic press board)
Skew Width	Skew width measured in slot number

To define general stator data:

1. To open the **Stator Data Properties** window, double-click the **Machine** > **Stator** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Enter the **Outer Diameter** of the stator.
3. Enter the **Inner Diameter** of the stator.
4. Enter the length of the stator core in the **Length** field.
5. Enter the stacking factor for the stator core in the **Stacking Factor** field.
6. Select a **Steel Type** for the stator core:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Click the button for **Steel Type**.
b. In the **Select Definition** window that appears, select a steel type from the list, or define a new steel type.
c. Click **OK** to close the **Select Definition** window and return to the **Properties** window.

7. Enter the **Number of Slots** in the stator.
8. Select the **Slot Type**:
   a. Click the button for the **Slot Type**.
   b. In the **Select Slot Type** window that appears, select a slot type (available types include 1 through 6). Slot types **1** though **4** are filled with round wire. Slot types **5** and **6** are filled with rectangular wire. If **Auto Design** is enabled, the software designs an optimum slot geometry; in this case, you can input the tooth width dimension, and the software determines the slot width accordingly.

      > **Note:** When you place the mouse cursor over the slot type, a schematic of the selected type appears, displaying the slot dimension variables.

      Optionally, check **User Defined Slot** if you wish to define the slot dimensions using the **Slot Editor**.

   c. Click **OK** to close the **Select Slot Type** window and return to the **Properties** window.
9. Enter the number of sectors in the **Lamination Sectors** field.
10. Enter the thickness of the magnetic press board in the **Pressboard Thickness** field. Enter **0** for a non-magnetic press board.
11. Enter the skew width, measured in slot number, in the **Skew Width** field.
12. Click **OK** to close the **Properties** window.

Defining Stator Slots for a Three-Phase NSSM

To define the slot dimensions:

1. To open the **Stator Slot Data Properties** window, double-click the **Machine** > **Stator** > **Slot** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Optionally, to automatically design the dimensions of slots **Hs2**, **Bs1**, and **Bs2**, select the **Auto Design** check box.
3. Enter the available slot dimensions:

   Hs0 Always available

   Hs2 Available only when **Auto Design** is cleared. When **Auto Design** is selected, this slot dimension is determined automatically.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Bs0 Always available

Bs1 Available only when **Auto Design** is cleared. When **Auto Design** is selected, this slot dimension is determined automatically.

Bs2 Available only when **Auto Design** is cleared. When **Auto Design** is selected, this slot dimension is determined automatically.

4. Click **OK** to close the **Properties** window.

Defining Stator Windings and Insulation for a Three-Phase NSSM

Double-click the icon **Machine** > **Stator** > **Winding** in the project tree to display the **Properties** dialog box, which has two tab sheets: **Winding** and **End/Insulation**.

Define Wires, Conductors and Windings of NSSM Stator

In the **Winding** tab, define the wire, conductor, and winding of the stator.

Winding Layers	The number of layers in the stator winding. Select the winding layers from the drop-down menu (available choices 1 and 2).
Winding Type	The type of the stator winding. Set the winding type to **Editor** to use the **Winding Editor** dialog to design the coil windings
Parallel Branches	The number of parallel branches in one phase of the stator winding
Conductors per Slot	The total number of conductors in each stator slot. This value is the number of turns per coil multiplied by the number of layers. Enter 0 to have RMxprt auto-design this value.
Coil Pitch	The coil pitch measured in number of slots. The coil pitch is the number of slots separating one winding. For example, if a coil starts in slot 1 and ends in slot 6, it has a coil pitch of 5.
Number of Strands	The number of wires per conductor. Enter 0 to have RMxprt auto-design this value.
Wire Wrap	The thickness of the double-sided wire wrap. Enter 0 to automatically obtain this value from the wire library.
Wire Size	Wire size (0 for auto-design). You can assign wire size of round wires or rectangle wires. When the slot type you selected is 1 to 4, round wires are used. When the slot type you selected is 5 or 6, rectangle wires are used.

Define End Windings and Insulation of NSSM Stator

In the tab sheet **End/Insulation**, define the end winding and the insulation of the stator.

Input Half-turn Length	Select or clear this check box to specify whether or not you want to enter the half-turn length. When this check box is selected, the row **Half Turn Length**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	appears the next time you open the **Properties** dialog box. When this check box is cleared, the row **End Adjustment** appears instead.
Half-turn Length	The half-turn length of the armature winding. It is available when **Input Half-turn Length** is selected.
End Adjustment	The end length adjustment of the stator coils, which is the distance of one end of the conductor extending vertically beyond the end of the stator. It is available when **Input Half-turn Length** is cleared.
Base Inner Radius	The inner radius of the base corner
Tip Inner Diameter	The inner diameter of the coil tip.
End Clearance	The end clearance between two adjacent stator coils
Coil Wrap	Single-side coil wrap insulation thickness
Slot Liner	The thickness of the slot liner insulation
Wedge Thickness	The thickness of the wedge insulation
Limited Fill Factor	The limited slot fill factor for the wire design.
Layer Insulation	The thickness of the insulation layer
Bottom Insulation	Bottom insulation thickness

Winding Editor

For a non-salient synchronous motor, you may want to specify a different number of conductors for each stator slot. The **Winding Editor** makes this possible by enabling you to specify the number of turns for each coil. To enable the **Winding Editor**, you must have set the **Winding Property** for the **Winding Type** to **Editor**.

Stator Vent Data for Three-Phase NSSM

To insert a vent on a stator for a three-phase synchronous machine:

1. Right-click on the stator icon in the project tree to display the shortcut menu.
2. Click **Insert Vent**.

The vent icon appears in the project tree under the stator.

To remove an existing vent item:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Right-click on the stator icon in the project tree to display the shortcut menu.
2. Click **Remove Vent**.

This removes the vent item from the project tree.

To access the Vent properties for a vent, double-click on a vent item. The **Vent Properties** window contains the following fields.

Vent Ducts	The number of radial vent ducts
Duct Width	The width of the radial vent ducts
Magnetic spacer width	Width of magnetic spacer which holds vent ducts. 0 for non-magnetic spacer.
Duct pitch	Center-to-Center distance between two adjacent Vent ducts

Define NSSM Rotor Data

Double-click the icon **Machine** > **Rotor** in the Project Manager tree to display the **Properties** window, which has one tab sheet, **Rotor**, that is used to define the rotor general data.

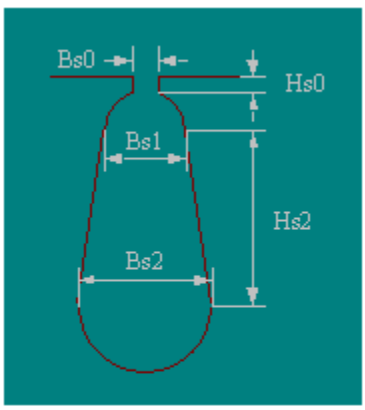
Outer Diameter	Outer diameter of the rotor core
Inner Diameter	Inner diameter of the rotor core
Length	Length of the rotor core
Stacking Factor	Stacking factor of the rotor core
Steel Type	Select a steel type for the rotor core material
Press Board Thickness	Magnetic press board thickness, 0 for non-magnetic press board
Indexing Slots	Number of indexing slots of the rotor core used to determine slot pitch
Real Slots	Number of Slots of the rotor core
Slot Type	Slot type of the rotor core. There are six types of rotor slots.

Define NSSM Rotor Slot

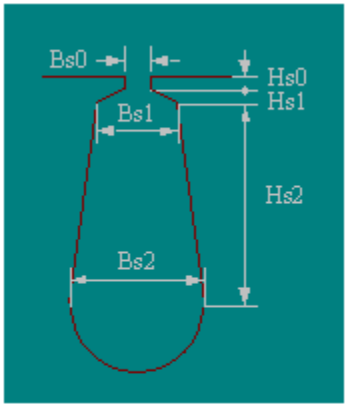
Double-click the icon **Machine** > **Rotor** > **Slot** in the project tree to display the **Properties** dialog box.

In the **Slot** tab, define the available rotor slot dimensions as illustrated. There are in total six types of slots that are available:

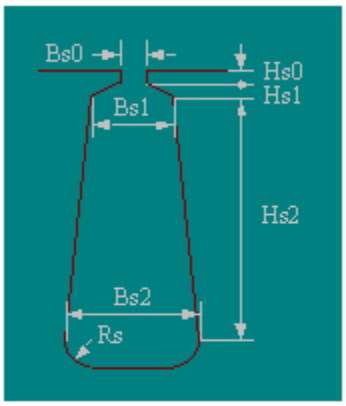
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



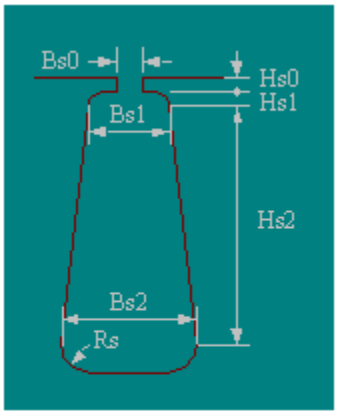
Type 1 Slot



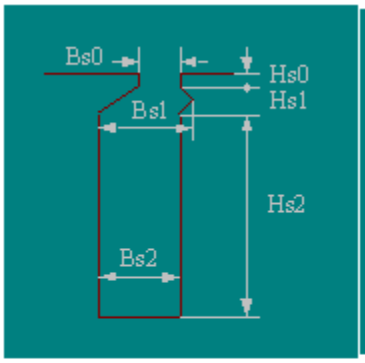
Type 2 Slot



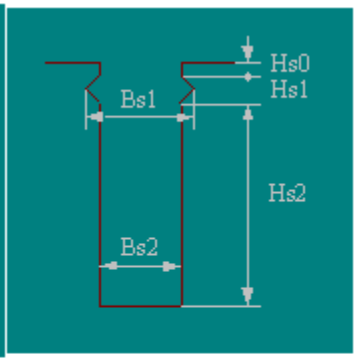
Type 3 Slot



Type 4 Slot



Type 5 Slot



Type 6 Slot

Define NSSM Rotor Winding

The rotor winding is equipped on the rotor pole to provide the excitation for the magnetic field.

Double-click the icon **Machine** > **Rotor** > **Winding** in the project tree to display the **Properties** dialog box, where you define the wires and physical dimensions of the rotor winding.

In the **Winding** tab, the following are defined:

Parallel Branches	The number of parallel branches in the rotor winding
Conductors per Slot	The number of conductors per slot (0 for auto-design)
Number of	The number of wires per conductor (0 for auto-design)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Strands	
Wire Wrap	The thickness of the double-sided wire wrap (0 to automatically obtain this value from the wire library)
Wire Size	Wire size (0 for auto-design). You can assign wire size of round wires or rectangle wires. When you select Round Wire for Winding Type, round wires are used (refer to section 8.4.1 Assign Round Wire Sizes). Otherwise, rectangle wires are used (refer to section 8.4.2 Assign Rectangular Wire Size).

In the **End/Insulation** tab the following are defined:

Input Half-turn Length	Select or clear this check box to specify whether or not you want to enter the half-turn length. When this check box is selected, the row **Half Turn Length** appears the next time you open the **Properties** dialog box. When this check box is cleared, the row **End Adjustment** appears instead.
Half-turn Length	The half-turn length of the armature winding. It is available when **Input Half-turn Length** is selected.
End Adjustment	One-side end extended length
Inner Fillet Radius	Inner fillet radius at the span corner
End Clearance	End clearance between two adjacent coils
Coil Wrap	Insulation: single-side coil wrap thickness
Slot Liner	Insulation: slot liner thickness
Wedge Thickness	Insulation: wedge thickness
Bottom Insulation	Insulation: bottom insulation thickness
Limited Cross Height	The limited cross-section height for the winding design or arrangement, or Overall Height as shown in Figure 12.12 (0 for available maximum area)
Winding Fillet	The size of the winding fillet

Rotor Vent Data for NSSMs

By option, you can add vents to a rotor in a three-phase NSSM.

To add a vents to the rotor:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the rotor icon in the project tree.
2. Right-click to display the pop-up menu and select **Insert Vent**.

The vent icon appears in the project tree under the rotor.

To remove a vent to stator in a three-phase induction motor:

1. Select the rotor icon in the project tree.
2. Right-click to display the pop-up menu and select **Remove Vent**.

The vent icon disappears in the project tree under the stator.

The Vent data for the NSSM rotor includes the following fields.

Surface Ducts	Number of surface tangential vent ducts
Surface Duct Width	Width of surface tangential vent ducts
Surface Duct Depth	Depth of surface tangential vent ducts
Surface Duct Pitch	Pitch of surface tangential vent ducts
Axial Ducts	Number of axial vent ducts per pole
Axial Duct Width	Width of axial vent ducts in main teeth
Axial Duct Depth	Depth of axial vent ducts in main teeth

Define NSSM Shaft Data

To define the shaft:

1. Click the icon **Machine** > **Shaft** in the project tree to display the **Properties** dialog box.
2. In the **Shaft** tab, select or clear the check box **Magnetic Shaft** to specify whether or not the shaft is to be made of the magnetic material.
3. Click **OK** to close the **Properties** dialog box.

Analysis Setup for Three-Phase Non-Salient Synchronous Machines

To set up the solution data:

1. Right-click the **Analysis** icon in the Project Manager tree, then click **Add Solution Setup** from the shortcut menu to display the Properties window. There are two tabs: the **General** tab and the **NSSM** tab.
2. On the **General** tab, define the solution setup data.

Operation Two options from the drop-down menu: Generator and Motor

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type	
Load Type	Select a load type for the motor or generator from the drop-down menu (refer to section 7.8 Assign Load Types)
Rated Apparent Power	The output electric apparent power in kVA developed at the terminal for the generator, or **Rated Output Power**: The output mechanical power in kW developed at the shaft for the motor.
Rated Voltage	The RMS line-to-line voltage
Rated Speed	The desired synchronous speed
Operating Temperature	The temperature at which the system functions, and select the units. The Operating Temperature will affect all winding resistances and therefore affect all ohmic losses.

3. On the **NSSM** tab, define the connection data:

Rated Power Factor	The rated power factor. For generators, the rated output power is determined by the rated apparent power multiplying the rated power factor.
Winding Connection	Select Wye or Delta from the drop-down menu.
Exciter Efficiency	The percentage efficiency of the exciter used to supply the rotor winding with the DC current if it is mechanically connected to the shaft of the generator. The efficiency value ranges between 0% and 100% and will only affect the total efficiency result.
Input Exciting Current	If the check box is selected, the accompanying edit box is enabled. You need to input the exciting current value and select the units if needed.
Exciting Current	Exciting current for rated operation

4. Click **OK** to close the pop-up dialog box.

Validate NSSM Solution Setup

1. Click **RMxprt** > **Validation Check** to display the information box Validation Check.
2. If any items do not pass validation, use the diagnostic information in the window to resolve any issues.
3. Click **Close** to close the information box Validation Check.
4. When the design has been validated, click **RMxprt** > **Analyze All**.
5. The analysis progress is shown in the Progress window and the analysis message is shown in the Message Manager.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Design Output for Non-Salient Synchronous Machines

When **RMxprt** has completed a solution, you can display and analyze the results in the following ways:

- View Performance
- View Design Sheet
- View Curves
- Create Reports

View Performance

To view the solutions:

Click **RMxprt** > **Results** > **Solution Data** to display the information box Solutions. It has three tabs.

In the **Performance** tab, from the **Data** drop-down list, you have thirteen different data tables for the line start permanent magnet motor, which can be used to define Output Variables for design optimization:

- FEA input Data
- Field Winding
- Full-load Magnetic Variables
- Important Factors
- Material Consumption
- No-load Magnetic Variables
- Rated Operation
- Stator Slot
- Stator Winding
- Steady State Parameters per Unit
- Transient Data
- Transient Data per Unit
- Unsaturated Steady State Parameters

View Design Sheet

In the tab sheet Design Sheet, you have twelve sets of information, as follows:

- General Data
- Stator Data
- Stator Winding Data

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

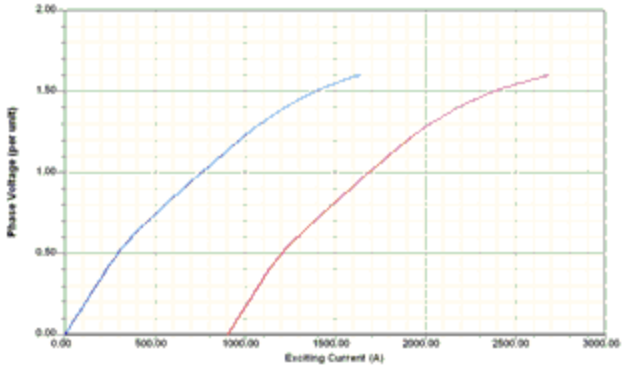
- Rotor Data
- Field Winding Data
- Some Factors and Material Consumption
- Unsaturated Steady State Parameters
- No Load Magnetic Data
- Full Load Magnetic Data
- Full Load Electric Data
- Transient Parameters and Time Constants
- Transient FEA Input Data

> **Note:** To print the **Design Sheet**: Right-click the **Design Sheet**, select **Print** from the shortcut menu, select the printer and other parameters from the dialog box **Print**, and click **OK** to print.

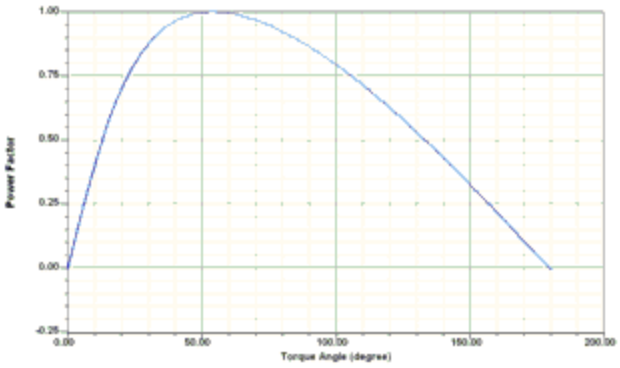
View Curves

In the tab sheet **Curves**, from the **Name** drop-down menu, there are ten curves, as shown:

Phase Voltage vs Exciting Current

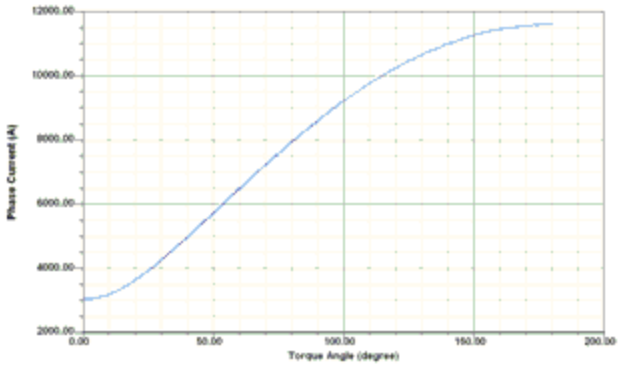


Power Factor vs Torque Angle

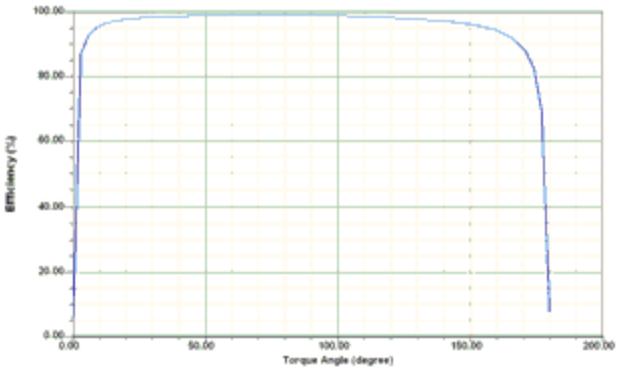


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

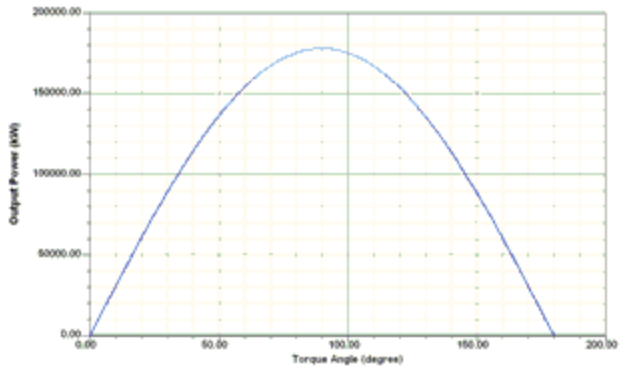
Armature Phase Current vs Torque Angle



Efficiency vs Torque Angle

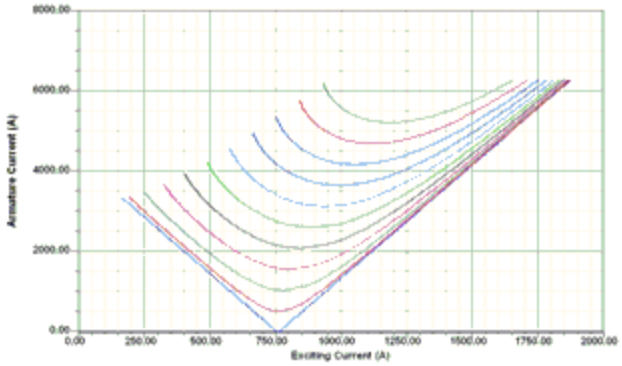


Output Power vs Torque Angle

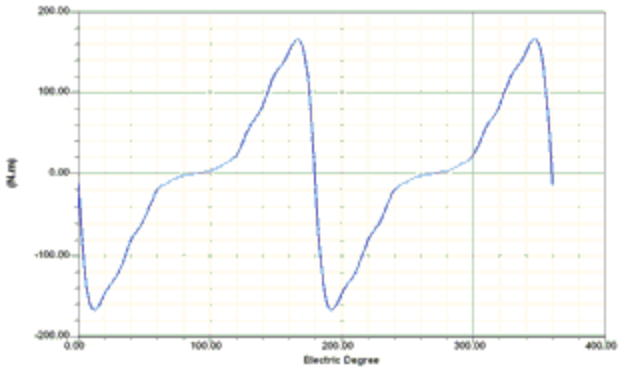


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

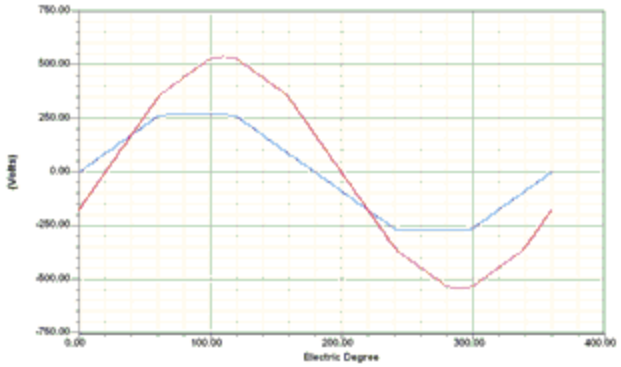
Armature Current vs Exciting Current



Cogging Torque in Two Teeth

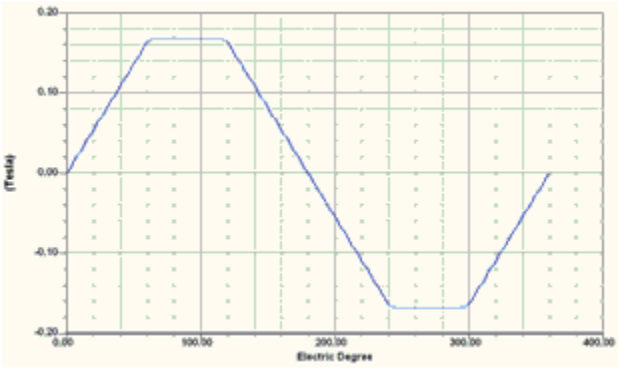


Induced Coil Voltages at No Load

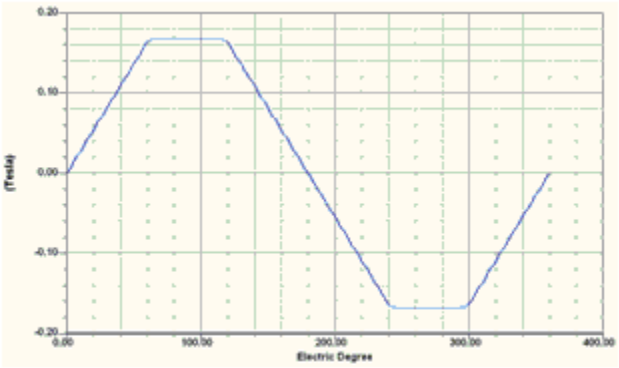


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Air-Gap Flux Density at No-Load



Induced Winding Voltages at No-Load



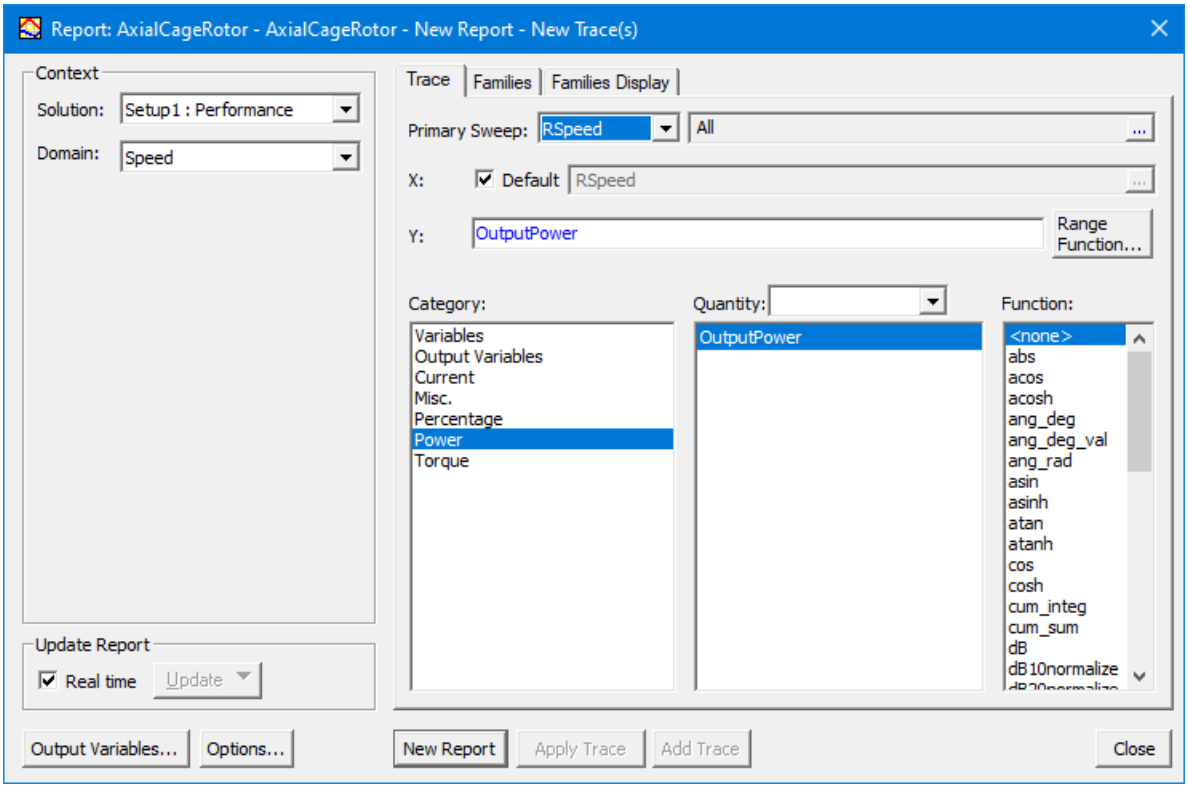
Note: To print the plots from the **Curve**: Right-click on the plot, select **Print** from the shortcut menu, select the printer and other parameters from the dialog box **Print**, and click **OK** to print.

Create Reports

1. Click **RMxprt** > **Results** > **Create RMxprt Report** > **Rectangular Plot**.

   The **Report** window appears as shown:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Under the tab sheet **Trace**, there are **Variables**, **Output Variables**, **Current, Misc**, **Percentage**, and **Power** under the **Category** column. Select one from the **Category** column, select the traces that belong to it from the **Quantity** column, and click the button **Add Trace** to add them one by one. Finally click the button **New report** to create the plot. You can always add additional curves to the same plot by repeating the process.
3. Double-click the icon **Results** > **XY Plot1** to display the graph with multiple traces in a new window.

> **Note:** To print the plots from the **Curves**: Right-click on the plot, select **Print** from the shortcut menu, select the printer and other parameters from the dialog box **Print**, and click **OK** to print.
>
> To get a screen shot of from the **Curves**: Right-click on the plot, select **Copy Image**, then paste to a destination file.

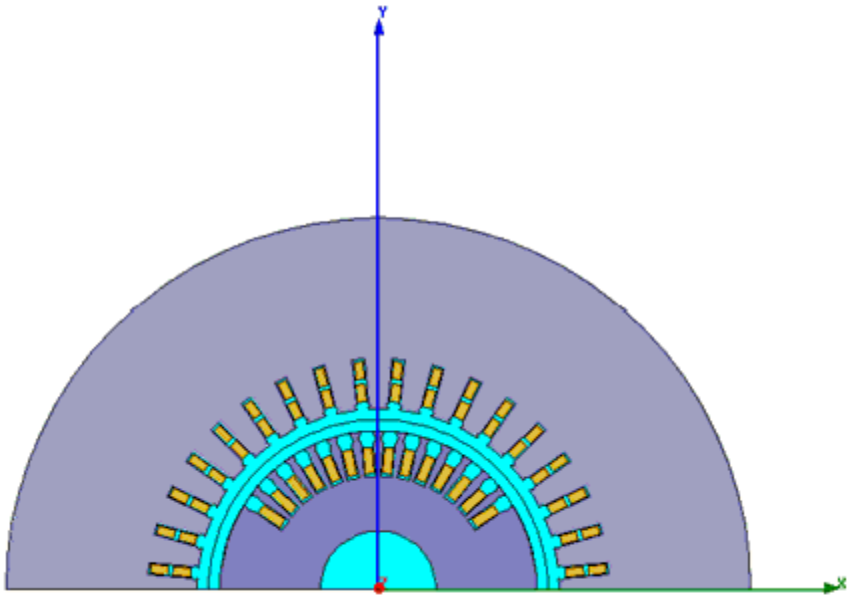
Transient FEA of the Non-Salient Synchronous Machines

If you expect to continue the transient or electromagnetic-field FEA with Maxwell 2D, you can create a Maxwell 2D design directly from RMxprt. For transient FEA, RMxprt can create a Maxwell 2D design with all setups completed.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create Maxwell 2D Design

Click the command **RMxprt** > **Analysis Setup** > **Create Maxwell Design…** in RMxprt to create a Maxwell 2D design with Auto setup checked. A Maxwell 2D design called *Maxwell2DDesign1* is created with the displayed geometry as shown below. All setups are automatically completed by RMxprt.



Review Maxwell 2D Design Setups

This section reviews all setups automatically completed by RMxprt. For detailed setup process, refer to Setting Up a Maxwell Design.

Solution Type Setup

Click **Maxwell 2D** > **Solution Type…** in Maxwell 2D, you can review that the **Solution Type** is set as **Magnetic Transient**.

Model Setup

1. **Model Depth**:
    a. Click **Maxwell 2D** > **Design Setting**.
    b. Click the **Set Model Depth** tab to review the Model Depth: 3590 mm.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. **Motion Type**:
    a. Double-click on **Maxwell2DDesign1** > **Model** > **MotionSetup1** in the Project Manager window.
    b. In the **Type** tab, review that the **Motion Type** is set as **Rotation**, and the **Moving Vector**is set as **Positive Global: Z**.
3. **Initial Position**:
    a. Click the **Data** tab of the **Motion Setup** panel.
    b. Review the **Initial Position**, it is set as **100** deg with **Rotate Limit** unchecked. The rotor initial position is set to such a position that the initial flux linkage of the phase-A winding is at its negative maximum value.
4. **Mechanical Load**:
    a. Click the **Mechanical** tab of the **Motion Setup** panel.
    b. Check that **Angular Velocity** is set as **3000** rpm with **Consider Mechanical Transient** unchecked.
5. **Symmetry Multiplier**:
    a. Right-click on **Maxwell2DDesign1** > **Model** in the Project Manager window.
    b. Select **Set Geometry Multiplier** in the pop-up panel.
    c. Check that the **Symmetry Multiplier** is set as **2**.

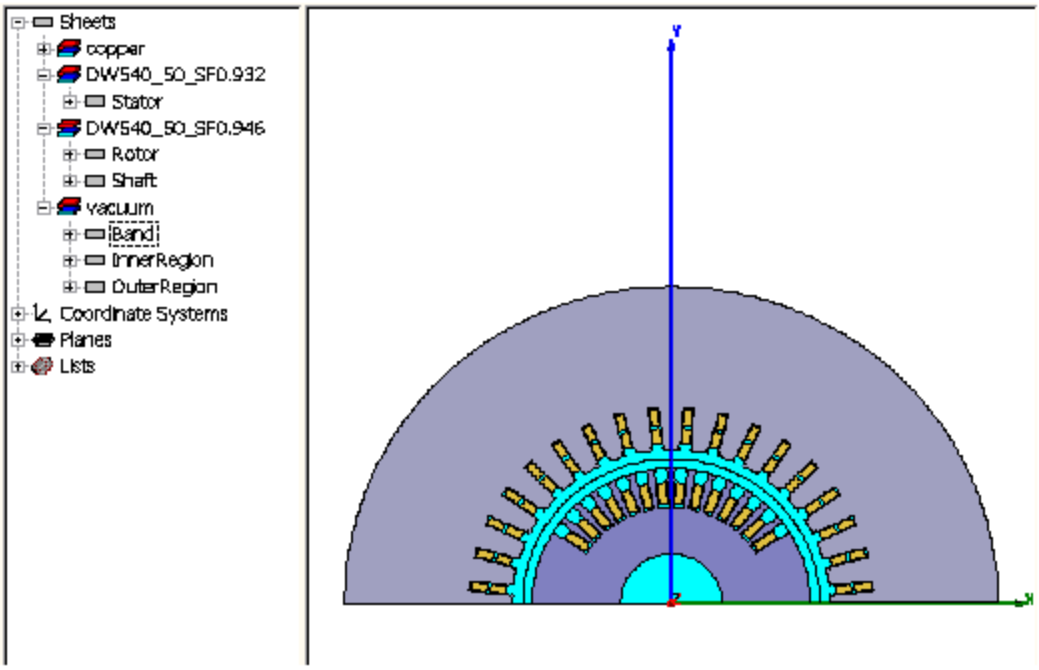
Boundary Setup

1. **Vector Potential Boundary**
    a. Double-click on **Maxwell2DDesign1** > **Boundaries** > **VectorPotential1** in the Project Manager window.
    b. Check that the highlighted outer half circle in the geometry is set as the **Vector Potential Boundary**, and its value is set as **0**.
2. **Independent Boundary**
    a. Click on **Maxwell2DDesign1** > **Boundaries** > **Independent1** in the Project Manager window.
    b. Check that the highlighted arrowhead line from left to right in the geometry is set as the **Independent Boundary**.
3. **Dependent Boundary**
    a. Double-click on **Maxwell2DDesign1** > **Boundaries** > **Dependent1** in the Project Manager window.
    b. Check that the highlighted arrowhead line from right to left in the geometry is set as the **Dependent Boundary**, and the relation of the dependent boundary to the independent boundary is set as **Bs = -Bm**. This is because the geometry includes only one magnetic pole of the machine.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Material Assignment

In the Maxwell 2D modeler windows history tree, you can see that all stator and rotor coil terminals are assigned to material copper by default. **Band**, **InnerRegion** and **OuterRegion** are assigned as vacuum as shown:



Two new materials called DW540_50_SF0.932, and DW540_50_SF0.946 are automatically created for **Stator** and **Rotor**, based on the original material of DW540_50 used in RMxprt and the equivalent stacking factors of 0.932 and 0.946. Shaft is also assigned as DW540_50_ SF0.946, because the shaft is defined as magnetic in RMxprt.

Excitation Setup

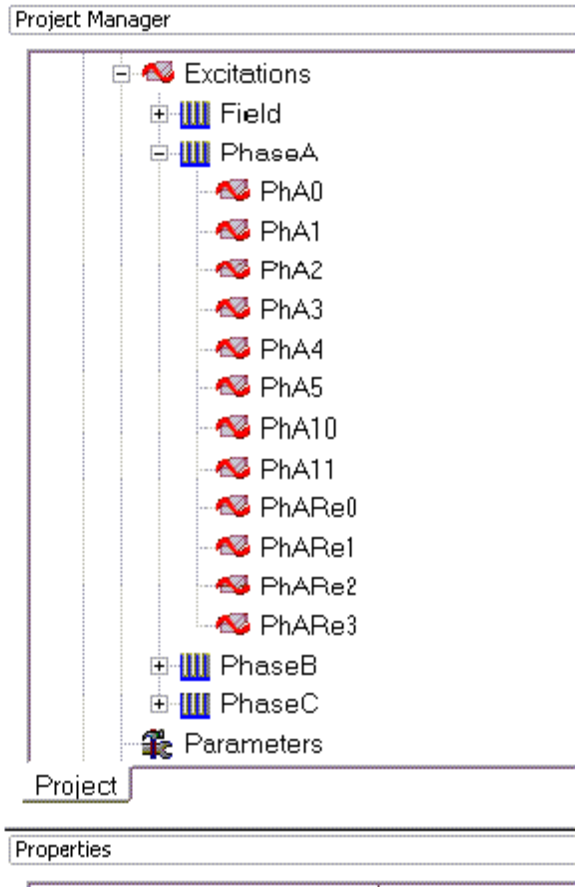
Windings

1. Click on **Maxwell2DDesign1** > **Excitations** > **PhaseA** in the Project Manager window, all objects assigned to this phase are highlighted in the modeler window.
2. In the Properties window, review all winding properties:
    - **Voltage** for **Winding Type**
    - **Stranded** for **IsSolid**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **0.00226117** ohms for **Resistance**
- **8.87325e-005** H for **Inductance**
- **1** for **Number ofParallel Branches**
- 11267.7 * sin(2*pi*50*time-43.4944*pi/180) for **Voltage**, where **50** is the **frequency** in Hz, **11267.7** is the **phase peak voltage** in Volts, **pi** is a predefined constant, and **time** is a predefined variable for time. By using sin function instead of cos function, the applied voltage and back EMF are in phase. Therefore, a phase shift in the applied voltage source will be the power angle of the motor. 43.4944 degrees is the power angle at full load operation. The values for resistance, inductance and number of parallel branches are obtained from the TRANSIENT FEA INPUT DATA section in RMxprt design sheet.

3. Click on **PhaseB**, **PhaseC**, or **Field** to review all objects assigned to this winding in the modeler window, and winding properties in the **Properties** window.



Coil Terminals

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

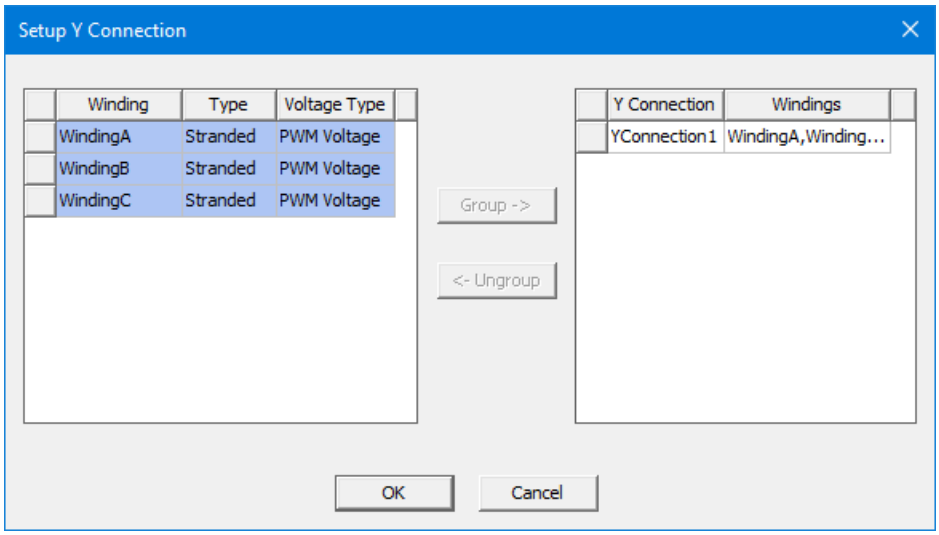
A winding consists of several coil terminals, and two coil terminals represent a coil in a complete 2D model. Because we are working with only one-half of the motor structure, one coil terminal can represent one complete coil with independent/dependent boundary conditions provided.

A coil terminal has properties of **Number of Conductors** and **Polarity Type**. **Number of Conductors** is the number of turns per coil, and it is equal to the **Number of Turns** given in RMxprt divided by number of coils per phase. **Polarity Type** defines the direction of the current in the coil; it can be either positive or negative.

1. Expand a winding and click on a coil terminal to review the object corresponding to this coil terminal in the modeler window and all coil terminal properties in the **Properties** window. In this example, **Number of Conductors** of A, B, and C coil terminals is assigned as 1, and it is 12 for the Field windings.
2. Click on **PhaseB**, **PhaseC**, or **Field** to review all objects assigned to this winding in the modeler window, and winding properties in the properties window.

Y-Connection for Three-Phase Windings:

1. Right-click on **Maxwell2DDesign1** > **Excitations** in the Project Manager Window, and click **Setup Y Connection…** in the pop-up panel.
2. Group three windings into a Y-connection.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- A mix of stranded winding and solid winding in one Y-connection group is not allowed.
- A mix of PWM voltage winding and (normal) voltage winding in one Y-connection group is not allowed.
- Only one Y Connection is allowed in a given Maxwell design.

Mesh Operation Setup

The Maxwell 2D mesh generator can create meshes according to predefined mesh operations. A mesh operation defines one or more conditions for some selected objects for mesh maker to create meshes that satisfy the conditions. RMxprt automatically sets up some mesh operations for different machine parts based on geometry sizes.

For this example, mesh operations include Length_Coil (set maximum mesh length as 18 mm for all coils), Length_Field (set maximum mesh length as 19 mm for field winding coils), Length_Main (set the maximum mesh length as 135 mm for all other parts), SurfApprox_Main (set the limited Surface Deviation as 1.175 mm and the limited Normal Deviation as 30 deg for all parts with true-surface arcs).

Click on one of the mesh operations under **Maxwell2DDesign1** > **Mesh** in the Project Manager window, you can review its properties in the Properties window.

Solution Setup

1. Click on **Maxwell2DDesign1** > **Analysis** > **Setup1** in the Project Manager window.
2. Review its properties in the Properties window:
    - 0.2 s for Stop time, that is 10 periods
    - 0.0002 s for Timestep (100 steps per period)

Analyze Maxwell 2D Design

Before analyzing the Maxwell 2D design, you may want to generate and plot a mesh. You may also want to create several Quick Reports to display results.

To analyze the Maxwell 2D design: right-click on **Maxwell2DDesign1** > **Analysis** > **Setup1** in the project tree, and click **Analyze**. While the design is being analyzed, you can update one or all result reports and view the reports.

To update all reports: right-click on **Maxwell2DDesign1** > **Results** in the project tree, and select **Update All Reports**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To update one report: right-click on the report under **Maxwell2DDesign1** > **Results** in the project tree, and select **Update Report**.

To view all traces of a report: when you double-click on the report under **Maxwell2DDesign1** > **Results** in the project tree, the Modeler window changes to the Results window, and all traces (a curve in a report is a trace) of the selected reports are displayed in the Results window.

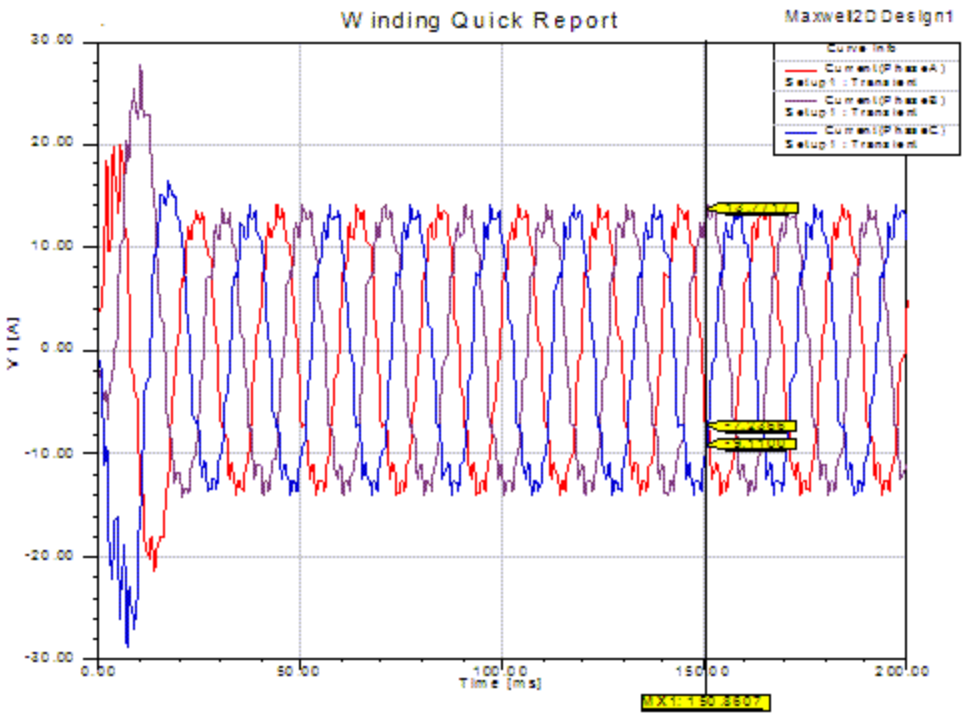
To view a trace of a report: when you click on a trace of a report under **Maxwell2DDesign1** > **Results** in the project tree, the selected trace is highlighted in the Results window.

To cancel the simulation: right-click on the progress bar in the progress window, and pick up **Abort** in the pop-up panel.

To stop the simulation so that you can continue the simulation later: right-click on the progress bar in the progress window, and pick up **Clean Stop** in the pop-up panel.

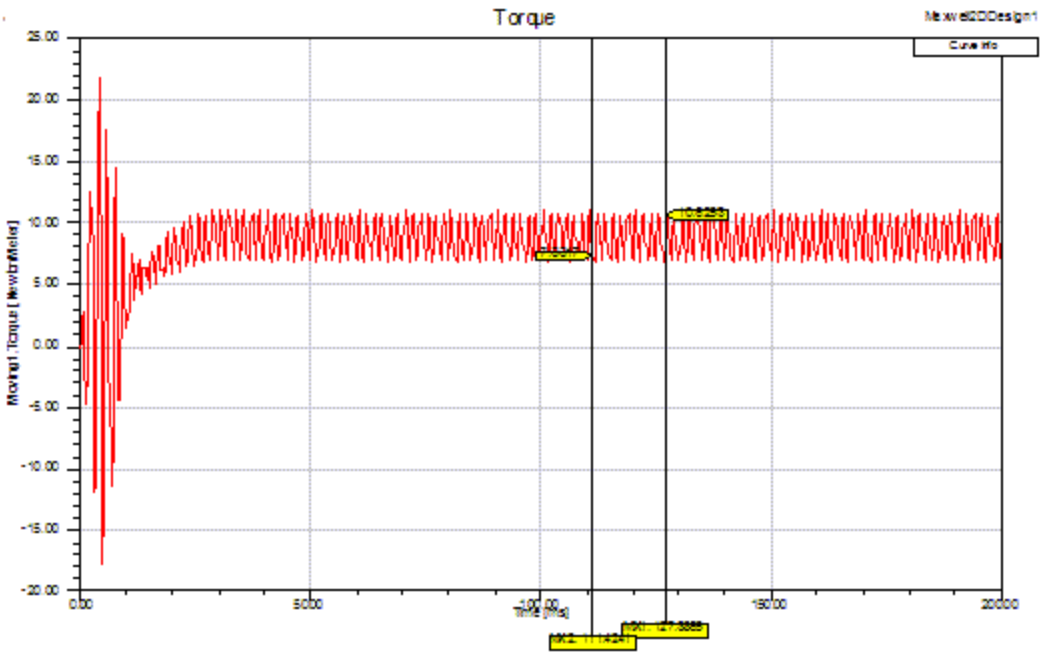
For this example, the simulated three-phase currents and the electromagnetic torque are shown in the figures that follow.

Right-click on the **Winding Quick Report** in the Results window, and select **Marker** > **Add X Marker** in the pop-up panel, yellow-shaded boxes are added in the report to indicate X and all Y values. Click on the X box (or the vertical line), and drag it to some place where you see the steady-state peak value of a phase current as shown:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Add X makers in Torque Quick Report to indicate the steady-state maximum and minimum values of torque as shown below. The average torque can be approximately obtained from the maximum and minimum values as Tav = (Tmax + Tmin) / 2 = (10.63 + 7.33) / 2 = 8.98 Nm.



Generic Rotating Machines

The Generic Rotating Machine (GRM) templates allow you to define any of several machine types using the core types listed below.

Core Type	Description
SLOT_AC	Slotted core with AC winding
SLOT_CAGE	Slotted core with squirrel-cage damper
SALNT_POLE	Salient-pole core with field winding
PM_INTERIOR	Interior permanent magnet core
SOLID	Solid core with/without copper/hysteresis sleeve
AXIAL_AC	Axial-flux slotted core with AC winding
AXIAL_CAGE	Axial-flux slotted core with squirrel-cage damper
AXIAL_PM	Axial-flux permanent magnet core
NONS_RELU	Non-salient reluctance core without winding

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The machine templates that can be selected, with their associated core type combinations, source types, and position control usage, are listed in the following table.

> **Note:** Refer to Defining a Generic Rotating Machine for detailed information on selecting GRM templates and defining the corresponding machine types.

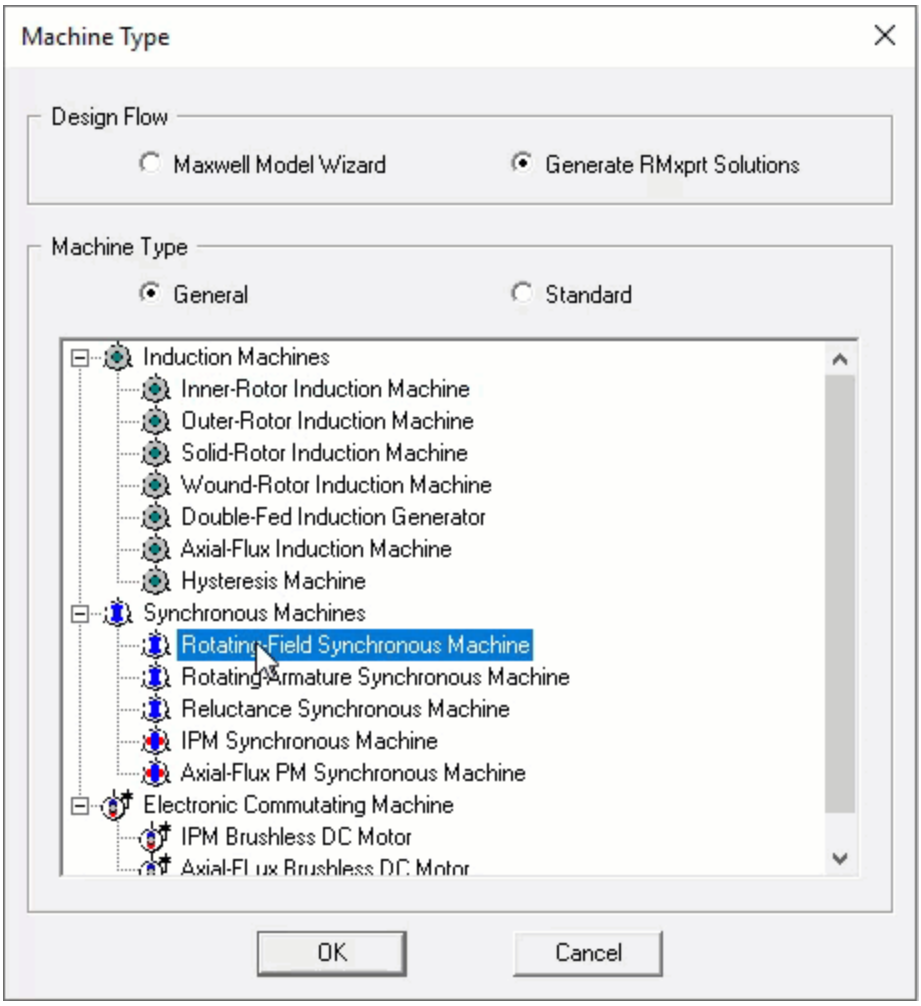
GRM Machine Type	Stator Type / Rotor Type	Source Type	Position Control
Inner-Rotor Induction Machine	SLOT_AC / SLOT_CAGE	AC	No
Outer-Rotor Induction Machine	SLOT_AC / SLOT_CAGE	AC	No
Solid-Rotor Induction Machine	SLOT_AC / SOLID	AC	No
Wound-Rotor Induction Machine	SLOT_AC / SLOT_AC	AC	No
Double-Fed Induction Generator	SLOT_AC / SLOT_AC	AC	No
Axial-Flux Induction Machine	AXIAL_AC / AXIAL_CAGE	AC	No
Hysteresis Machine	SLOT_AC / SOLID	AC	No
Rotating-Field Synchronous Machine	SLOT_AC / SALNT_POLE	AC	No
Rotating Armature Synchronous Machine	SLOT_AC / SALNT_POLE	AC	No
Reluctance Synchronous Machine	SLOT_AC / NONS_RELU	AC	No
IPM Synchronous Machine	SLOT_AC / PM_INTERIOR	AC	No
Axial-Flux PM Synchronous Machine	AXIAL_AC / AXIAL_PM	AC	No
IPM Brushless DC Motor	SLOT_AC / PM_INTERIOR	DC	Yes
Axial-Flux Brushless DC Motor	AXIAL_AC / AXIAL_PM	DC	Yes

Defining a Generic Rotating Machine

The general procedure for defining a generic rotating machine is as follows:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. To insert a Generic Rotating Machine into a new or existing project, select **Generate RMxprt Solutions** for the **Design Flow** and **General** for the **Machine Type**.
2. Select the desired machine from the expandable list:



Machine types on the list include the following:

Induction Machines

- Defining an Inner-Rotor Induction Machine
- Defining an Outer-Rotor Induction Machine
- Defining a Solid-Rotor Induction Machine
- Defining a Wound-Rotor Induction Machine
- Defining a Double-Fed Induction Generator

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Defining an Axial-Flux Induction Machine
- Defining a Hysteresis Machine

Synchronous Machines

- Defining a Rotating-Field Synchronous Machine
- Defining a Rotating-Armature Synchronous Machine
- Defining a Reluctance Synchronous Machine
- Defining an IPM Synchronous Machine
- Defining an Axial-Flux PM Synchronous Machine

Electronic Commutating Machine

- Defining an IPM Brushless DC Motor
- Defining an Axial-Flux Brushless DC Motor

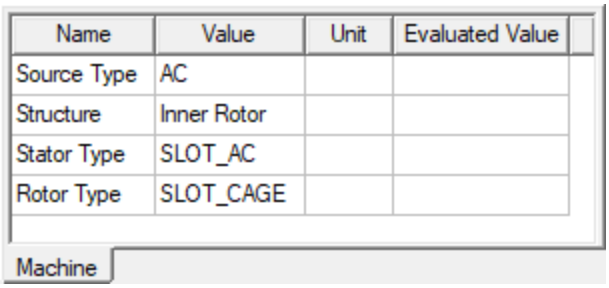
Refer to the *Generic Rotating Machine* application note, on the technical support page of the Ansys web site, for a specific example of a problem using a Generic Rotating Machine.

Defining an Inner-Rotor Induction Machine

> **Note:** Refer to Analysis Approach for Three-Phase Induction Motors for general information on this machine type.

The general procedure for defining an Inner-Rotor Induction Machine is as follows:

1. Insert an **Inner-Rotor Induction Machine** into a new or existing project by choosing **Generate RMxprt Solutions** for the **Design Flow**, **General** for the **Machine Type**, then select **Inner-Rotor Induction Machine** from the expandable list.
2. Click the **Machine** entry in the project tree to set the machine properties as shown:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click to expand the **Machine** > **Stator** entry in the project tree to define data for the stator and its associated core and core slot, winding, and optionally, vent for the SLOT_AC core type.
4. Click to expand the **Machine** > **Rotor** entry in the project tree to define data for the rotor and its associated core, damper and damper slot, and optionally, vent for the SLOT_CAGE core type.
5. Click the **Machine** > **Shaft** entry in the project tree to define the magnetism, frictional and windage losses, and reference speed of the shaft.
6. Optionally, right-click the **Machine** entry in the project tree and click **Insert Housing** to add a machine housing.
7. Right-click **Analysis** in the project tree, and click **Add Solution Setup** to define the solution data.
8. Choose **File** > **Save** to save the project.
9. Choose **RMxprt** > **Analyze** to analyze the design.

Once analyzed, the model can be used to create a new Maxwell 2D or 3D design.

Analysis Approach for Three-Phase Induction Motors

For a three-phase induction motor, the stator winding (with a sinusoidal spatial distribution and p pairs of poles) is connected to a three-phase symmetric voltage power supply. The resulting currents in the stator produce a rotating magnetic field. The rotor winding is often a squirrel-cage type with the number of poles dictated by the number of poles in the stator. Currents are induced in the rotor bars and produce, in turn, a second rotating magnetic field. The two rotating fields produce a resultant rotating magnetic field in the air gap of the machine. The interaction of this field in the air gap with the rotor bar currents produces an electromagnetic torque, which acts on the rotor in the direction of the rotation of the field in the air gap. A torque of equal value acts upon the stator in the opposite direction.

The stator winding, which is connected to a phase of the supply system, has p coils, each with a symmetric spatial distribution and an opening of $\pi D/2p$, where D is the diameter of the winding. In this case, the magnetic field in the air gap has p periods, and the winding has p pairs of poles.

The performance of three-phase induction motors (IndM3) is analyzed based on the equivalent circuit of one phase in the frequency domain as shown in Figure 1.

In the figure, R1 is the stator resistance, X1 is stator leakage reactance, which consists of stator slot leakage reactance, end-winding leakage reactance, and differential leakage reactance. X2 and R2 are rotor leakage reactance and rotor resistance, respectively. X2 includes rotor slot leakage reactance, end-ring leakage reactance, differential leakage reactance, and skewing leakage reactance. Due to the saturation of the leakage field, X1 and X2 are nonlinear. The parameters in the equivalent circuit are dependent on the stator and rotor currents. Due to the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

skin effects, X2 and R2 are the equivalent values from a distributed-parameter circuit, as shown in Figure 2.

They vary with the rotor slip *s*. All rotor parameters have been referred to the stator side.

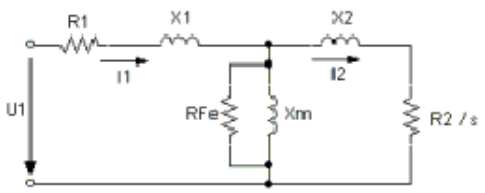


Figure 1

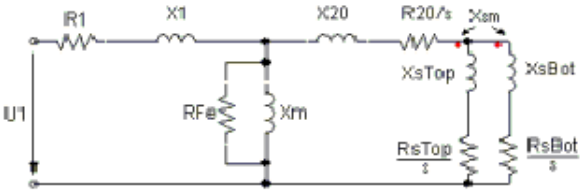


Figure 2

In the exciting branch, Xm is the magnetizing reactance, and RFe is the resistance corresponding to iron-core losses. Xm is a linearized nonlinear parameter that varies with the saturation of the main field.

After a phase voltage U1 is applied to the phase terminals, stator phase current I1 and rotor current I2, which has been referred to the stator, can be easily computed by the circuit analysis. The electromagnetic power Pm, or air-gap power, is computed by the following:

$$Pm = 3 * I2^2 * R2/s$$

The electromagnetic torque Tm is

$$Tm = Pm/\omega$$

where ω is the synchronous speed in rad/s.

The output mechanical shaft torque T2 is

$$T2 = Tm - Tfw$$

where Tfw is the frictional and wind torque.

The output power is

$$P2 = T2 * \omega 2$$

where $\omega 2 = \omega * (1 - s)$ and is rotor speed in rad/s.

The input power is

$$P1 = P2 + Pfw + Pcu2 + PFe + Pcu1 + Ps$$

where Pfw, Pcu2, PFe, Pcu1, and Ps are frictional and wind loss, rotor copper loss, iron-core loss, stator copper loss, and stray loss, respectively.

The power factor is derived from

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$PF = P1/(m * U1 * I1)$$

The efficiency is computed by

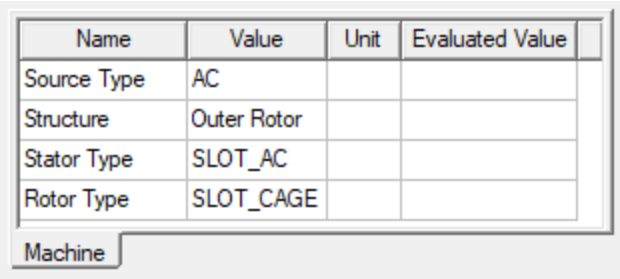
$$\eta = \frac{P_2}{P_1} \times 100\%$$

Defining an Outer-Rotor Induction Machine

> **Note:** Refer to Analysis Approach for Three-Phase Induction Motors for general information on this machine type.

The general procedure for defining an Outer-Rotor Induction Machine is as follows:

1. Insert an **Outer-Rotor Induction Machine** into a new or existing project by choosing **Generate RMxprt Solutions** for the **Design Flow**, **General** for the **Machine Type**, then select **Outer-Rotor Induction Machine** from the expandable list.
2. Click the **Machine** entry in the project tree to set the machine properties as shown:



Name	Value	Unit	Evaluated Value
Source Type	AC		
Structure	Outer Rotor		
Stator Type	SLOT_AC		
Rotor Type	SLOT_CAGE		

Machine

3. Click to expand the **Machine** > **Stator** entry in the Project Manager tree to define data for the stator and its associated core and core slot, winding, and optionally, vent for the SLOT_AC core type.
4. Click to expand the **Machine** > **Rotor** entry in the Project Manager tree to define data for the rotor and its associated core, damper and damper slot, and optionally, vent for the SLOT_CAGE core type.
5. Click the **Machine** > **Shaft** entry in the Project Manager tree to define the magnetism, frictional and windage losses, and reference speed of the shaft.
6. Right-click **Analysis** in the Project Manager tree, and click **Add Solution Setup** to define the solution data.
7. Choose **File** > **Save** to save the project.
8. Choose **RMxprt** > **Analyze** to analyze the design.

Once analyzed, the model can be used to create a new Maxwell 2D or 3D design.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Analysis Approach for Three-Phase Induction Motors

For a three-phase induction motor, the stator winding (with a sinusoidal spatial distribution and *p* pairs of poles) is connected to a three-phase symmetric voltage power supply. The resulting currents in the stator produce a rotating magnetic field. The rotor winding is often a squirrel-cage type with the number of poles dictated by the number of poles in the stator. Currents are induced in the rotor bars and produce, in turn, a second rotating magnetic field. The two rotating fields produce a resultant rotating magnetic field in the air gap of the machine. The interaction of this field in the air gap with the rotor bar currents produces an electromagnetic torque, which acts on the rotor in the direction of the rotation of the field in the air gap. A torque of equal value acts upon the stator in the opposite direction.

The stator winding, which is connected to a phase of the supply system, has *p* coils, each with a symmetric spatial distribution and an opening of $\pi D/2p$, where *D* is the diameter of the winding. In this case, the magnetic field in the air gap has *p* periods, and the winding has *p* pairs of poles.

The performance of three-phase induction motors (IndM3) is analyzed based on the equivalent circuit of one phase in the frequency domain as shown in Figure 1.

In the figure, R1 is the stator resistance, X1 is stator leakage reactance, which consists of stator slot leakage reactance, end-winding leakage reactance, and differential leakage reactance. X2 and R2 are rotor leakage reactance and rotor resistance, respectively. X2 includes rotor slot leakage reactance, end-ring leakage reactance, differential leakage reactance, and skewing leakage reactance. Due to the saturation of the leakage field, X1 and X2 are nonlinear. The parameters in the equivalent circuit are dependent on the stator and rotor currents. Due to the skin effects, X2 and R2 are the equivalent values from a distributed-parameter circuit, as shown in Figure 2.

They vary with the rotor slip *s*. All rotor parameters have been referred to the stator side.

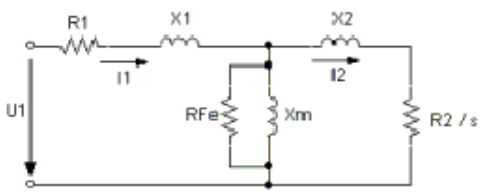


Figure 1

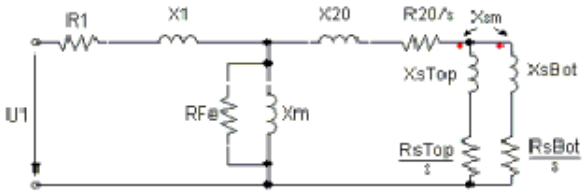


Figure 2

In the exciting branch, Xm is the magnetizing reactance, and RFe is the resistance corresponding to iron-core losses. Xm is a linearized nonlinear parameter that varies with the saturation of the main field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After a phase voltage U1 is applied to the phase terminals, stator phase current I1 and rotor current I2, which has been referred to the stator, can be easily computed by the circuit analysis. The electromagnetic power Pm, or air-gap power, is computed by the following:

$$Pm = 3 * I2^2 * R2/s$$

The electromagnetic torque Tm is

$$Tm = Pm/\omega$$

where ω is the synchronous speed in rad/s.

The output mechanical shaft torque T2 is

$$T2 = Tm - Tfw$$

where Tfw is the frictional and wind torque.

The output power is

$$P2 = T2 * \omega 2$$

where $\omega 2 = \omega * (1 - s)$ and is rotor speed in rad/s.

The input power is

$$P1 = P2 + Pfw + Pcu2 + PFe + Pcu1 + Ps$$

where Pfw, Pcu2, PFe, Pcu1, and Ps are frictional and wind loss, rotor copper loss, iron-core loss, stator copper loss, and stray loss, respectively.

The power factor is derived from

$$PF = P1/(m * U1 * I1)$$

The efficiency is computed by

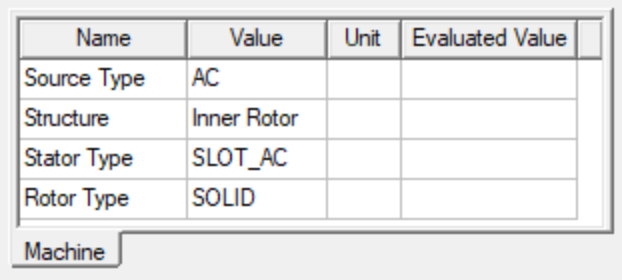
$$\eta = \frac{P_2}{P_1} \times 100\%$$

Defining a Solid-Rotor Induction Machine

The general procedure for defining a Solid-Rotor Induction Machine is as follows:

1. Insert a **Solid-Rotor Induction Machine** into a new or existing project by choosing **Generate RMxprt Solutions** for the **Design Flow**, **General** for the **Machine Type**, then select **Solid** > **Rotor Induction Machine** from the expandable list.
2. Click the **Machine** entry in the project tree to set the machine properties as shown:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Click to expand the **Machine** > **Stator** entry in the project tree to define data for the stator and its associated core and core slot, winding, and optionally, vent for the SLOT_AC core type.
4. Click to expand the **Machine** > **Rotor** entry in the project tree to define data for the rotor core for the SOLID core type.
5. Click the **Machine** > **Shaft** entry in the project tree to define the magnetism, frictional and windage losses, and reference speed of the shaft.
6. Optionally, right-click the **Machine** entry in the project tree and click **Insert Housing** to add a machine housing.
7. Right-click **Analysis** in the project tree, and click **Add Solution Setup** to define the solution data.
8. Choose **File** > **Save** to save the project.
9. Choose **RMxprt** > **Analyze** to analyze the design.

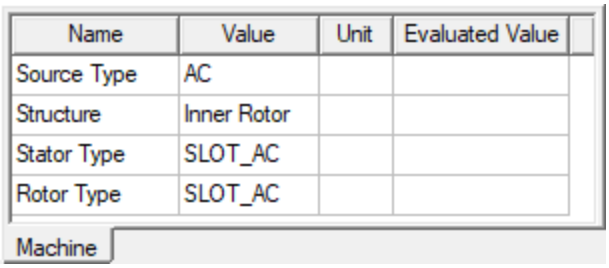
Once analyzed, the model can be used to create a new Maxwell 2D or 3D design.

Defining a Wound-Rotor Induction Machine

The general procedure for defining a Wound-Rotor Induction Machine is as follows:

1. Insert a **Wound-Rotor Induction Machine** into a new or existing project by choosing **Generate RMxprt Solutions** for the **Design Flow**, **General** for the **Machine Type**, then select **Wound-Rotor Induction Machine** from the expandable list.
2. Click the **Machine** entry in the Project Manager tree to set the machine properties as shown:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Click to expand the **Machine** > **Stator** entry in the project tree to define data for the stator and its associated core and core slot, winding, and optionally, vent for the SLOT_AC core type.
4. Click to expand the **Machine** > **Rotor** entry in the project tree to define data for the rotor and its associated core and core slot, winding, and optionally, vent for the SLOT_AC core type.
5. Click the **Machine** > **Shaft** entry in the project tree to define the magnetism, frictional and windage losses, and reference speed of the shaft.
6. Optionally, right-click the **Machine** entry in the project tree and click **Insert Housing** to add a machine housing.
7. Right-click **Analysis** in the project tree, and click **Add Solution Setup** to define the solution data.
8. Choose **File** > **Save** to save the project.
9. Choose **RMxprt** > **Analyze** to analyze the design.

Once analyzed, the model can be used to create a new Maxwell 2D or 3D design.

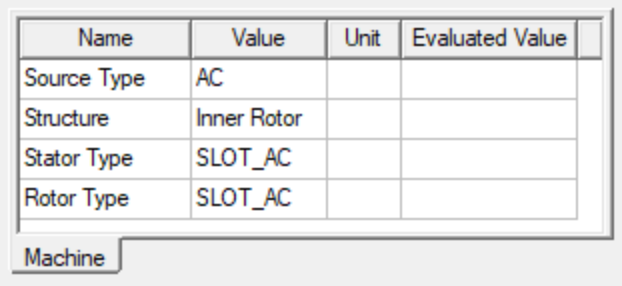
Defining a Double-Fed Induction Generator

> **Note:** Refer to Analysis Approach for a Doubly Fed induction Generator for general information on this machine type.

The general procedure for defining a Double-Fed Induction Generator is as follows:

1. Insert a **Double-Fed Induction Generator** into a new or existing project by choosing **Generate RMxprt Solutions** for the **Design Flow**, **General** for the **Machine Type**, then select **Double-Fed Induction Generator** from the expandable list.
2. Click the **Machine** entry in the Project Manager tree to set the machine properties as shown:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Click to expand the **Machine** > **Stator** entry in the project tree to define data for the stator and its associated core and core slot, winding, and optionally, vent for the SLOT_AC core type.
4. Click to expand the **Machine** > **Rotor** entry in the project tree to define data for the rotor and its associated core and core slot, winding, and optionally, vent for the SLOT_AC core type.
5. Click the **Machine** > **Shaft** entry in the project tree to define the magnetism, frictional and windage losses, and reference speed of the shaft.
6. Optionally, right-click the **Machine** entry in the project tree and click **Insert Housing** to add a machine housing.
7. Right-click **Analysis** in the project tree, and click **Add Solution Setup** to define the solution data, where **Operation Type** is set to **Wind Generator**.
8. Choose **File** > **Save** to save the project.
9. Choose **RMxprt** > **Analyze** to analyze the design.

Once analyzed, the model can be used to create a new Maxwell 2D or 3D design.

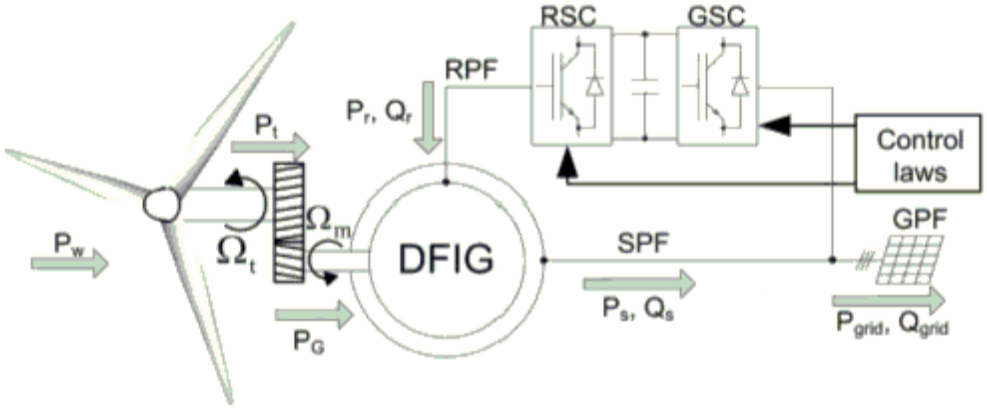
Analysis Approach for a Doubly Fed Induction Generator

A Doubly Fed Induction Generator (DFIG) can be configured to operate as a Generator.

DFIG Operating as a Generator

Doubly-fed induction generators (DFIGs) are widely used in wind power systems. A DFIG works as a component of a wind power system, as shown below, where the wind turbine transforms wind energy into mechanical energy, and the DFIG transforms mechanical energy into electrical energy.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For a DFIG, both the stator and the rotor are equipped with poly-phase AC windings. The stator and rotor windings may, or may not, have the same number of phases, but they must have the same number of poles *p*. In order to produce terminal voltages with desired frequency *f* in the stator winding, the rotor winding must be excited by balanced poly-phase currents with the slip frequency sf via an AC-DC-AC convert. Slip *s* is defined as

$$s = \frac{1-n}{n_0}$$

where *n* is the rotor speed, and n_0 is the synchronous speed as given below:

$$n_0 = \frac{60f}{p}$$

When the rotor speed is *lower* than the synchronous speed, the rotor currents have the same phase sequence as the stator currents, and the rotor winding gets power from the converter. However, when the rotor speed is *higher* than the synchronous speed, the phase sequence of the rotor currents is different from that of the stator currents, and the rotor winding outputs power to the converter.

For a given wind turbine, the power coefficient (the ratio of turbine power to the wind power), is a function of the tip speed ratio (the ratio of the blade tip speed to the wind speed). In order to track the maximum power point, the tip speed ratio must keep constant - at its optimal value. The input mechanical power with Maximum Power Point Tracking (MPPT) must satisfy:

$$P_{mech} = P_{\mathrm{m_ref}}\left(\frac{\omega_m}{\omega_{ref}}\right)^3$$

where P_{m_ref} is the turbine power with MPPT at a reference speed of ω_{ref} based on the optimal tip speed ratio, and ω_m is the rotor speed in rad/s.

The rotor mechanical loss is

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$P_f = P_{f_ref}\left(\frac{\omega_m}{\omega_{ref}}\right)^3$$

where P_{f_ref} is mechanical loss measured at a reference speed of ω_{ref}.

The electromagnetic power in the air gap is

$$P_{em} = (P_{mech} - P_f)/(1 - s)$$

Therefore, the stator output electrical power at rated operation is

$$P_1 = P_{em} - m_1 I_1^2 R_1 = m_1 V_1 I_1 \cos\varphi$$

where m_1 is the number of phases of the stator winding, R_1 is the stator phase resistance, V_1 is the stator rated phase voltage, I_1 is the rated stator phase current to be determined, and cosφ is the rated power factor. Solving for I_1 gives the following equation:

$$I_1 = \frac{2P_{em}/m_1}{V_1\cos\varphi + \sqrt{(V_1\cos\varphi)^2 + 4R_1 P_{em}/m_1}}$$

Then, based on the equivalent circuit shown below, one obtains:

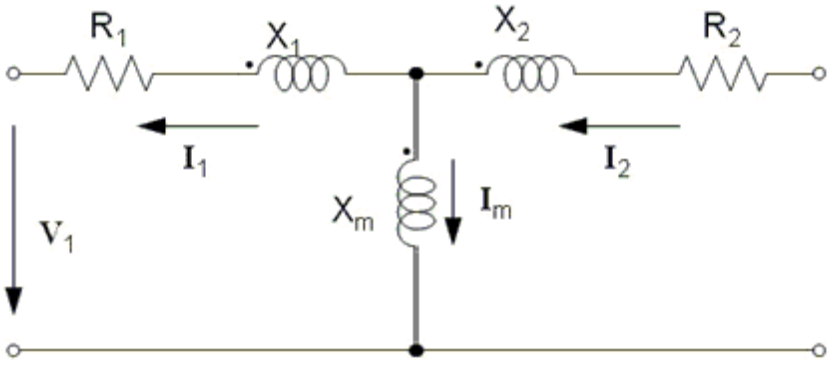
$$V_1 = V_1\angle 0$$

$$I_1 = I_1\angle -\varphi$$

$$E_m = V_1 + I_1(R_1 + jX_1)$$

$$I_m = (E_m/X_m(E_m))$$

$$I_2 = I_2\angle -\varphi_2 = I_1 + I_m$$



Now, rotor input electrical power can be computed as

$$P_2 = sP_{em} + m_2 I_2^2 R_2$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where m_2 is the number of phases of the rotor winding.

The electromagnetic torque T_{em} is

$$T_{em} = \frac{P_{em}}{\omega}$$

where ω denotes the synchronous speed in rad/s.

The input mechanical torque on the shaft is

$$T_{mech} = T_{em} + T_f$$

where T_f denotes the frictional torque.

The total electrical output power is

$$P_{elec} = P_1 - P_2 - p_{Fe}$$

where p_{Fe} is the core loss.

The efficiency is defined as

$$\eta = \frac{P_{elec}}{P_{mech}} \times 100\%$$

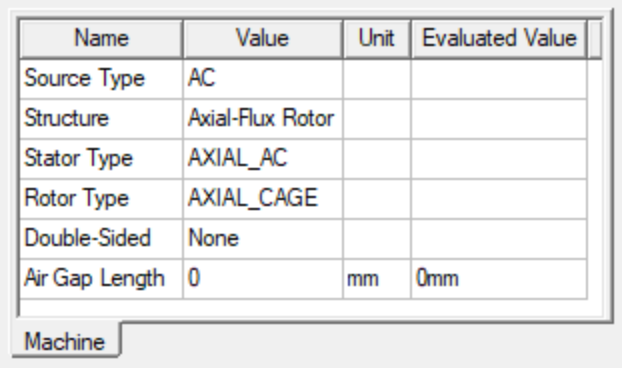
Defining an Axial-Flux Induction Machine

> **Note:** Refer to Analysis Approach for Three-Phase Induction Motors for general information on this machine type.

The general procedure for defining an Axial-Flux Induction Machine is as follows:

1. Insert an **Axial-Flux Induction Machine** into a new or existing project by choosing **Generate RMxprt Solutions** for the **Design Flow**, **General** for the **Machine Type**, then select **Axial-Flux Induction Machine** from the expandable list.
2. Click the **Machine** entry in the project tree to set the machine properties as shown:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Name	Value	Unit	Evaluated Value
Source Type	AC		
Structure	Axial-Flux Rotor		
Stator Type	AXIAL_AC		
Rotor Type	AXIAL_CAGE		
Double-Sided	None		
Air Gap Length	0	mm	0mm

Machine

3. Click to expand the **Machine** > **Stator** entry in the project tree to define data for the stator and its associated core and core slot, and winding for the AXIAL_AC core type.
4. Click to expand the **Machine** > **Rotor** entry in the project tree to define data for the rotor and its associated core, damper and damper slot for the AXIAL_CAGE core type.
5. Click the **Machine** > **Shaft** entry in the project tree to define the magnetism, frictional and windage losses, and reference speed of the shaft.
6. Right-click **Analysis** in the project tree, and click **Add Solution Setup** to define the solution data.
7. Choose **File > Save** to save the project.
8. Choose **RMxprt > Analyze** to analyze the design.

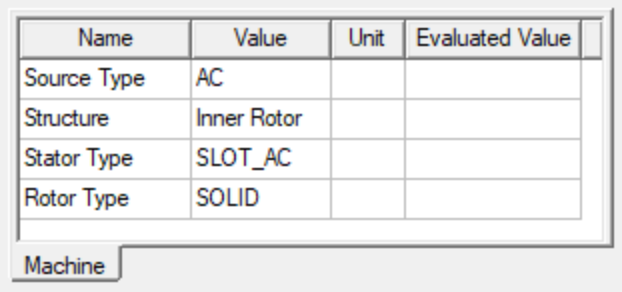
Once analyzed, the model can be used to create a new Maxwell 2D or 3D design.

Defining a Hysteresis Machine

The general procedure for defining a Hysteresis Machine is as follows:

1. Insert a **Hysteresis Machine** into a new or existing project by choosing **Generate RMxprt Solutions** for the **Design Flow**, **General** for the **Machine Type**, then select **Hysteresis Machine** from the expandable list.
2. Click the **Machine** entry in the project tree to set the machine properties as shown:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Click to expand the **Machine** > **Stator** entry in the project tree to define data for the stator and its associated core and core slot, winding, and optionally, vent for the SLOT_AC core type.
4. Click to expand the **Machine** > **Rotor** entry in the project tree to define data for the rotor core for the SOLID core type, where **Sleeve Type** is set to **Hysteresis**.
5. Click the **Machine** > **Shaft** entry in the project tree to define the magnetism, frictional and windage losses, and reference speed of the shaft.
6. Optionally, right-click the **Machine** entry in the project tree and click **Insert Housing** to add a machine housing.
7. Right-click **Analysis** in the project tree, and click **Add Solution Setup** to define the solution data.
8. Choose **File** > **Save** to save the project.
9. Choose **RMxprt** > **Analyze** to analyze the design.

Once analyzed, the model can be used to create a new Maxwell 2D or 3D design.

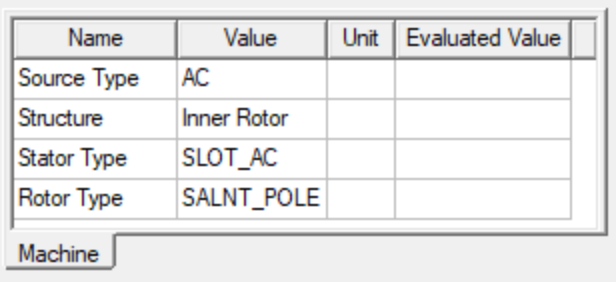
Defining a Rotating-Field Synchronous Machine

> **Note:** Refer to Analysis Approach for Three-Phase Synchronous Machines for general information on this machine type.

The general procedure for defining a Rotating-Field Synchronous Machine is as follows:

1. Insert a **Rotating-Field Synchronous Machine** into a new or existing project by choosing **Generate RMxprt Solutions** for the **Design Flow**, **General** for the **Machine Type**, then select **Rotating-Field Synchronous Machine** from the expandable list.
2. Click the **Machine** entry in the project tree to set the machine properties as shown:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Click to expand the **Machine** > **Stator** entry in the project tree to define data for the stator and its associated core and core slot, winding, and optionally, vent for the SLOT_AC core type.
4. Click to expand the **Machine** > **Rotor** entry in the project tree to define data for the rotor and its associated core, pole, optional brush, and optional damper and damper slot for the SALNT_POLE core type.
5. Click the **Machine** > **Shaft** entry in the project tree to define the magnetism, frictional and windage losses, and reference speed of the shaft.
6. Optionally, right-click the **Machine** entry in the project tree and click **Insert Housing** to add a machine housing.
7. Right-click **Analysis** in the project tree, and click **Add Solution Setup** to define the solution data.
8. Choose **File** > **Save** to save the project.
9. Choose **RMxprt** > **Analyze** to analyze the design.

Once analyzed, the model can be used to create a new Maxwell 2D or 3D design.

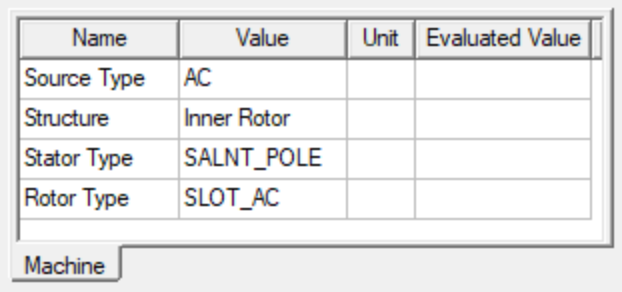
Defining a Rotating-ArmatureSynchronous Machine

> **Note:** Refer to Analysis Approach for Three-Phase Synchronous Machines for general information on this machine type.

The general procedure for defining a Rotating-Armature Synchronous Machine is as follows:

1. Insert a **Rotating-Armature Synchronous Machine** into a new or existing project by choosing **Generate RMxprt Solutions** for the **Design Flow**, **General** for the **Machine Type**, then select **Rotating-Armature Synchronous Machine** from the expandable list.
2. Click the **Machine** entry in the project tree to set the machine properties as shown:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Click to expand the **Machine** > **Stator** entry in the project tree to define data for the stator and its associated core, pole, and optional damper and damper slot for the SALNT_POLE core type.
4. Click to expand the **Machine** > **Rotor** entry in the project tree to define data for the stator and its associated core and core slot, winding, and optionally, vent for the SLOT_AC core type.
5. Click the **Machine** > **Shaft** entry in the project tree to define the magnetism, frictional and windage losses, and reference speed of the shaft.
6. Optionally, right-click the **Machine** entry in the project tree and click **Insert Housing** to add a machine housing.
7. Right-click **Analysis** in the project tree, and click **Add Solution Setup** to define the solution data.
8. Choose **File** > **Save** to save the project.
9. Choose **RMxprt** > **Analyze** to analyze the design.

Once analyzed, the model can be used to create a new Maxwell 2D or 3D design.

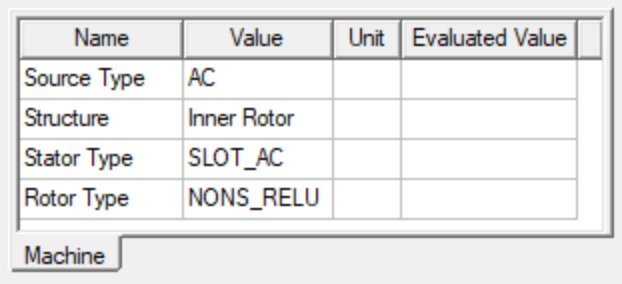
Defining a Reluctance Synchronous Machine

> **Note:** Refer to Analysis Approach for Three-Phase Synchronous Machines for general information on this machine type.

The general procedure for defining a Reluctance Synchronous Machine is as follows:

1. Insert a **Reluctance Synchronous Machine** into a new or existing project by choosing **Generate RMxprt Solutions** for the **Design Flow**, **General** for the **Machine Type**, then select **Reluctance Synchronous Machine** from the expandable list.
2. Click the **Machine** entry in the project tree to set the machine properties as shown:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Click to expand the **Machine** > **Stator** entry in the project tree to define data for the stator and its associated core and core slot, winding, and optionally, vent for the SLOT_AC core type.
4. Click to expand the **Machine** > **Rotor** entry in the project tree to define data for the rotor and its associated core and pole for the NONS_RELU core type.
5. Click the **Machine** > **Shaft** entry in the project tree to define the magnetism, frictional and windage losses, and reference speed of the shaft.
6. Optionally, right-click the **Machine** entry in the project tree and click **Insert Housing** to add a machine housing.
7. Right-click **Analysis** in the project tree, and click **Add Solution Setup** to define the solution data.
8. Choose **File** > **Save** to save the project.
9. Choose **RMxprt** > **Analyze** to analyze the design.

Once analyzed, the model can be used to create a new Maxwell 2D or 3D design.

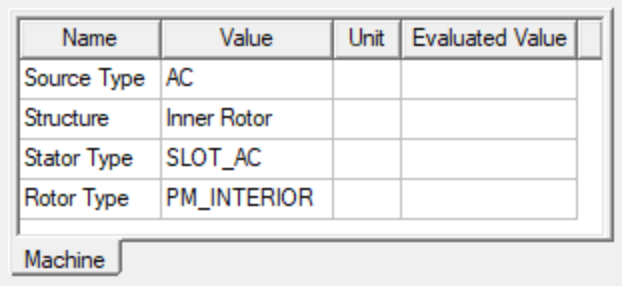
Defining an IPM Synchronous Machine

> **Note:** Refer to Analysis Approach for Three-Phase Synchronous Machines for general information on this machine type.

The general procedure for defining an IPM Synchronous Machine is as follows:

1. Insert an **IPM Synchronous Machine** into a new or existing project by choosing **Generate RMxprt Solutions** for the **Design Flow**, **General** for the **Machine Type**, then select **IPM Synchronous Machine** from the expandable list.
2. Click the **Machine** entry in the project tree to set the machine properties as shown:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Click to expand the **Machine** > **Stator** entry in the project tree to define data for the stator and its associated core and core slot, winding, and optionally, vent for the SLOT_AC core type.
4. Click to expand the **Machine** > **Rotor** entry in the project tree to define data for the rotor and its associated core and pole for the PM_INTERIOR core type.
5. Click the **Machine** > **Shaft** entry in the project tree to define the magnetism, frictional and windage losses, and reference speed of the shaft.
6. Optionally, right-click the **Machine** entry in the project tree and click **Insert Housing** to add a machine housing.
7. Right-click **Analysis** in the project tree, and click **Add Solution Setup** to define the solution data.
8. Choose **File** > **Save** to save the project.
9. Choose **RMxprt** > **Analyze** to analyze the design.

Once analyzed, the model can be used to create a new Maxwell 2D or 3D design.

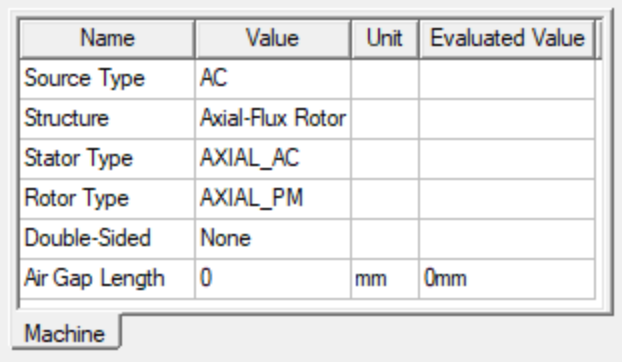
Defining an Axial-Flux PM Synchronous Machine

> **Note:** Refer to Analysis Approach for Three-Phase Synchronous Machines for general information on this machine type.

The general procedure for defining an Axial-Flux PM Synchronous Machine is as follows:

1. Insert an **Axial-Flux PM Synchronous Machine** into a new or existing project by choosing **Generate RMxprt Solutions** for the **Design Flow**, **General** for the **Machine Type**, then select **Axial-Flux PM Synchronous Machine** from the expandable list.
2. Click the **Machine** entry in the project tree to set the machine properties as shown:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Name	Value	Unit	Evaluated Value
Source Type	AC		
Structure	Axial-Flux Rotor		
Stator Type	AXIAL_AC		
Rotor Type	AXIAL_PM		
Double-Sided	None		
Air Gap Length	0	mm	0mm

Machine

3. Click to expand the **Machine** > **Stator** entry in the project tree to define data for the stator and its associated core and core slot, and winding for the AXIAL_AC core type.
4. Click to expand the **Machine** > **Rotor** entry in the project tree to define data for the rotor and its associated core, damper and damper slot for the AXIAL_PM core type.
5. Click the **Machine** > **Shaft** entry in the project tree to define the magnetism, frictional and windage losses, and reference speed of the shaft.
6. Right-click **Analysis** in the project tree, and click **Add Solution Setup** to define the solution data.
7. Choose **File** > **Save** to save the project.
8. Choose **RMxprt** > **Analyze** to analyze the design.

Once analyzed, the model can be used to create a new Maxwell 2D or 3D design.

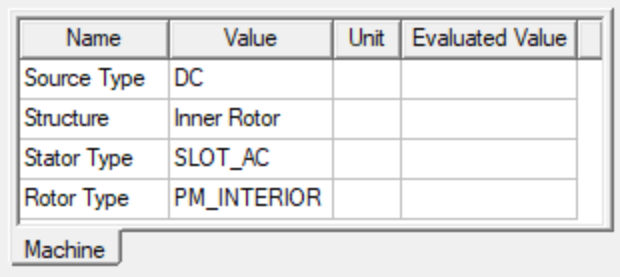
Defining an IPM Brushless DC Motor

> **Note:** Refer to Analysis Approach for Brushless PMDC Motors for general information on this machine type.

The general procedure for defining an IPM Brushless DC Motor is as follows:

1. Insert an **IPM Brushless DC Motor** into a new or existing project by choosing **Generate RMxprt Solutions** for the **Design Flow**, **General** for the **Machine Type**, then select **IPM Brushless DC Motor** from the expandable list.
2. Click the **Machine** entry in the project tree to set the machine properties as shown:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Click to expand the **Machine** > **Stator** entry in the project tree to define data for the stator and its associated core and core slot, winding, circuit, and optionally, vent for the SLOT_AC core type.
4. Click to expand the **Machine** > **Rotor** entry in the project tree to define data for the rotor and its associated core and pole for the PM_INTERIOR core type.
5. Click the **Machine** > **Shaft** entry in the project tree to define the magnetism, frictional and windage losses, and reference speed of the shaft.
6. Optionally, right-click the **Machine** entry in the project tree and click **Insert Housing** to add a machine housing.
7. Right-click **Analysis** in the project tree, and click **Add Solution Setup** to define the solution data.
8. Choose **File** > **Save** to save the project.
9. Choose **RMxprt** > **Analyze** to analyze the design.

Once analyzed, the model can be used to create a new Maxwell 2D or 3D design.

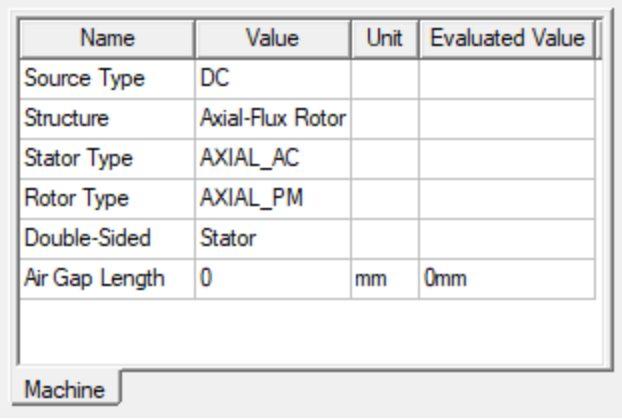
Defining an Axial-Flux Brushless DC Motor

> **Note:** Refer to Analysis Approach for Brushless PMDC Motors for general information on this machine type.

The general procedure for defining an Axial-Flux Brushless DC Motor is as follows:

1. Insert an **Axial-Flux Brushless DC Motor** into a new or existing project by choosing **Generate RMxprt Solutions** for the **Design Flow**, **General** for the **Machine Type**, then select **Axial-Flux Brushless DC Motor** from the expandable list.
2. Click the **Machine** entry in the project tree to set the machine properties as shown:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Click to expand the **Machine** > **Stator** entry in the project tree to define data for the stator and its associated core and core slot, and winding for the AXIAL_AC core type.
4. Click to expand the **Machine** > **Rotor** entry in the project tree to define data for the rotor and its associated core, damper and damper slot for the AXIAL_PM core type.
5. Click the **Machine** > **Shaft** entry in the project tree to define the magnetism, frictional and windage losses, and reference speed of the shaft.
6. Right-click **Analysis** in the project tree, and click **Add Solution Setup** to define the solution data.
7. Choose **File > Save** to save the project.
8. Choose **RMxprt > Analyze** to analyze the design.

Once analyzed, the model can be used to create a new Maxwell 2D or 3D design.

Generic Rotating Machine Core Types

The Generic Rotating Machine (GRM) templates allow you to define any of several machine types using the core types listed below.

Core Type	Description
SLOT_AC	Slotted core with AC winding
SLOT_CAGE	Slotted core with squirrel-cage damper
SALNT_POLE	Salient-pole core with field winding
PM_INTERIOR	Interior permanent magnet core
SOLID	Solid core with/without copper/hysteresis sleeve
AXIAL_AC	Axial-flux slotted core with AC winding
AXIAL_CAGE	Axial-flux slotted core with squirrel-cage

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Core Type	Description
	damper
AXIAL_PM	Axial-flux permanent magnet core
NONS_RELU	Non-salient reluctance core without winding

Note: Refer to Defining a Generic Rotating Machine for detailed information on selecting GRM templates and defining the corresponding machine types that use these core types.

Defining Data for the SLOT_AC Core Type

The general procedure for defining a SLOT_AC core type is as follows:

1. Double-click the **Machine** > **Stator** (or **Machine** > **Rotor**, if applicable) entry in the project tree to define data for the stator (or rotor). The **Stator** (or **Rotor**) **Properties** dialog box contains the following fields:

Number of Poles — The number of poles on which the winding is wound. Default is 2.

Number of Slots — The number of slots of the iron core. Default is 18.

Circuit Type — The drive circuit type.

Select from the following six types (default is Y3):

- Y3: Y-Type, 3-Phase
- L3: Loop-Type, 3-Phase
- S3: Star-Type, 3-Phase
- C2: Cross-Type, 2-Phase
- L4: Loop-Type, 4-Phase
- S4: Star-Type, 4-Phase

Note: When you place the mouse cursor over the circuit type, a circuit diagram of the selected type appears in the dialog.

Slot Type — The slot type of the iron core. Click the **Slot Type** button to open the **Select Slot Type** dialog box. Select the desired type. Default is Type 1.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** When you place the mouse cursor over the slot type, a diagram of the selected type appears in the dialog box, displaying the slot dimension variables.

Optionally, check **User Defined Slot** if you wish to define the slot dimensions using the **Slot Editor**.

Position Control	Determines if the drive circuit is controlled by position signals from a position sensor. (Checked for **IPM Brushless DC Motor** core type. Unchecked for all other types.)

2. Double-click the **Core** entry in the project tree to define data for the stator core. The core data **Properties** dialog box contains the following fields:

Outer Diameter	The outer diameter of the core
Inner Diameter	The inner diameter of the core
Length	The length of the core
Stacking Factor	The effective magnetic length of the core. Default is 0.95.
Steel Type	The steel type of the core. Click the button to open the **Select Definition** window and select the desired material.
Press Board Thickness	The thickness of the pole press boards
Magnetic Press Board	Check the box if the press board is made of magnetic material.
Skew Width	The skew width measured in degrees
Lamination Sectors	The number of lamination sectors

3. Double-click the **Slot** entry in the project tree to define the stator slot dimensions. Enter the available slot dimensions.

   An image of the selected slot type showing the dimension labels is displayed on the Diagram tab. The dimensions properties that appear depend on the **Slot Type** (selected above) and on whether or not **Auto Design** or **Parallel Tooth** is selected:

Auto Design	Check the box to have RMxprt auto design the Hs2, Bs1, and Bs2 dimensions.
Parallel Tooth	Check the box to have RMxprt design Bs1 and Bs2 based on **Tooth Width**. Available only when **Auto Design** is cleared.
Tooth Width	The tooth width for parallel tooth. Available only when **Parallel Tooth** is selected.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Hs0	Always available
Hs1	Available only when the slot type is 2, 3, 4, 5, or 6
Hs2	Available only when **Auto Design** is cleared. When **Auto Design** is selected, this slot dimension is determined automatically.
Bs0	Available only when the slot type is 1, 2, 3, 4, or 5
Bs1	Available only when **Auto Design** is cleared. When **Auto Design** is selected, this slot dimension is determined automatically. When **Parallel Tooth** is selected, this slot dimension is determined based on the value entered in the **Tooth Width** field.
Bs2	Available only when **Auto Design** is cleared. When **Auto Design** is selected, this slot dimension is determined automatically. When **Parallel Tooth** is selected, this slot dimension is determined based on the value entered in the **Tooth Width** field.
Rs	Available only when the slot type is 3 or 4

4. Double-click the **Winding** entry in the project tree to define the conductors, windings, and insulation data for the winding
5. For an IPM Brushless DC Motor stator only, double-click the **Circuit** entry in the project tree to define the properties for the position control circuit.
6. Optionally, you can right-click on Core in the project tree to insert or remove Vent data.

Defining Data for the SLOT_CAGE Core Type

The general procedure for defining a SLOT_CAGE core type is as follows:

1. Double-click the **Machine** > **Rotor** entry in the project tree to define data for the rotor. In the **Rotor Properties** dialog box, enter the number of slots in the **Number of Slots** field. There are no additional settings.
2. Double-click the **Core** entry in the project tree to define data for the rotor core. The core data **Properties** dialog box contains the following fields:

Outer Diameter	The outer diameter of the core
Inner Diameter	The inner diameter of the core
Length	The length of the core
Stacking Factor	The effective magnetic length of the core. Default is 0.95.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Steel Type	The steel type of the core. Click the button to open the **Select Definition** window and select the desired material.
Press Board Thickness	The thickness of the pole press boards
Magnetic Press Board	Check the box if the press board is made of magnetic material
Skew Width	The skew width measured in degrees
Half Slot	Check the box to set the rotor damper slots to half slots.
Double Cage	Check the box to add a bottom damper.

3. Double-click the **Machine** > **Rotor** > **Core** > **Damper** (or **Bottom Damper**) entry in the project tree to open the **Properties** dialog box. (You can also enter values for the following properties in the **Properties** window of the desktop without opening a separate dialog box.)

Slot Type	The slot type of the damper. Click the **Slot Type** button to open the **Select Slot Type** dialog box. Select the desired type. Default is Type 1.

> **Note:** When you place the mouse cursor over the slot type, a diagram of the selected type appears in the dialog box, displaying the slot dimension variables.

Cast Cage	Check the box if the cage is cast; not available for **Bottom Damper**.
Bar Fitting Gap	The gap between the slot edge and bar edge
Bar End Extension	The single-side end extended bar length; not available for **Bottom Damper**.
End Ring Width	The axial length of the end ring, not available for **Bottom Damper**.
End Ring Height	The radial height of the end ring, not available for **Bottom Damper**.
Bar Conductor Type	Click the button to open the **Select Definition** window, and select the desired conductor material.
End Ring Conductor	Click the button to open the **Select Definition** window, and select the desired conductor material.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type	Not available for **Bottom Damper**.

4. Double-click the **Damper** > **Slot** (and **Bottom Damper** > **Slot**, if used) entry in the project tree to define the slot dimensions. Enter the slot dimensions.

   An image of the selected slot type showing the dimension labels is displayed on the Diagram tab. Below are the slot dimensions properties that appear:

Hs0	Slot dimension Hs0
Hs01	Slot dimension Hs01
Hs2	Slot dimension Hs2
Bs0	Slot dimension Bs0
Bs1	Slot dimension Bs1
Bs2	Slot dimension Bs2

5. Optionally, you can right-click on Core in the project tree to insert or remove Vent data.

Defining Data for the SOLID Core Type

To define a SOLID core type, double-click the **Machine** > **Rotor** > **Core** entry in the Project Manager tree.

The **Core Properties** dialog box contains the following fields:

Outer Diameter	The outer diameter of the core
Inner Diameter	The inner diameter of the core
Length	The length of the core
Stacking Factor	The effective magnetic length of the core; the default is 0.95.
Steel Type	The steel type of the core. Click the button to open the **Select Definition** window and select the desired material.
Sleeve Type	Select the Sleeve type: • **No Sleeve** No Sleeve is selected by default if the machine type is **Solid-Rotor Induction**. • **Hysteresis** Hysteresis will be selected by default if the machine type is **Hysteresis Machine**. • **Conductor**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sleeve Thickness	The sleeve thickness; available only when the **Sleeve Type** is either **Hysteresis** or **Conductor**.
Sleeve Material	The sleeve material type of the core. Click the button to open the **Select Definition** window and select the desired material. If the **Sleeve Type** is **Hysteresis**, the message manager informs you if your choice is not a hysteresis type material. Similarly, If the **Sleeve Type** is **Conductor**, the message manager informs you if your choice is not a conductor type material. (Available only when the **Sleeve Type** is either **Hysteresis** or **Conductor**.)

Defining Data for the AXIAL_AC Core Type

To define an AXIAL_AC core type is as follows:

1. Double-click the **Machine** > **Stator** entry in the project tree to define data for the stator. The **Stator Properties** dialog box contains the following fields:

Number of Poles	The number of poles on which the winding is wound. Default is 2.
Number of Slots	The number of slots of the iron core. Default is 18.
Circuit Type	The drive circuit type. Select from the following six types (default is Y3): • Y3: Y-Type, 3-Phase • L3: Loop-Type, 3-Phase • S3: Star-Type, 3-Phase • C2: Cross-Type, 2-Phase • L4: Loop-Type, 4-Phase • S4: Star-Type, 4-Phase **Note:** When you place the mouse cursor over the circuit type, a circuit diagram of the selected type appears in the dialog,
Slot Type	The slot type of the iron core. Click the **Slot Type** button to open the **Select Slot Type** dialog box. Select the desired type. Default is Type 1.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** When you place the mouse cursor over the slot type, a diagram of the selected type appears in the dialog box, displaying the slot dimension variables.

Optionally, check **User Defined Slot** if you wish to define the slot dimensions using the **Slot Editor**.

Position Control	Determines if the drive circuit is controlled by position signals from a position sensor. (Checked for **Axial-Flux Brushless DC Motor** core type. Unchecked for all other types.)

2. Double-click the **Core** entry in the project tree to define data for the stator core. The core data **Properties** dialog box contains the following fields:

Outer Diameter	The outer diameter of the core
Inner Diameter	The inner diameter of the core
Length	The length of the core
Stacking Factor	The effective magnetic length of the core; default is 0.95.
Steel Type	The steel type of the core. Click the button to open the **Select Definition** window and select the desired material.

3. Double-click the **Slot** entry in the project tree to define the stator slot dimensions. Enter the available slot dimensions.
An image of the selected slot type showing the dimension labels is displayed on the Diagram tab. The dimensions properties that appear depend on the **Slot Type** (selected above) and on whether or not **Auto Design** or **Parallel Tooth** is selected:

Auto Design	Check the box to have RMxprt auto design the Hs2, Bs1, and Bs2 dimensions.
Parallel Tooth	Check the box to have RMxprt design Bs1 and Bs2 based on **Tooth Width**. Available only when **Auto Design** is cleared.
Tooth Width	The tooth width for parallel tooth. Available when **Parallel Tooth** is selected.
Hs0	Always available
Hs1	Available only when the slot type is 2, 3, 4, 5, or 6
Hs2	Available only when **Auto Design** is cleared. When **Auto Design** is selected, this slot dimension is determined automatically.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Bs0	Available only when the slot type is 1, 2, 3, 4, or 5
Bs1	Available only when **Auto Design** is cleared. When **Auto Design** is selected, this slot dimension is determined automatically. When **Parallel Tooth** is selected, this slot dimension is determined based on the value entered in the **Tooth Width** field.
Bs2	Available only when **Auto Design** is cleared. When **Auto Design** is selected, this slot dimension is determined automatically. When **Parallel Tooth** is selected, this slot dimension is determined based on the value entered in the **Tooth Width** field.
Rs	Available only when the slot type is 3 or 4

4. Double-click the **Winding** entry in the project tree to define the conductors, windings, and insulation data for the winding.
5. For an Axial-Flux Brushless DC Motor stator only, double-click the **Circuit** entry in the project tree to define the properties for the position control circuit.

Defining Data for the AXIAL_CAGE Core Type

The general procedure for defining a AXIAL_CAGE core type is as follows:

1. Double-click the **Machine** > **Rotor** entry in the project tree to define data for the rotor. In the **Rotor Properties** dialog box, enter the number of slots in the **Number of Slots** field. There are no additional settings.
2. Double-click the **Core** entry in the project tree to define data for the rotor core. The core data **Properties** dialog box contains the following fields:

Outer Diameter	The outer diameter of the core
Inner Diameter	The inner diameter of the core
Length	The length of the core
Stacking Factor	The effective magnetic length of the core; default is 0.95.
Steel Type	The steel type of the core. Click the button to open the **Select Definition** window and select the desired material.
Press Board Thickness	The thickness of the pole press boards
Magnetic	Check the box if the press board is made of

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Press Board	magnetic material.
Skew Width	The skew width measured in degrees
Half Slot	Check the box to set the rotor damper slots to half slots.
Double Cage	Check the box to add a bottom damper.

3. Double-click the **Machine** > **Rotor** > **Core** > **Damper** (or **Bottom Damper**) entry in the project tree to open the **Properties** dialog box. (You can also enter values for the following properties in the **Properties** window of the desktop without opening a separate dialog box.)

Slot Type	The slot type of the damper. Click the **Slot Type** button to open the **Select Slot Type** dialog box. Select the desired type. Default is Type 1. **Note:** When you place the mouse cursor over the slot type, a diagram of the selected type appears in the dialog box, displaying the slot dimension variables.
Cast Cage	Check the box if the cage is cast. Not available for **Bottom Damper**.
Bar Fitting Gap	The gap between the slot edge and bar edge
Bar End Extension	The single-side end extended bar length. Not available for **Bottom Damper**.
End Ring Width	The axial length of the end ring. Not available for **Bottom Damper**.
End Ring Height	The radial height of the end ring. Not available for **Bottom Damper**.
Bar Conductor Type	Click the button to open the **Select Definition** window and select the desired conductor material.
End Ring Conductor Type	Click the button to open the **Select Definition** window and select the desired conductor material. Not available for **Bottom Damper**.

4. Double-click the **Damper** > **Slot** (and **Bottom Damper** > **Slot**, if used) entry in the project tree to define the slot dimensions. Enter the slot dimensions.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

An image of the selected slot type showing the dimension labels is displayed on the Diagram tab. Below are the slot dimensions properties that appear:

Hs0	Slot dimension Hs0
Hs01	Slot dimension Hs01
Hs2	Slot dimension Hs2
Bs0	Slot dimension Bs0
Bs1	Slot dimension Bs1
Bs2	Slot dimension Bs2

Defining Data for the SALNT_POLE Core Type

The general procedure for defining a SALNT_POLE core type is as follows:

1. Double-click the **Machine** > **Rotor** (or **Machine** > **Stator**, if applicable) entry in the project tree to define data for the rotor (or stator). In the **Rotor** (or **Stator**) **Properties** dialog box, enter the number of poles in the **Number of Poles** field (default is 2). There are no additional settings.
2. Double-click the **Core** entry in the project tree to define data for the rotor or stator core. The core data **Properties** dialog box contains the following fields:

Outer Diameter	Outer diameter of the core
Inner Diameter	Inner diameter of the core
Length	Length of the core
Stacking Factor	Effective magnetic length of the core; default is 0.95.
Steel Type	Steel type of the core; click the button to open the **Select Definition** window and select the desired material.
Press Board Thickness	Thickness of the pole press boards
Magnetic Press Board	Check the box if the press board is made of magnetic material.
Skew Width	Skew width measured in degrees
Pole Type	Click the **Pole Type** button to open the **Select Pole Type** dialog box. The SALNT_POLE rotor core type supports one pole type: Type 1; while the SALNT_POLE stator supports two pole types. A

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

diagram displaying the pole dimension variables appears on the Diagram tab.

> **Note:** When you place the mouse cursor over the pole type, a diagram of the selected type appears in the dialog box, displaying the pole dimension variables.

3. Double-click the **Machine** > **Rotor** > **Core** > **Pole** or **Machine** > **Stator** > **Core** > **Pole** entry in the project tree to open its **Properties** dialog box.

Second Air Gap	Second air gap between the pole body and the yoke
Pole Body Width	Pole body width dimension
Pole Body Height	Pole body height dimension
Pole Shoe Width	Pole shoe width dimension
Pole Shoe Height	Pole shoe height dimension
Pole Arc Offset	Pole-arc center offset from the rotor center
Second Pole Arc	Check if you want the pole surface to include a second arc or line in a half-pole range. When checked, enables Off2_x and Off2_y.
Off2_x	Offset of the second arc perpendicular to the pole-center line Available when **Second Pole Arc** is checked
Off2_y	Offset of the second arc in parallel with the pole-center line Available when **Second Pole Arc** is checked
Stacking Factor	Effective magnetic length of the rotor pole
Steel Type	Steel material type; click the button to open the **Select Definition** window and select the steel material type from the list. Use the **Material Filter** tab settings to filter for **Steel** materials.
Bp1	Pole dimension Bp1 Available only for stator core **Pole Type** 1

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Bp2	Pole dimension Bp2 – Available only for stator core **Pole Type** 1
Rp1	Pole dimension Rp1 Available only for stator core **Pole Type** 2
Rp2	Pole dimension Rp2 Available only for stator core **Pole Type** 2

4. Optionally, you can add a damper by right-clicking **Core** in the project tree and selecting **Insert Damper**. You can then double-click the **Machine** > **Rotor** > **Core** > **Damper** entry in the project tree to open the **Properties** dialog box. (You can also enter values for the following properties in the **Properties** window of the desktop without opening a separate dialog box.)

Damper Slots Per Pole	The number of damper slots per pole
Slot Type	The slot type of the damper. Click the **Slot Type** button to open the **Select Slot Type** dialog box. Select the desired type. Default is Type 1. **Note:** When you place the mouse cursor over the slot type, a diagram of the selected type appears in the dialog box, displaying the slot dimension variables.
Cast Rotor	Check the box if the cage is cast.
Bar Conductor Type	Click the button to open the **Select Definition** window and select the desired conductor material.
End Length	The single-side end bar length. Available only when the **End Ring Type** is 2 or 3.
End Ring Width	The axial length of the end ring. Available only when the **End Ring Type** is 2 or 3.
End Ring Height	The radial height of the end ring. Available only when the **End Ring Type** is 2 or 3.
End Ring Conductor Type	Click the button to open the **Select Definition** window and select the desired conductor material. Not available for **Bottom Damper**.
End Ring Type	Click the button to open the **Select Definition** window and select the desired end ring type.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** When you place the mouse cursor over the end ring type, an explanation appears in the dialog box, describing the end ring.

Slot Pitch	The slot pitch for the damper slots
Center Slot Pitch	The center slot pitch for the damper slots
Bar Fitting Gap	The gap between the slot edge and bar edge

5. If a damper is present, double-click the **Damper** > **Slot** entry in the project tree to define the slot dimensions. Enter the slot dimensions.

   An image of the selected slot type showing the dimension labels is displayed on the Diagram tab. Below are the slot dimensions properties that appear:

Hs0	Slot dimension Hs0
Hs01	Slot dimension Hs01
Hs2	Slot dimension Hs2
Bs0	Slot dimension Bs0
Bs1	Slot dimension Bs1
Bs2	Slot dimension Bs2

6. Double-click the **Machine** > **Rotor** > **Field** or **Machine** > **Stator** > **Field** entry in the project tree to open its **Properties** dialog box and define the Field Data.

7. Double-click the **Machine** > **Rotor** > **Field** > **Winding** or **Machine** > **Stator** > **Field** > **Winding** entry in the project tree to open its **Properties** dialog box and define the Field Winding Data.

8. Optionally for machine rotors only, you can right-click on **Machine** > **Rotor** in the project tree and select **Insert Brush** to define brush data. Double-click **Brush** in the project tree to open its **Properties** dialog box contains the following fields:

Diameter	Brush surface diameter
Brush Width	Brush width
Brush Length	Brush length
Brush Drop	Voltage drop of a brush
Contact Resistance	Contact resistance of a brush
Brush Press	Brush pressure per unit area
Frictional	Frictional coefficient of a brush

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coefficient

Defining SALNT_POLE Stator and Rotor Field Data for a Generic Rotating Machine.

To define the stator and rotor windings and insulation data:

1. To open the Stator or Rotor Field **Properties** window, double-click the **Machine** > **Stator** > **Field** entry in the Project Manager tree. (You can also enter values in the **Properties** pane in the Desktop without opening a separate window.)
2. Enter the thickness of the insulation under the pole shoe in the **Shoe Insulation** field.
3. Enter the thickness of the insulation at the pole body side in the **Pole Insulation** field.
4. Enter the minimum gap in the **Winding Clearance** field. The winding clearance is one of the following: the minimum gap between two field windings, or the minimum gap between a field winding and a commutating winding.
5. Enter the exciter efficiency value in the **Exciter Efficiency** field.
6. Enter the exciting current value in the **Exciting Current** field.
7. Click **OK** to close the **Properties** window.

Defining SALNT_POLE Stator and Rotor Field Winding Data for a Generic Rotating Machine.

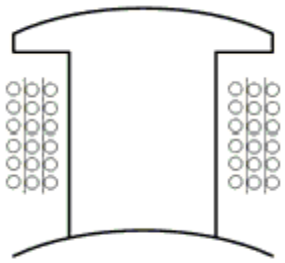
> **Note:** Not applicable for SALNT_STEP stators.

1. To open the rotor or stator field winding **Properties** dialog box, double-click the **Machine** > **Stator** > **Field** > **Winding** or **Machine** > **Rotor** > **Field** > **Winding** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate dialog box.)
2. Select a **Winding Type**:
   a. Click the button for **Winding Type** to open the **Winding Type** window.
   b. Select **Round**, **Cylinder**, or **EdgeWise** winding.

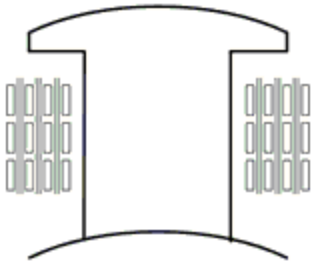
      When you place the mouse cursor over a winding button, an outline of the selected winding appears. The following table describes the three types of windings:

Type	Description
Round	A Round winding:

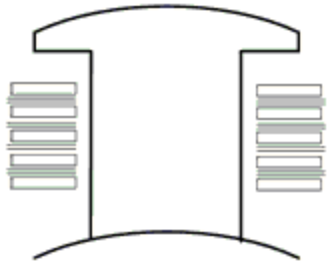
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Cylinder A Cylinder winding:



EdgeWise An EdgeWise winding:

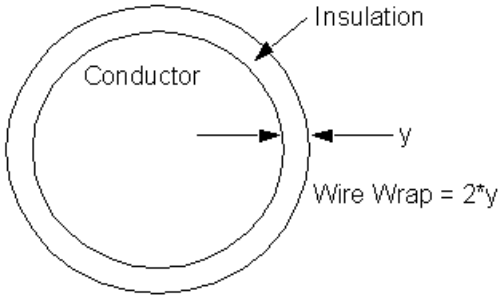


3. Once you have selected a winding, click **OK** to close the **Winding Type** dialog box and return to the **Properties** window.
4. Enter the number of parallel branches in one phase of the winding in the **Parallel Branches** field.
5. Enter the total number of conductors for each pole in the **Conductors per Pole** field. Odd numbers for input and output leads on different sides. Enter 0 to have RMxprt auto-design this value.
6. For Round and Cylinder winding types, enter the number of wires per conductor in the **Number of Strands** field. Enter 0 to have RMxprt auto-design this value.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** The **Number of Strands** field is invisible (not used) when the EdgeWise winding type has been selected.

7. Enter the thickness of the double-sided wire wrap in the **Wire Wrap** field. Enter 0 to automatically obtain this value from the wire library.



8. For an **EdgeWise** winding type, enter the thickness of the inter-turn insulation in the **Inter-turn Insulation** field.

> **Note:** The **Inter-turn Insulation** field is invisible (not used) when either the Round or Cylinder winding type has been selected.

9. Select the **Wire Size**:
   a. Click the button for **Wire Size** to open the **Wire Size** dialog box.
   b. Select a value from the **Wire Diameter** drop-down menu.
   c. Select a wire gauge from the **Gauge** drop-down menu. You can select from the following options:

<number>	You can select a specific gauge number. When you select a gauge number, the **Wire Diameter** field is automatically updated.
USER	This option allows you to manually enter the **Wire Diameter**. This is useful when you want to enter a diameter that does not correspond to a particular wire gauge.
AUTO	This option sets the **Wire Diameter** to zero, and RMxprt automatically calculates the optimal value. The diameter information is then written to the output file when you analyze the design.
MIXED	This option allows you to define a conductor that is made of different size wires. For example, a single conductor may consist of five wires, three wires with a diameter of 0.21 mm and 2 wires with a diameter of 0.13 mm.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The gauge number is based on AWG settings. You can create your own wire table using **Machine > Wire**, and then you can select this wire table using the **Tools** > **Options** > **General Options** > **Machines** command.

d. When you are done setting the wire size, click **OK** to close the **Wire Size** dialog box and return to the **Properties** dialog box.

10. When you are done setting the wire size, click **OK** to close the **Wire Size** dialog box and return to the **Properties** dialog box.
11. Click the **Conductor Type** button to open the **Select Definition** window **Materials** tab and selecting the desired conductor material type. Use the **Material Filter** tab settings to filter for **Conductor** materials. Click **OK** to close the window.
12. Enter the axial gap between the field winding and pole body in the **Axial Clearance** field.
13. Enter the inner radius at winding fillet in the **Winding Fillet** field.
14. Enter the inner diameter of the coil tip in the **Tip Inner Diameter** field.
15. Enter the limited cross-section width for winding design or arrangement in the **Limited Cross Width** field. Enter 0 for maximum available area.
16. Enter the limited cross-section height for winding design or arrangement in the **Limited Cross Height** field. Enter 0 for maximum available area.
17. Click **OK** to close the **Properties** dialog box.

Defining Data for the NONS_RELU Core Type

The general procedure for defining a NONS_RELU core type is as follows:

1. Double-click the **Machine** > **Rotor** entry in the Project Manager tree to define data for the rotor. In the **Rotor Properties** window, enter the number of poles in the **Number of Poles** field (default is 2). There are no additional settings.

2. Double-click the **Core** entry in the Project Manager tree to define data for the rotor or stator core. The core data **Properties** window contains the following fields:

Outer Diameter	Outer diameter of the core
Inner Diameter	Inner diameter of the core
Length	Length of the core
Stacking Factor	Effective magnetic length of the core; default is 0.95.
Steel Type	Steel type of the core; click the button to open the **Select Definition** window and select the desired material.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Press Board Thickness	Thickness of the pole press boards
Magnetic Press Board	Check the box if the press board is made of magnetic material.
Skew Width	Skew width measured in degrees
Pole Type	Click the **Pole Type** button to open the **Select Pole Type** dialog box. The NONS_RELU rotor core type supports four pole types: • 1 – Laminated • 2 – Concentric-polyline • 3 – Hyperbolic-curve • 4 – Hyperbolic-polyline Default type is 4 – Hyperbolic-polyline.

> **Note:** When you place the mouse cursor over the pole type, a diagram of the selected type appears in the dialog.

3. Double-click the **Machine** > **Rotor** > **Core** > **Pole** entry in the project tree to open its **Properties** dialog box, which contains the following fields:

> **Note:** The currently selected pole type diagram displays on the machine editor window Diagram tab.

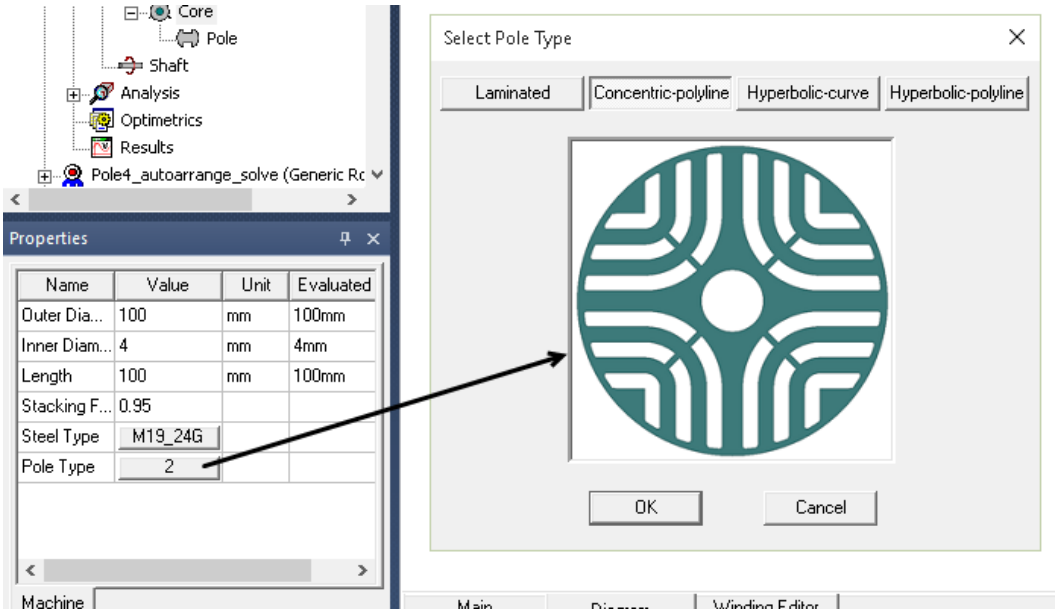
Barriers per Pole	Number of barriers in one pole. Invisible when pole type is 1 (Laminated).
H	Bridge thickness. Invisible when pole type is 1 (Laminated).
W	Rib width at the barrier center. Invisible when pole type is 1 (Laminated).
R	Barrier fillet radius. Invisible when pole type is 1 (Laminated).
R0	Radius of the bottom barrier arch for Pole Type 2 (Concentric-polyline); or line connecting arch radius for Pole Type 4 (Hyperbolic-polyline). Visible only when Pole Type is 2 or 4.
Rb	In-circle radius of the bottom barriers.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

B0	First (from inner to outer) barrier width at the barrier center. Invisible when pole type is 1 (Laminated).
Y0	First yoke width (between the first and the second barriers) at the barrier center.
Barrier Auto Arrangement	Check the box to let barriers be auto-arranged. Invisible when pole type is 1 (Laminated).
Extra Barriers	For pole types 2, 3, or 4, if the number of **Barriers per Pole** is greater than 1, and **Barrier Auto Arrangement** has not been enabled, the **Extra BarriersSetup** button opens the **Barriers Setup** dialog in which you can specify the **Bi** (bottom widths of the additional barriers) and **Yi** (bottom widths of the additional yokes) dimensions. Invisible if the number of **Barriers per Pole** is 1, or if **Barrier Auto Arrangement** has been enabled.

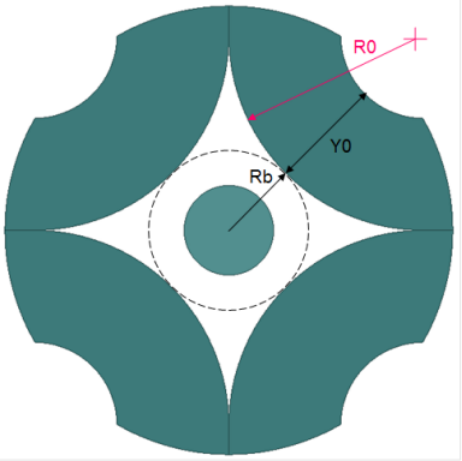
NONS_RELU Rotor Core Pole Types

The NONS_RELU rotor core type supports four pole types. You can choose the pole type by selecting the **Machine** > **Rotor** > **Core** entry in the project tree, then clicking the **Pole Type** button in its Properties window to open the **Select Pole Type** window.

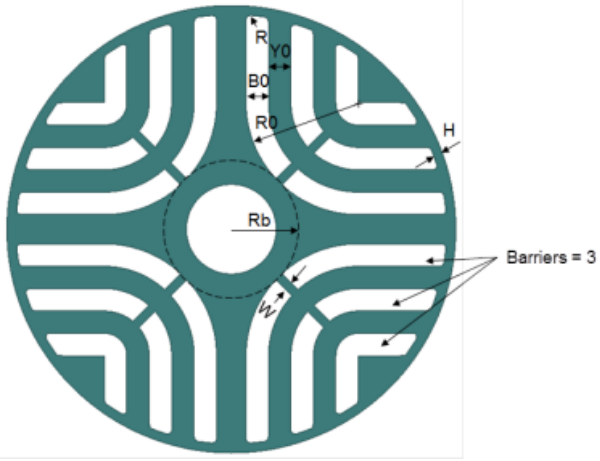


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

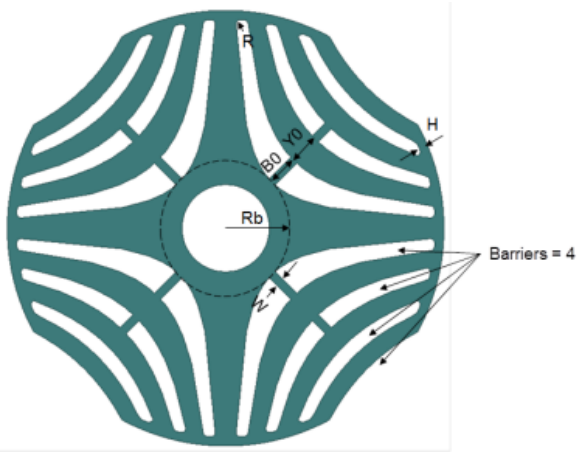
The four available pole types are shown below. Refer to *Defining Data for the NONS_RELU Core Type* for details on defining the various pole properties.



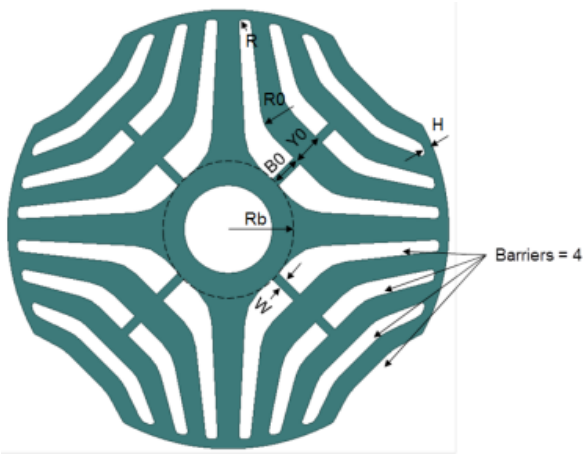
Laminated (Type 1)



Concentric-polyline (Type 2)



Hyperbolic-curve (Type 3)



Hyperbolic-polyline (Type 4)

Defining Data for the PM_INTERIOR Core Type

The general procedure for defining a PM_INTERIOR core type is as follows:

1. Double-click the **Machine** > **Rotor** entry in the project tree to define data for the rotor. In the **Rotor Properties** dialog box, enter the number of poles in the **Number of Poles** field

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(default is 4). There are no additional settings.

2. Double-click the **Core** entry in the project tree to define data for the rotor or stator core. The core data **Properties** dialog box contains the following fields:

Outer Diameter	Outer diameter of the core
Inner Diameter	Inner diameter of the core
Length	Length of the core
Stacking Factor	Effective magnetic length of the core; default is 0.95.
Steel Type	Steel type of the core; click the button to open the **Select Definition** window and select the desired material.
Pole Type	Click the **Pole Type** button to open the **Select Pole Type** dialog box. The PM_INTERIOR rotor core type supports six pole types: Default type is 3.

> **Note:** When you place the mouse cursor over the pole type, a diagram of the selected type appears in the dialog.

3. Double-click the **Machine** > **Rotor** > **Core** > **Pole** entry in the project tree to open its **Properties** dialog box, which contains the following fields:

> **Note:** The currently selected pole type diagram displays on the machine editor window Diagram tab.

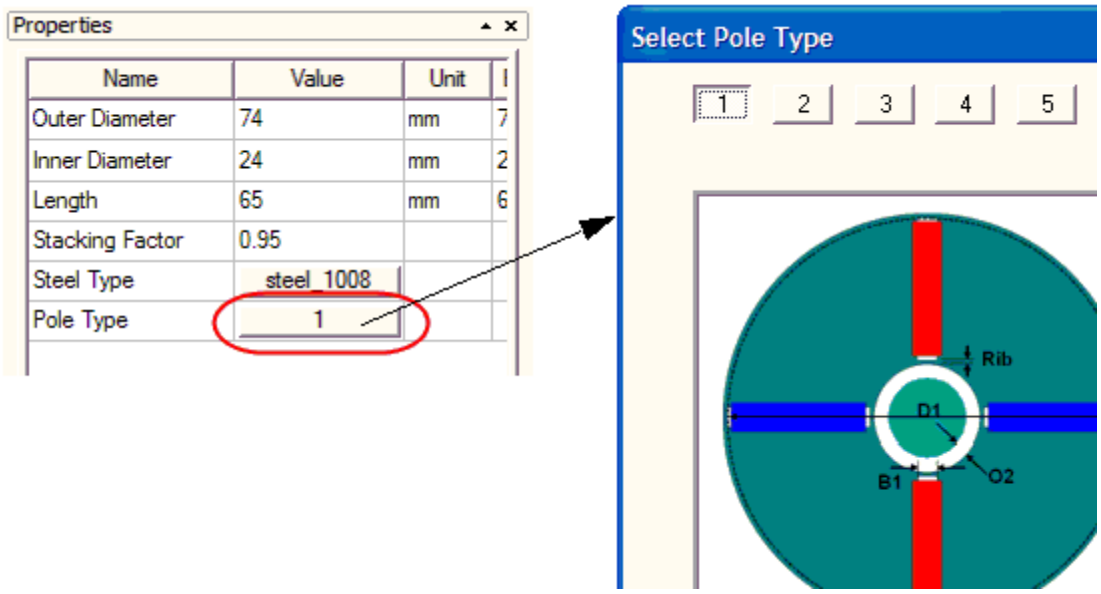
D1	Limited diameter for magnet ducts
O1	Magnet duct dimension; invisible when pole type is 1 or 2.
O2	Magnet duct dimension
B1	Magnet duct dimension
Rib	Magnet duct dimension
HRib	Magnet duct dimension; invisible when pole type is 1,2, or 6.
Layers	Number of duct layers
Layer Pitch	Pitch between two duct layers

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Magnet Thickness	Magnet thickness, or duct thickness
Magnet Width	Total width of all magnets per pole
Magnet Type	Magnet material type; click the button to open the **Select Definition** window and select the magnet material type from the list. Use the **Material Filter** tab settings to filter for **Magnet** materials.

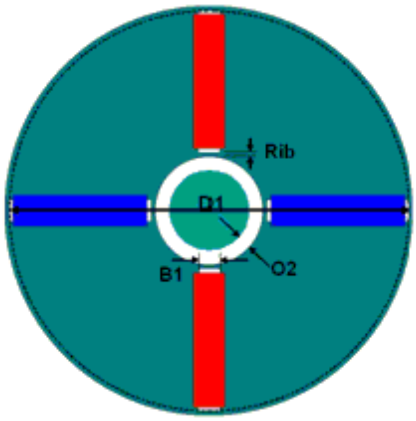
PM_INTERIOR Rotor Core Pole Types

The PM_INTERIOR rotor core type supports six pole types. You can choose the pole type by selecting the **Machine** > **Rotor** > **Core** entry in the Project Manager tree, then clicking the **Pole Type** button in its Properties window to open the **Select Pole Type** window.

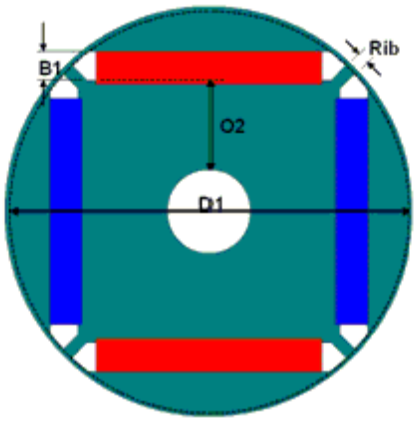


The six available pole types are shown below. Refer to *Defining Data for the PM_INTERIOR Core Type* for details on defining the various pole properties.

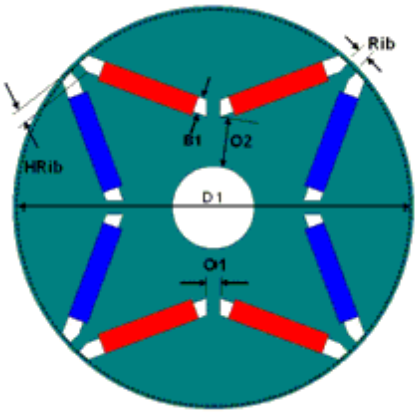
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



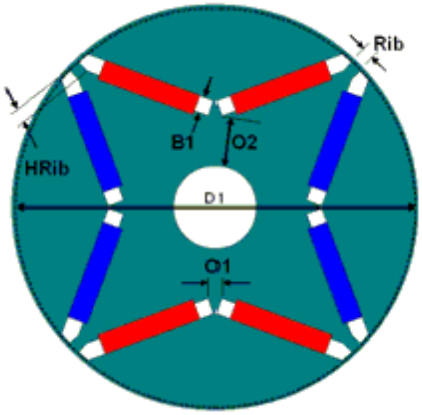
Pole Type 1



Pole Type 2

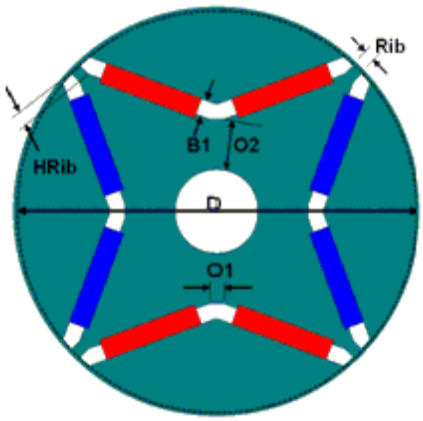


Pole Type 3

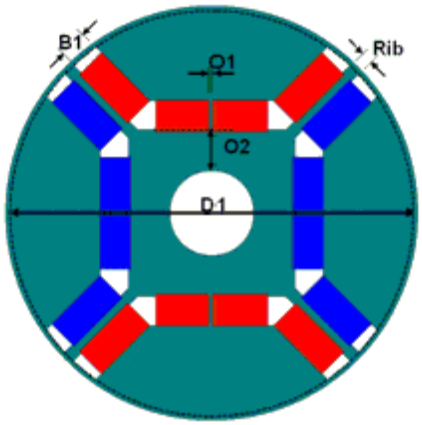


Pole Type 4

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Pole Type 5



Pole Type 6

Defining Data for the AXIAL_PM Core Type

The general procedure for defining a AXIAL_PM core type is as follows:

1. Double-click the **Machine** > **Rotor** entry in the project tree to define data for the rotor. In the **Rotor Properties** dialog box, enter the number of poles in the **Number of Poles** field (default is 2). There are no additional settings.
2. Double-click the **Core** entry in the project tree to define data for the rotor or stator core. The core data **Properties** dialog box contains the following fields:

Outer Diameter	Outer diameter of the core
Inner Diameter	Inner diameter of the core
Length	Length of the core
Stacking Factor	Effective magnetic length of the core; default is 0.95.
Steel Type	Steel type of the core; click the button to open the **Select Definition** window and select the desired material.
Skew Width	Skew width measured in degrees

3. Double-click the **Machine** > **Rotor** > **Core** > **Pole** entry in the project tree to open its **Properties** dialog box, which contains the following fields:

Embrace	Pole embrace value
Magnet	Magnet material type; click the button to open the **Select Definition**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type	window and select the magnet material type from the list. Use the **Material Filter** tab settings to filter for **Magnet** materials.
Magnet Length	Radial length of the magnet
Magnet Thickness	Axial thickness of the magnet, per side

Defining the Stator and Rotor Windings for a Generic Rotating Machine

> **Note:** This section is not applicable to
>
> - AXIAL_PM type stators or rotors.
> - PM_INTERIOR, AXIAL_CAGE, and SOLID type rotors.
> - SALNT_POLE type stators and rotors. Refer to Defining SALNT_POLE Stator and Rotor Field Winding Data for a Generic Rotating Machine for winding information for SALNT_POLE stators and rotors.
> - SALNT_STEP type stators.

To define the wires, conductors, insulation, and windings of a stator or rotor:

1. To open the rotor or stator slot winding **Properties** dialog box, double-click the **Machine** > **Stator** > **Winding** or **Machine** > **Rotor** > **Winding** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate dialog box.)
2. Click the **Winding** tab.
3. Choose the desired number of layers in the winding from the drop-down menu in the **Winding Layers** field.
4. Select a **Winding Type**:
    a. Click the button for **Winding Type**.

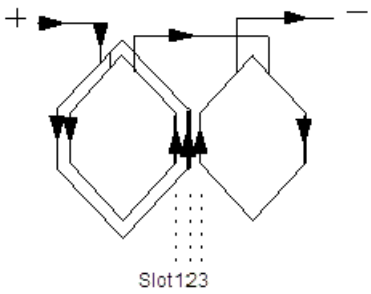
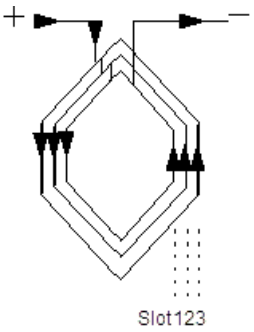
       The **Winding Type** window appears.

    b. Select from one of the following three types of winding:

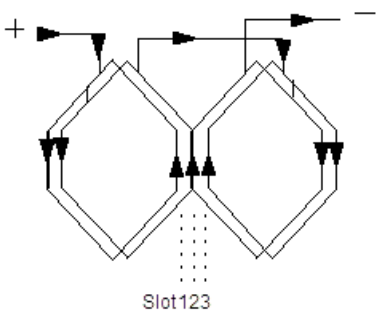
       - **Whole Coiled**
       - **Half Coiled**
       - **Editor**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you place the mouse cursor over a winding button, an outline of the selected winding appears. The following table describes the six types of windings that are possible (three for one-layer and three for two-layer):

Type	Description
Winding Editor (one-layer)	A user-defined one-layer winding arrangement. You need to set up the winding arrangement for each slot in the Winding Editor.
Whole-Coiled (one-layer)	A one-layer whole-coiled winding: 
Half-Coiled (one-layer)	A one-layer concentric half-coiled winding: 
Winding Editor (two-layer)	A user-defined two-layer winding arrangement. When you select for winding layers you can specify a different winding arrangement for each slot in the Winding Editor.
Whole-Coiled (two-layer)	A two-layer whole-coiled winding:

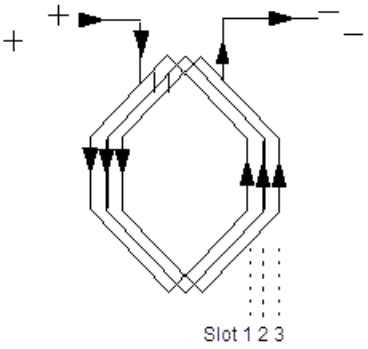
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The phase belt for this winding configuration is equal to 360/2*m*, where *m* is the phase number.

Half-Coiled (two-layer)

A two-layer half-coiled winding:



There is only one coil per phase per pair of poles.

> **Note:** For a two-layer winding, if you check Constant Pitch in the Winding Editor, only the top layer needs to be defined; the bottom layer is determined according to the coil pitch.

Once you have clicked a button to select a winding, click **OK** to close the **Winding Type** dialog box and return to the **Properties** window.

5. Enter the number of parallel branches in one phase of the winding in the **Parallel Branches** field.
6. Enter the total number of conductors in each slot in the **Conductors per Slot** field. This value is the number of turns per coil multiplied by the number of layers. Enter 0 to have RMxprt auto-design this value.
7. Enter the coil pitch, measured in number of slots, in the **Coil Pitch** field. The coil pitch is the number of slots separating one winding. For example, if a coil starts in slot 1 and ends

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in slot 6, it has a coil pitch of 5. This field is not displayed when the number of **Winding Layers** is 1.

8. Enter the number of wires per conductor in the **Number of Strands** field. Enter 0 to have RMxprt auto-design this value.
9. Enter the thickness of the double-sided wire wrap in the **Wire Wrap** field. Enter 0 to automatically obtain this value from the wire library.
10. Select the **Wire Size**:
    a. Click the button for **Wire Size**.

       The **Wire Size** dialog box appears.

    b. Select a value from the **Wire Diameter** drop-down menu.

    c. Select a wire gauge from the **Gauge** drop-down menu. You can select from the following options:

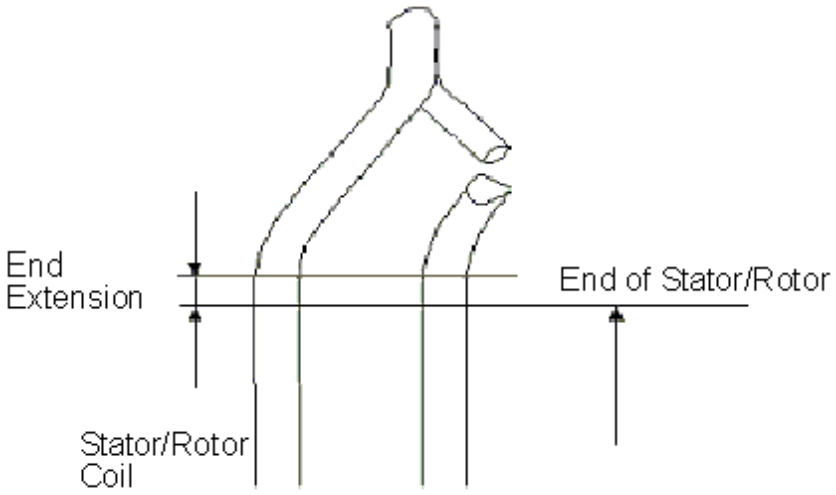
< ***number*** **>**	You can select a specific gauge number. When you select a gauge number, the **Wire Diameter** field is automatically updated.
USER	This option allows you to manually enter the **Wire Diameter**. This is useful when you want to enter a diameter that does not correspond to a particular wire gauge.
AUTO	This option sets the **Wire Diameter** to zero, and RMxprt automatically calculates the optimal value. The diameter information is then written to the output file when you analyze the design.
MIXED	This option allows you to define a conductor that is made of different size wires. For example, a single conductor may consist of five wires: three wires with a diameter of 0.21 mm and two wires with a diameter of 0.13 mm.

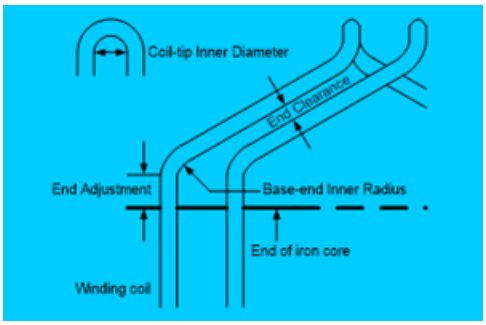
       The gauge number is based on AWG settings. You can create your own wire table using **Machine** > **Wire**, and then you can select this wire table using the **Tools** > **Options** > **Machine Options** option.

    When you are done setting the wire size, click **OK** to close the **Wire Size** dialog box and return to the **Properties** dialog box.
11. Click the button for **Conductor Type**. This opens the **Select Definition** dialog box in which you select the material type to be used for the winding.
12. Click the **End/Insulation** tab. Select or clear the **Input Half-turn Length** check box. Do one of the following:
    - If you selected **Input Half-turn Length**, then enter the half-turn length of the armature winding in the Half Turn Length field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you cleared **Input Half-turn Length**, then enter the end length adjustment of the stator/rotor coils in the **End Extension** field. The end extension is the distance one end of the conductor extends vertically beyond the end of the stator or rotor.





13. Enter the span length correction factor to scale the end span length in the **Correction Factor** field.
14. Enter the inner radius of the base corner in the **Base Inner Radius** field.
15. Enter the inner diameter of the coil tip in the **Tip Inner Diameter** field.
16. Enter the distance between two adjacent coils in the **End Clearance** field.
17. Enter the thickness of the single-side coil wrap insulation in the **Coil Wrap** field.

    This field is applicable and displayed only for slot types 5 and 6.
18. Enter the thickness of the slot liner insulation in the **Slot Liner** field.
19. Enter the thickness of the wedge insulation in the **Wedge Thickness** field.
20. Enter the thickness of the insulation layer in the **Layer Insulation** field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This field is applicable and displayed only when the **Winding Layers** value is 2.

21. Enter the bottom insulation thickness in the **Bottom Insulation** field.

    This field is applicable and displayed only for slot types 5 and 6.

22. Enter the limited slot fill factor for the wire design in the **Limited Fill Factor** field. The slot fill factor is the ratio between the cross-sectional area of all conductors in one slot and the entire slot area.

    This field is applicable and displayed only for slot types 1, 2, 3, and 4.

23. Enter the top spare slot space for a dual-winding machine in the **Top Spare Space** field.
24. Enter the bottom spare slot space for a dual-winding machine in the **Bottom Spare Space** field.
25. Click **OK** to close the **Properties** dialog box.

Stator and Rotor Winding Data for Generic Rotating Machines

> **Note:** This section is not applicable to
> - AXIAL_PM type stators or rotors
> - PM_INTERIOR, AXIAL_CAGE, and SOLID type rotors
> - SALNT_POLE type stators and rotors. Refer to *Defining SALNT_POLE Stator and Rotor Field Winding Data for a Generic Rotating Machine* for winding information for SALNT_POLE stators and rotors.

To access the core slot data, double-click either the **Machine** > **Rotor** > **Core** > **Slot** or the **Machine** > **Rotor** > **Core** > **Slot** entry in the project tree.

The winding data **Properties** dialog box contains the following fields:

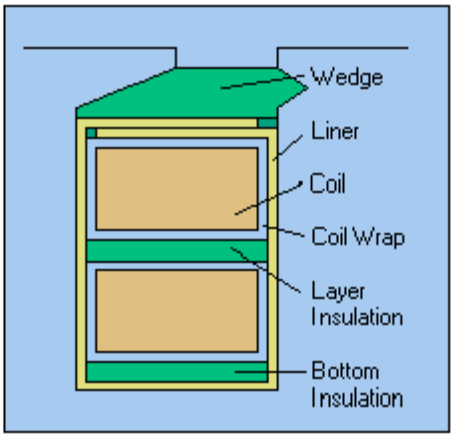
Winding tab	**Winding Layers**	The number of winding layers. (The bottom layer is for another side slot if the core is double-sided.) Select 1 or 2 in the drop-down menu. Default value is 2. **Winding Layers** is always 2 if the core is double-sided.
	Winding Type	The type of stator winding. Click the button to open the **Winding Type** window and choose from **Whole-Coiled**, **Half-Coiled**, and **Editor**. Default value is **Whole-Coiled**.
	Parallel Branches	The number of parallel branches in the winding; default value is 1.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Conductors per Slot	The number of conductors per slot (0 for auto-design)
	Coil Pitch	The coil pitch measured in number of slots. This field is displayed only when the number of **Winding Layers** is 2. **Coil Pitch** is always 0 if the core is double-sided.
	Number of Strands	The number of wires per conductor (0 for auto-design); default value is 1.
	Wire Wrap	The thickness of the double-sided wire wrap (0 to automatically obtain this value from the wire library).
	Wire Size	The diameter of the wire (0 for auto-design). Click the button to open the **Wire Size** dialog box where you can specify units, wire type, diameter, and gauge.
End/Insulation tab	**Input Half-turn Length**	Select or clear this check box to specify whether or not you want to enter the half-turn length. When this check box is selected, the **Half Turn Length** field appears the next time you open the **Properties** window. When this check box is selected, the **End Extension** field appears instead. Default value is unchecked.
	Half Turn Length	The average half-turn length of the armature winding. Visible only when **Input Half Turn Length** is checked.
	End Extension	The end length adjustment of the coils, which is the distance one end of the conductor extends vertically beyond the end of the stator or rotor. Invisible when **Input Half Turn Length** is checked.
	Correction Factor	End span length correction factor to scale the end span length. Must be > 0. Default value is 1.0. Invisible when **Input Half Turn Length** is checked
	Base Inner Radius	The inner radius of the base corner
	Tip Inner Diameter	The inner diameter of the coil tip. Invisible if the core is double-sided.
	End Clearance	The end clearance between two adjacent coils
	Coil Wrap	The thickness of the single-side coil wrap. This field is displayed only when the **Slot Type** is 5 or 6.
	Slot Liner	The thickness of the slot liner insulation
	Wedge Thickness	The thickness of the wedge insulation
	Layer Insulation	The thickness of the insulation layer. Invisible when number of winding layers is 1 or the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	core is double-sided.
Bottom Insulation	Thickness of the bottom insulation. This field is displayed only when the **Slot Type** is 5 or 6.
Limited Fill Factor	The limited slot fill factor for the wire design. The slot fill factor is the ratio between the cross-sectional area of all conductors in one slot and the entire slot area. This field is displayed only for Slot Types 1, 2, 3, or 4.
Top Spare Space	The top spare slot space for a dual-winding machine. The value must be greater-than or equal-to 0 and less-than 1.
Bottom Spare Space	The bottom spare slot space for a dual-winding machine. The value must be greater-than or equal-to 0. Also, the sum of the **Top Spare Space** and **Bottom Spare Space** values must be less than 1.
Conductor Type	Conductor material type of the Stator/Rotor Winding



Winding Editor for a Generic Rotating Machine

For a generic rotating machine, you may want to specify a different number of conductors for each stator or rotor slot. The **Winding Editor** enables you to specify the number of turns for each coil. To enable the **Winding Editor**, you must have set the **Winding Property** for the **Winding Type** to **Editor**.

To specify the number of turns for each coil:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Machine** > **Winding** > **Edit Layout**.

   The **Winding Editor** dialog box appears.

2. In the table in the upper left, set which **Phase** you want for each coil and which slot is the “In” and “Out” slot for the current in each coil.
3. If you are working on a quarter or half model, you may want to specify a multiplier by selecting a value from the **Periodic Multiplier** drop-down menu.
4. Select or deselect the **Constant Turns** or **Constant Pitch** check boxes, depending on whether you want to be able to change these setting in the table above. When these options are selected, you cannot change the turns or pitch.
5. When you are satisfied with the coil settings, click **OK** to close the **Winding Editor** dialog box.

Defining Different Size Wires for a Generic Rotating Machine

Use the **Gauge** option if you have a conductor that is made up different size wires.

To define different size wires:

1. In the **Wire Size** window, select **MIXED** from the **Gauge** drop-down menu.
2. Select either **Round** or **Rectangular** as the **Wire Type**.
3. Enter the appropriate wire data in the table:
   - For a round wire:
     - Click **Add** to add the new wire data.
     - Enter the **Diameter** in the table.
     - Enter a **Number** in the table to specify how many of the conductor’s wires have this diameter.
   - For a rectangular wire:
     - Click **Add** to add the new wire data.
     - Enter the **Width** of the wire in the table. The width should be greater than the thickness.
     - Enter the **Thickness** of the wire in the table.
     - Enter the **Fillet** value in the table.
     - Enter a **Number** in the table to specify how many of the conductor’s wires have this data.
4. Repeat step 3 for each size wire you want to add.
5. When you are finished defining the wires, click **OK** to close the **Wire Size** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** For example, if one conductor is made up of five wires, and three of those wires have a diameter of 0.21 mm, and the other 2 have a diameter of 0.13 mm, then the mixed wire size table will have two lines. The first line will list Diameter = 0.21 and Number = 3. The second line will list Diameter = 0.13 and Number = 2. An equivalent wire diameter is displayed as Wire Size value in the Winding tab in the Properties window.

Defining the Stator and Rotor Circuits for a Generic Rotating Machine

You can define stator and rotor Circuit parameters for generic rotating machines whose Source Type is DC.

> **Note:**
>
> This procedure is not applicable to
>
> - AXIAL_PM type stators or rotors
> - SALNT_POLE type stators or rotors
> - AXIAL_AC, AXIAL_CAGE, SLOT_CAGE, and SOLID type rotors
> - PM_INTERIOR type rotors

To define the circuit parameters of a stator or rotor:

1. To define the circuit parameters click either the stator or rotor Circuit icon in the project tree. You can then edit the brush parameters directly in the Properties window. (You can also open the rotor or stator circuit **Properties** dialog box by double-clicking the **Machine** > **Stator** > **Circuit** or **Machine** > **Rotor** > **Circuit** entry in the project tree on the desktop.)
2. Select the **Control Type** from the drop down menu. Choices are: DC, CCC, PWM, and HCC.
3. If the stator or rotor **Position Control** has been enabled, set the **Lead Angle of Trigger** value.
4. If the **Control Type** chosen is either DC or CCC, set the **Trigger Pulse Width**. The default value is 120.
5. Set the **Transistor Drop** (voltage drop of one transistor).
6. Set the **Diode Drop**.
7. If the **Control Type** is CCC, set the **Maximum Current** and **Minimum Current** values.
8. If the **Control Type** is PWM, set the **Modulation Index** and **Carrier Frequency Times**.
9. If the **Control Type** is HCC, set the **Reference Amplitude** and **Hysteresis Band**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stator and Rotor Circuit Data for Generic Rotating Machines

To access the stator or rotor circuit data, double-click either the **Machine** > **Stator** > **Circuit** or the **Machine** > **Rotor** > **Circuit** entry in the project tree.

> **Note:** This section is not applicable to PM_INTERIOR and SALNT_STEP rotors.

The circuit data **Properties** dialog box contains the following fields:

Control Type	Control Type: DC, CCC (chopping current control), PWM, HCC (hysteresis current control). Default is DC. **Note:** For SALNT_STEP Stators, only DC and CCC control types are available.
Lead Angle of Trigger	Lead angle of trigger in electrical degrees. Visible only when the machine stator or rotor **Position Control** is enabled.
Trigger Pulse Width	Trigger pulse width in electrical degrees. Visible only when **Control Type** is DC or CCC. Default value is 120.
Transistor Drop	Voltage drop of one transistor
Diode Drop	Voltage drop of one diode, or the total voltage for star-type circuits in the discharge loop
Maximum Current	Maximum current for chopping current control. Visible only when **Control Type** is CCC.
Minimum Current	Minimum current for chopping current control. Visible only when **Control Type** is CCC.
Modulation Index	Modulation index (the ratio of the sine-wave amplitude to the triangular amplitude). Visible only when **Control Type** is PWM.
Carrier Frequency Times	Carrier frequency times (the ratio of the triangular frequency to the sine-wave frequency). Visible only when **Control Type** is PWM.
Reference Amplitude	The amplitude of the sine-wave reference current. Visible only when **Control Type** is HCC.
Hysteresis Band	The difference between the upper and lower hysteresis limits. Visible only when **Control Type** is HCC.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining the Axial AC Rotor Brush for a Generic Rotating Machine

Optionally, you can insert or remove brush data for generic rotating machines that have an Axial AC Rotor Structure. If you have inserted a brush, the icon appears under the core slot in the project tree.

To insert a brush:

1. Right-click on the rotor core icon to display the pop-up menu.
2. Click **Insert Brush**.
3. To edit the brush data, double-click the brush icon to open the brush properties dialog box.

The brush data **Properties** dialog box contains the following fields:

Diameter	Brush surface diameter
Brush Width	Brush width
Brush Length	Brush length
Brush Drop	Voltage drop of a brush
Contact Resistance	Contact resistance of a brush
Brush Press	Brush pressure per unit area
Frictional Coefficient	Frictional coefficient of a brush

To remove an existing brush:

1. Right-click on the rotor icon to display the drop-down menu.
2. Click **Remove Brush**.

Vent Data for Generic Rotating Machines

Optionally, you can insert or remove Vent data for generic rotating machines that have either an Inner Rotor or Outer Rotor Structure. If you have inserted a Vent, the icon appears under the core slot in the project tree.

> **Note:** This section does not apply to the SALNT_POLE, NONS_RELU, or SOLID rotor types.

To insert a vent:

1. Right-click on the stator or rotor core icon to display the pop-up menu.
2. Click **Insert Vent**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To remove an existing vent:

1. Right-click on the stator or rotor icon to display the pop-up menu.
2. Click **Remove Vent**.

The vent data **Properties** dialog box contains the following fields.

Vent Ducts	Number of radial vent ducts; default is 0.
Duct Width	Width of radial vent ducts
Magnetic Spacer Width	Width of magnetic spacer which hold vent ducts. 0 for non-magnetic spacer.
Duct Pitch	Vent ducts
Holes per Row	Number of axial vent holes per row. Not available for SLOT_CAGE rotor type.
Inner Hole Diameter	Diameter of vent holes in inner row. Not available for SLOT_CAGE rotor type.
Outer Hole Diameter	Diameter of vent holes in outer row. Not available for SLOT_CAGE rotor type.
Inner Hole Location	Center-to-center diameter of inner row hole vents. Not available for SLOT_CAGE rotor type.
Outer Hole Location	Center-to-center diameter of outer row hole vents. Not available for SLOT_CAGE rotor type.

Defining the Shaft Data for a Generic Rotating Machine

To define the shaft:

1. To open the shaft data **Properties** dialog box, double-click the **Machine** > **Shaft** entry in the project tree on the desktop. (You can also enter values in the **Properties** section of the desktop without opening a separate window.)
2. Select or clear the **Magnetic Shaft** check box to specify whether or not the shaft is to be made of magnetic material.
3. Enter the frictional loss in the **Frictional Loss** field.
4. Enter the windage loss (or power for wind power generators) measured at the **Reference Speed** in the **Windage Loss or Power** field.
5. Enter the reference speed at which frictional and windage losses are measured in the **Reference Speed** field.
6. Click **OK** to close the **Properties** window.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Access Shaft Data for General DC Machines

To access the shaft data, double-click the **Machine** > **Shaft** entry in the project tree.

The **Shaft Data Properties** window contains the following fields:

Magnetic Shaft	Select or clear this check box to indicate whether or not the shaft is made of magnetic material. When selected, the shaft is magnetic. Default is unchecked for PM_INTERIOR rotor type.
Frictional Loss	The frictional loss measured at the **Reference Speed**
Windage Loss or Power	The **Windage Loss** (or **Power** for wind power generators) measured at the **Reference Speed**.
Reference Speed	The speed at which the friction and windage losses are measured. Default is 3600 rpm for PM_INTERIOR and AXIAL_CAGE rotor types.

Setting Up Analysis Parameters for a Generic Rotating Machine

To define solution parameters for a generic rotating machine:

1. Right-click **Analysis** in the project tree, and click **Add Solution Setup**.

   The **Solution Setup** dialog box appears.

2. Click the **General** tab.

   a. If you wish to change the automatically assigned setup name, enter a name for the setup in the **Setup Name** field.

   b. The solution setup is enabled by default. Uncheck the **Enabled** box to disable the setup, if desired.

   c. Select an operation type (**Motor**, **Generator**, or **Wind Generator**) from the **Operation Type** drop-down menu. The **Operation Type** is set to **Motor** by default.

   d. Select the **Load Type** used in the machine.

   - If the **Operation Type** is **Motor**, select one of the following **Load Type** options:

Const Speed	The speed remains constant in the motor.
Const Power	The output power remains constant in the motor. (Default type.)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Const Torque	The torque remains constant regardless of the speed. In this case, $T_{load} = T_{rated}$, given by the output power divided by the given rated speed.
Linear Torque	The torque increases linearly with speed. In this case, $T_{load} = T_{rated} * (n/n_{rated})$ where T_{rated} is given by the output power divided by the given rated speed.
Fan Load	The load varies nonlinearly with speed. In this case, $T_{load} = T_{rated} * (n/n_{rated})^2$ where T_{rated} is given by the output power divided by the given rated speed.

- If the **Operation Type** is either **Generator** or **Wind Generator**, select one of the following **Load Type** options:

Infinite Bus	When the generator is connected to an infinite bus of the power system, the terminal voltages remain constant and the output power is determined by the output currents.
Independent Generator	The generator supplies electrical power to a load inductor. For an AC generator, the resistance and inductance of the load inductor are determined by the rated output power, the rated voltage, and the rated power factor. For a DC generator, the resistance of the load resistor is determined by the rated output power and the rated voltage.

e. Enter the rated mechanical or electrical output (apparent) power in the **Rated Output Power** field.

When the **Operation Type** is **Generator** and **Source Type** is **AC**, the property name changes to **Rated Apparent Power (kVA)**, and the unit column is blank.

f. Enter the applied or output rated voltage in the **Rated Voltage** field. (RMS value for AC source.) Default value is 208 V.

g. Enter the temperature at which the system functions in the **Operating Temperature** field. Default is 75 cel.

3. Click the **Generic Rotating Machine (GRM)** tab.

a. For AC source type machines, enter the rated power factor in the **Rated Power Factor** field. The default value is 0.8. This field is invisible when the **Source Type** is **DC**.

b. For AC source type machines, if you wish to determine load impedance when the phase current leads the phase voltage, enable the **Capacitive Power Factor** check box. This field is invisible when the **Source Type** is **DC**.

c. Enter the source frequency in the **Frequency** field and select the appropriate unit of measure. The default value is 60 Hz. This field is invisible when the **Source Type** is **DC**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. Optionally, you can click the **Use Defaults** button to restore the tab settings to default values.

4. Optionally, click the **Defaults** tab. This tab contains two buttons:
   - Click the **Save Defaults** button to save the currently-defined settings as defaults for future setups.
   - Click the **Revert to Standard Defaults** button to clear existing user-defined defaults and revert to the standard settings.
5. Click **OK** to close the **Solution Setup** window.

Solution Data for Generic Rotating Machines

To access the solution data, double-click the solution setup located under **Analysis** in the project tree to open the solution **Properties** dialog box. Solution data is also accessible in the desktop **Properties** window for the selected setup.

The solution setup Properties contains the following fields:

General tab:

Name	The name of the setup. Not editable.
Enabled	Check box that enables/disables the solution setup. Default is checked (enabled)
Operation Type	Set the type of operation for the machine analysis. drop-down menu selections are: Motor, Generator, Wind Generator. Default is Motor.
Load Type	Select the mechanical or electrical load type from the drop-down menu. For Motor **Operation Type** the selections are: Const Speed, Const Power, Const Torque, Linear Torque, Fan Load. Default is Const Power. For Generator and Wind Generator **Operation Type** the selections are: Infinite Bus and Independent Generator. Default is Infinite Bus.
Rated Output Power	Enter the rated mechanical or electrical output (apparent) power, and select the unit.
Rated Voltage	Enter a value for the rated voltage and select the unit.
Rated Speed	Type a value for the rated speed, and select the unit.
Operating Temperature	Type a value for the operating temperature, and select the unit.

GRM tab:

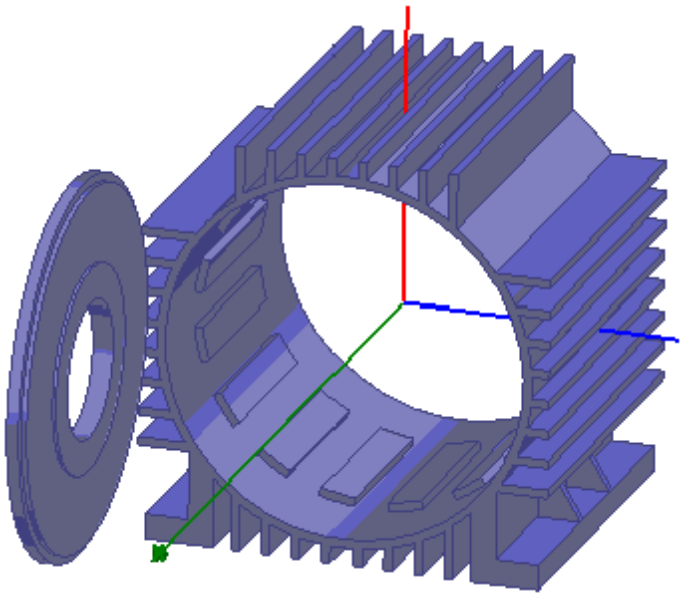
Rated Power Factor	Enter the rated power factor for AC Source Type. This field is displayed only for AC Source Type. Default value is 0.8.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Capacitive Power Factor	Check box that enables/disables use of a capacitive power factor. Used together with the **Rated Power Factor** when the phase current leads the phase voltage, to determine load impedance. Default is unchecked.
Frequency	Enter a value for the frequency, and select the unit.

Adding Machine Housings

You can add machine housing parts to all machine types. Note that machine housings for Generic Rotating Machines and Single-Phase Induction Motors are available only for Inner Rotor type machines.



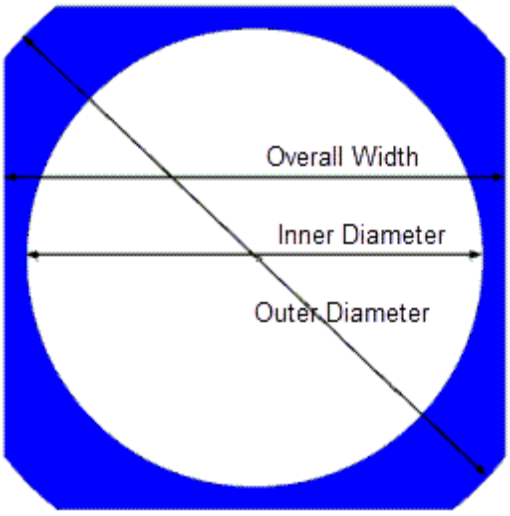
1. To include a Housing in the design, right-click the Machine icon in the Project Manager tree and select **Insert Housing**. A new Housing icon is added to the machine design containing Front Cap and Rear Cap housing elements.
2. Right-click on Housing in the Project Manager tree to include and specify parameter data for the following housing elements:
   - Duct
   - Fin
   - Flange
   - Foot
3. Specify properties for the Housing and related elements by selecting them and setting parameter data values either in the Properties window or Properties dialog.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. You can remove the Housing and all related elements by right-clicking Machine and selecting **Remove Housing**. Individual housing elements can be removed by right-clicking Housing and selecting the desired **Remove** *<objectname>* menu item.

> **Note:** The Front Cap and Rear Cap elements are always included with the housing and cannot be removed individually.

Housing Data



You can specify the following Housing properties:

Housing

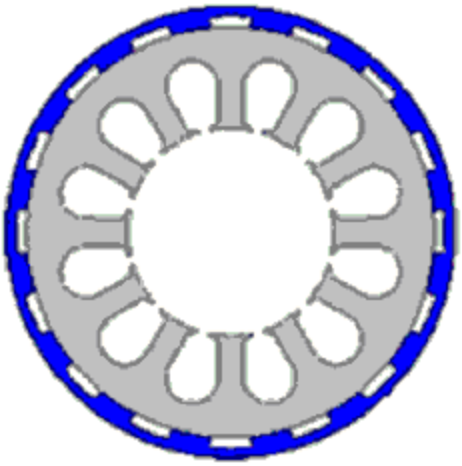
	Name	Value	Unit	Evaluated Value	Description
	Inner Diameter	120	mm		Housing inner diameter
	Overall Width	150	mm		Housing overall width of the squar...
	Outer Diameter	140	mm		Housing outer diameter of the circl...
	Housing Length	160	mm		Length of the housing

Name	Description	Notes
Inner Diameter	Housing inner diameter	Inner Diameter > 0
Outer	Housing outer diameter of the circle	Outer Diameter > Inner

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Diameter	profile	Diameter
Overall Width	Housing overall width of the square profile	Overall Width > Inner Diameter
Housing Length	Length of the housing	Housing Length > 0

Housing Duct Data



You can specify the following housing Duct properties:

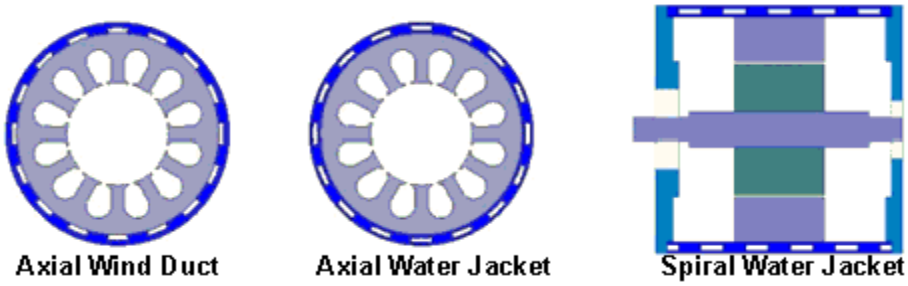
Duct

	Name	Value	Unit	Evaluated Value	Description
	Number of Ducts	16			Number of ducts, or water jack
	Duct Type	Axial Wind Duct			Duct type
	Duct Height	5	mm		Duct height in radial direction
	Duct Embrace	0.5			Ratio of duct span, or duct widt...
	Duct Length	65	mm		Axial length for axial wind duct o...

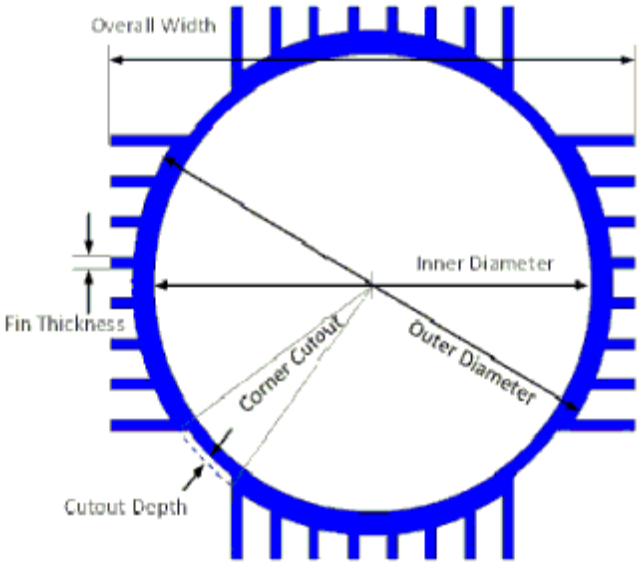
Name	Description	Notes
Number of Ducts	Number of ducts, or water jackets	Number of Ducts > 0
Duct Type	Duct type	Available types: • 1-Axial Wind Duct

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		• 2-Axial Water Jacket • 3-Spiral Water Jacket Default: Axial Wind Duct.
Duct Height	Duct height in radial direction	0 < Duct Height < Housing Thickness (half of the difference between the housing and stator outer diameters)
Duct Embrace	Ratio of duct span, or duct width, to duct pitch with uniform distribution	0 < Duct Embrace < 1
Duct Length	Axial length for axial wind duct only	0 < Duct Length < Housing Length

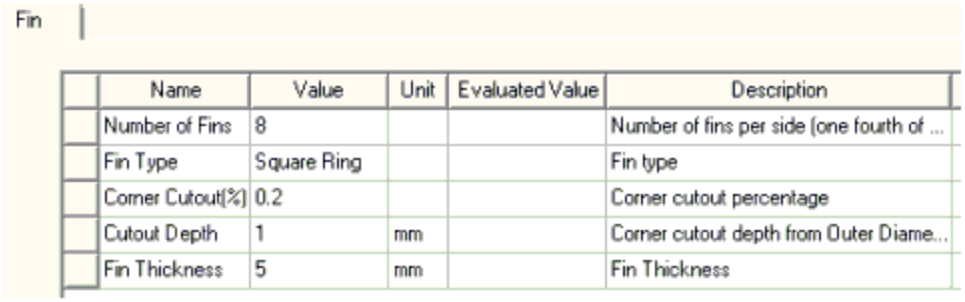


Housing Fin Data



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can specify the following housing Fin properties:

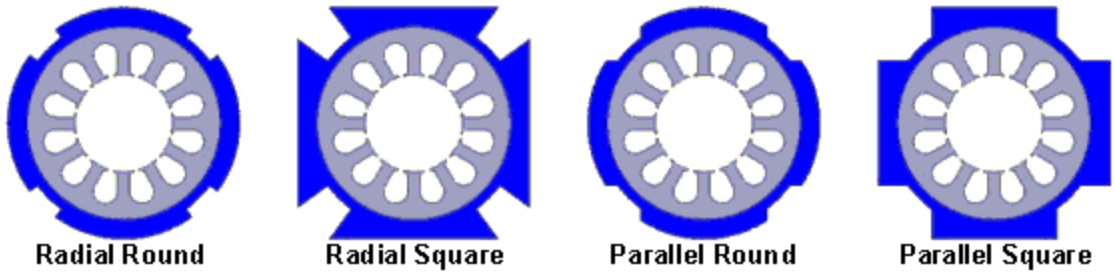

Fin

	Name	Value	Unit	Evaluated Value	Description
	Number of Fins	8			Number of fins per side (one fourth of ...
	Fin Type	Square Ring			Fin type
	Corner Cutout(%)	0.2			Corner cutout percentage
	Cutout Depth	1	mm		Corner cutout depth from Outer Diame...
	Fin Thickness	5	mm		Fin Thickness

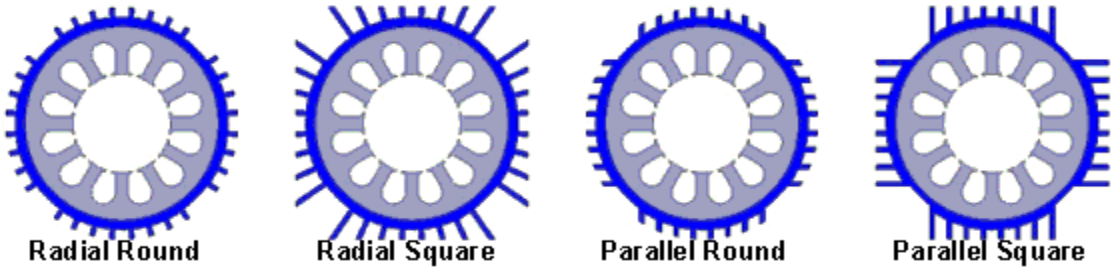
Name	Description	Notes
Number of Fins	Number of fins per side (one fourth of total fins)	Number of Fins > 0
Fin Type	Fin type	Available types: • 1-Radial Round • 2-Radial Square • 3-Parallel Round • 4-Parallel Square • 5-Square Ring • 6-Servo Ring. Default: Radial Round (Types 5 & 6 are disabled if Number of Fins = 1).
Corner Cutout (%)	Corner cutout percentage	0 ≤ Corner Cutout < 1
Cutout Depth	Corner cutout depth from Outer Diameter, or Fin Extension for Fin Type 5	Cutout Depth ³ 0
Fin Thickness	Fin Thickness	Fin Thickness > 0

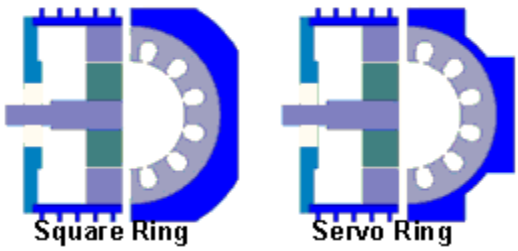
Fin Types when Number of fins = 1:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



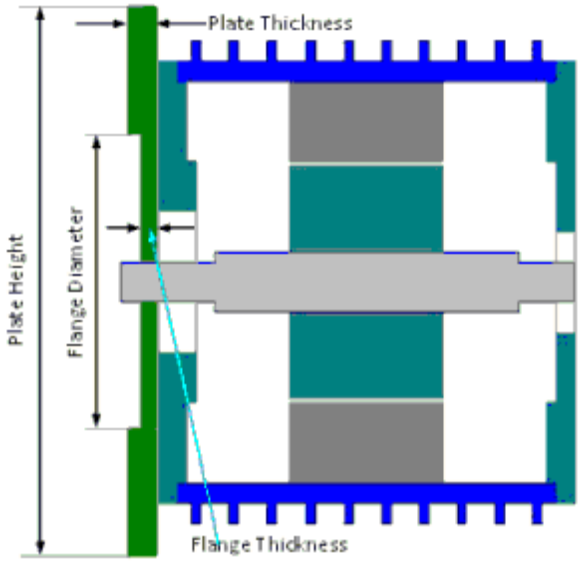
Fin types when number of fins = 2 or greater:





Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Housing Flange Data



You can specify the following housing Flange properties:

Flange

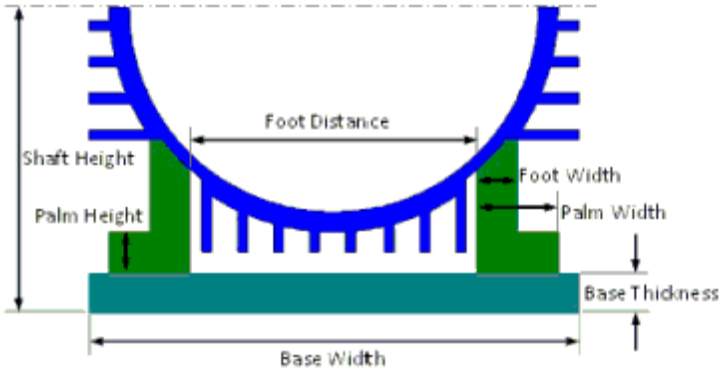
	Name	Value	Unit	Evaluated Value	Description
	Flange Diameter	100	mm		Diameter of the flange section that cover...
	Flange Thickn...	10	mm		Thickness of the flange section that cov...
	Plate Height	180	mm		Height of the cooling plate which the flan...
	Plate Thickness	20	mm		Thickness of cooling plate
	Plate Width	180	mm		Width of cooling plate

Name	Description	Notes
Flange Diameter	Diameter of the flange section that covers the front bearing	Flange Diameter > 0
Flange Thickness	Thickness of the flange section that covers the front bearing	Flange Thickness > 0
Plate Height	Height of the cooling plate which the flange mounts on	Plate Height > Flange Diameter
Plate Thickness	Thickness of cooling plate	Plate Thickness > Flange Thickness

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plate Width	Width of cooling plate	Plate Width > 0

Housing Foot Data



You can specify the following housing Foot properties:

Foot

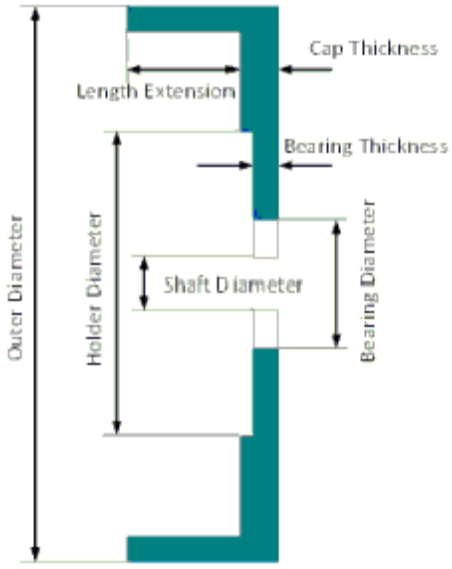
	Name	Value	Unit	Evaluated Value	Description
	Base Length	160	mm		Base plate length in axial direction
	Base Width	150	mm		Base plate width
	Base Thickness	20	mm		Base plate thickness
	Shaft Height	80	mm		Height of shaft from foot
	Foot Length	160	mm		Foot length in axial direction
	Foot Width	10	mm		Foot width at junction with housing
	Foot Distance	90	mm		Distance between two feet
	Palm Height	10	mm		Palm height
	Palm Width	30	mm		Palm width at junction with base plate

Name	Description	Notes
Base Length	Base plate length in axial direction	Base Length > 0
Base Width	Base plate width	Base Width > 0
Base Thickness	Base plate thickness	Base Thickness > 0
Shaft Height	Height of shaft from foot	Shaft Height > Base Thickness

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

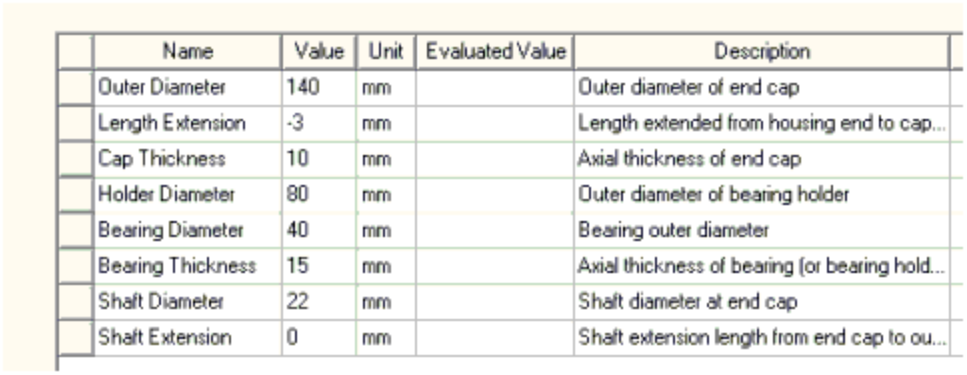
Foot Length	Foot length in axial direction	0 < Foot Length < Base Length
Foot Width	Foot width at junction with housing	Foot Width > 0
Foot Distance	Distance between two feet	0 < Foot Distance < Base Width
Palm Height	Palm height	0 < Palm Height < (Shaft Height - Base Thickness)
Palm Width	Palm width at junction with base plate	Palm Width ≤ (Base Width - Foot Distance)/2 Palm Width ³ Foot Width

Housing Front Cap and Rear Cap Data



You can specify the following housing Front Cap and Rear Cap properties:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



	Name	Value	Unit	Evaluated Value	Description
	Outer Diameter	140	mm		Outer diameter of end cap
	Length Extension	-3	mm		Length extended from housing end to cap...
	Cap Thickness	10	mm		Axial thickness of end cap
	Holder Diameter	80	mm		Outer diameter of bearing holder
	Bearing Diameter	40	mm		Bearing outer diameter
	Bearing Thickness	15	mm		Axial thickness of bearing (or bearing hold...
	Shaft Diameter	22	mm		Shaft diameter at end cap
	Shaft Extension	0	mm		Shaft extension length from end cap to ou...

Name	Description	Notes
Outer Diameter	Outer diameter of end cap	Cap Outer Diameter ³ Housing Outer Diameter
Length Extension	Length extended from housing end to cap inner surface, < 0 for rabbet length	
Cap Thickness	Axial thickness of end cap	Cap Thickness > 0
Holder Diameter	Outer diameter of bearing holder	Holder Diameter > 0
Bearing Diameter	Bearing outer diameter	Bearing Diameter < Holder Diameter
Bearing Thickness	Axial thickness of bearing (or bearing holder)	Bearing Thickness > 0
Shaft Diameter	Shaft diameter at end cap	Shaft Diameter < Bearing Diameter
Shaft Extension	Shaft extension length from end cap to outside for torque output or fan installation	Shaft Extension ³ 0

Stator Vent Data

Select a Machine Type to get more information of Stator Vents:

- Three-Phase Induction Motors
- Three-Phase Synchronous Machines
- Three-Phase Non-Salient Synchronous Machines

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rotor Vent Data

Select a Machine Type to get more information of Rotor Vents:

- Three-Phase Induction Motors
- Three-Phase Non-Salient Synchronous Machines

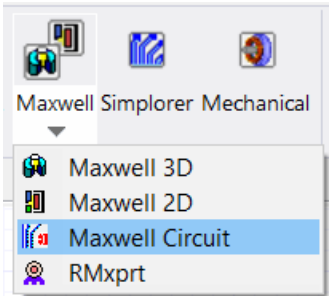
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

27 - Using the Maxwell Circuit Editor

To open Maxwell Circuit Editor:

1. Click **Start** > **Ansys EM Suite**.
2. In the Ansys Electronics Desktopinterface, make sure the **Desktop** tab is active.
3. From the Maxwell icon's drop-down list, select **Maxwell Circuit**.



The following menus are available in Maxwell Circuit Editor:

File menu
Edit menu
View menu
Project menu
Draw menu
Schematic menu
Maxwell Circuit menu
Tools menu
Window menu
Help menu

Related Topics:

Schematic Editor

The Component Libraries Window

Copying and Pasting Properties

Maxwell Circuit Editor Component Models

Placing Components in the Maxwell Circuit Editor Schematic

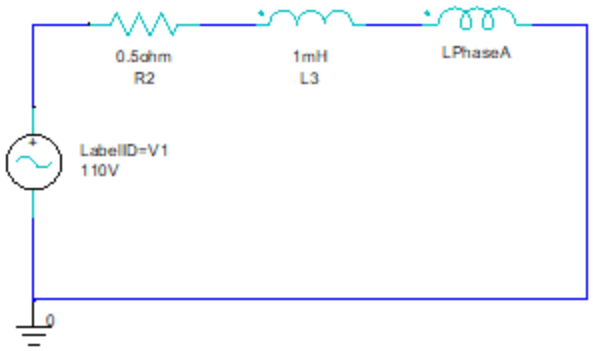
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning Component Properties in Maxwell Circuit Editor

Maxwell-Circuit Coupling

Circuit Voltage Equations

As a simple example, a single-phase winding excited by a voltage source is shown below.



The voltage equation of the circuit is

$$v(t) = R \bullet i(t) + L\frac{di(t)}{dt} + e\left(t\right)$$

where *R* and *L* are the winding resistance and end leakage inductance (not included in 2D FEA) of the winding, respectively, and *e(t)* is the induced voltage of the winding obtained from transient FEA. Integrate both sides of the voltage equation over a time step $[t_k, t_{k+1}]$, we get

$$\overline{v}(t_k, t_{k+1}) = R \bullet \overline{i}(t_k, t_{k+1}) + L\frac{i(t_{k+1}) - i(t_k)}{t_{k+1} - t_k} + \frac{\lambda(t_{k+1}) - \lambda(t_k)}{t_{k+1} - t_k}$$

where λ is the winding flux linkage from field analysis, $\overline{v}(t_k, t_{k+1})$ is the average source voltage, and $\overline{i}(t_k, t_{k+1})$ is the average winding current over the time step, which can be very accurately obtained from:

$$\begin{cases} \overline{v}(t_k, t_{k+1}) = \frac{1}{6}v(t_k) + \frac{2}{3}v(t_{k+0.5}) + \frac{1}{6}v(t_{k+1}) \\ \overline{i}(t_k, t_{k+1}) = \frac{1}{2}i(t_k) + \frac{1}{2}i(t_{k+1}) \end{cases}$$

If $\overline{v}(t_k, t_{k+1})$ and $\overline{i}(t_k, t_{k+1})$ are replaced by $v(t_{k+1})$ and $i(t_{k+1})$, respectively, the discrete method is called the *Backward Euler* integration method. With the *Backward Euler* integration method, the discrete format of the voltage equation is:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$v\left(t_{k+1}\right)=R\bullet i\left(t_{k+1}\right)+L\frac{i(t_{k+1})-i(t_k)}{t_{k+1}-t_k}+\frac{\lambda(t_{k+1})-\lambda(t_k)}{t_{k+1}-t_k}$$

If $\overline{v}\left(t_k,t_{k+1}\right)$ and $\overline{i}\left(t_k,t_{k+1}\right)$ are replaced by $v\left(t_{k+0.5}\right)$ and $i\left(t_{k+1}\right)$, respectively, the discrete format of the voltage equation will be:

$$v\left(t_{k+0.5}\right)=R\bullet i\left(t_{k+1}\right)+L\frac{i(t_{k+1})-i(t_k)}{t_{k+1}-t_k}+\frac{\lambda(t_{k+1})-\lambda(t_k)}{t_{k+1}-t_k}$$

Discrete Formula for Capacitors

The capacitor current can be derived from the capacitor voltage as

$$i_c\left(t\right)=C\frac{dv_c(t)}{dt}$$

or capacitor voltage is integrated from the capacitor current as

$$v_c\left(t\right)=v_c\left(t_k\right)+\frac{1}{C}\int_{t_k}^{t}i_c\left(t\right)dt$$

For Backward Euler formula, $i_c\,(t)$ is replaced by $i_c(t_{k+1})$, and the end-point voltage will be

$$v_c\left(t_{k+1}\right)=v_c\left(t_k\right)+r_ci_c\left(t_{k+1}\right)$$

where

$$r_c=\left(t_{k+1}-t_k\right)/C$$

If the capacitor current $i_c\,(t)$ is assumed to be change linearly from $i_c\,(t_k)$ to $i_c(t_{k+1})$, the end-point voltage will be

$$v_c\left(t_{k+1}\right)=v_c\left(t_k\right)+r_c\frac{i_c(t_k)+i_c(t_{k+1})}{2}$$

To balance the mid-point source and winding induced voltages, the middle-point capacitor voltage can be used in the circuit voltage equations. The middle-point capacitor voltage is

$$v_c\left(t_{k+0.5}\right)=v_c\left(t_k\right)+\frac{r_c}{2}\frac{3i_c(t_k)+i_c(t_{k+1})}{4}$$

In Maxwell 2D/3D transient solvers, the middle-point voltages $v(t_{k+0.5})$ have been employed for the voltage-excited windings for many releases, but the end-point voltages $v(t_{k+1})$ were still used for the circuit-excited windings until Release 2023 R1. Beginning with Release 2023 R2, the middle-point voltages are also used for the circuit-excited windings if the circuit does not include voltage/current controlled switches.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Error Analysis

The error of the current result depends on the total voltage error of the discrete voltage equation. The total voltage error of the end-point voltage formula is:

$$\varepsilon_{end} = [\overline{v}(t_k, t_{k+1}) - R \bullet \overline{i}(t_k, t_{k+1})] - [v(t_{k+1}) - R \bullet i(t_{k+1})]$$

$$= \frac{1}{6}[v(t_k) + 4v(t_{k+0.5}) - 5v(t_{k+1})] - \frac{1}{2}R \bullet [i(t_k) - i(t_{k+1})]$$

The total voltage error of the middle-point voltage formula is:

$$\varepsilon_{mid} = \frac{1}{6}[v(t_k) - 2v(t_{k+0.5}) + v(t_{k+1})] - \frac{1}{2}R \bullet [i(t_k) - i(t_{k+1})]$$

In an electro-magnetic device, the winding resistance voltage drop is usually negligible. When the time step is small enough so that the curve within the time step can be estimated by a tangent line,

$v(t_{k+0.5})\frac{1}{2}[v(t_k) + v(t_{k+1})]$, therefore:

$$\begin{cases} \varepsilon_{end} \approx \frac{1}{2}[v(t_k) - v(t_{k+1})] \\ \varepsilon_{mid} \approx 0 \end{cases}$$

which means $\varepsilon_{mid} < \varepsilon_{end}$ if the source voltage is not a DC type. Therefore, current results based on the middle-point voltages have higher accuracy than those based on the end-point voltage for AC voltage excitations. For DC voltage excitation, both results have the same accuracy.

For an electrical motor, the input power derived from the average value of *v(t)i(t)* of all phases will balance the output power plus all losses very well if the winding currents are derived based on the middle-point voltage formula. However, the electromagnetic power derived from the average value *e(t)i(t)* will not balance the output power plus the rotor losses because the output waveforms of *e(t)* and *i(t)* have an angle shift of a half time step.

In an external circuit, if you measure a voltage across a winding by a voltmeter, the measured voltage waveform will not be the same as the induced voltage waveform of the winding because both waveforms have an angle shift of a half time step, which is illustrated below.

In field analysis, the induced voltage of a winding derived from $[\lambda(t_{k+1}) - \lambda(t_k)] / [t_{k+1} - t_k]$ is the average value over each time step, which can be seen as the middle-point value. In circuit analysis, windings are treated as co-tree components. The voltage of a tree component is derived from its current and its parameters, while the voltage of a co-tree component is derived from the total voltages of all other components in the loop related to this co-tree component. For example, in the example circuit shown at the top, the measured voltage across winding **LPhaseA** is derived from the total voltages of components **V1**, **R2** and **L3**. Therefore, the measured winding voltage represents the end-point value for each time step.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In summary, from measured winding voltage waveforms, instead of winding induced voltage waveforms, we can validate Kirchhoff's Voltage Law in all winding related loops.

Schematic Editor

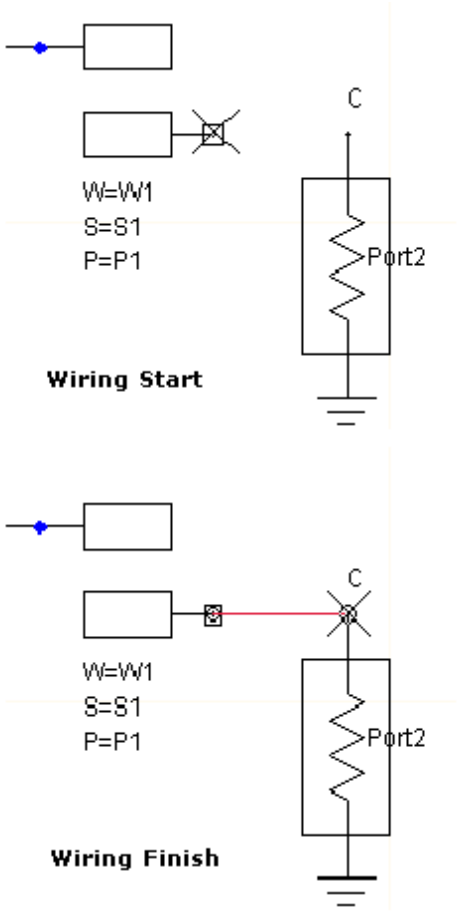
The Schematic editor is the tool for creating circuit schematics, or designs for a transient solution type. A design graphically represents and captures the electrical structure and characteristics of a circuit. You create such a design by starting the schematic editor and placing components, ports, connectors, and wires into a default empty schematic.

The Schematic Editor Window

The **Schematic Editor** window allows you to place components and wire them together. You can move components by simply selecting and dragging them. **Copy** and **Paste** can be used on components and their wires within the schematic editor. You can also copy and paste to other schematics.

As you place the cursor near a pin of a component, it changes from an arrow to an **X**. This indicates that the schematic editor is in the wiring mode. In the wiring mode, click to start drawing a wire. Click again to end the wire.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Commonly used items such as ports, grounds, and page connectors can be placed in the schematic by clicking their toolbar icons or by using the **Draw** menu.

Controls to **Zoom In**, **Zoom Out**, and **Fit Drawing** to the editor window are available on the **View** menu, the **Toolbar**, and on the shortcut menu that opens when you right-click in a schematic.

If your mouse has a wheel, you can zoom while moving the mouse wheel up (zoom in) or down (zoom out). You can scroll vertically by holding down the **Shift** key while moving the mouse wheel or scroll horizontally by holding down the **Ctrl** key while moving the mouse wheel. This behavior can be changed using the **Zoom with mouse wheel** setting located in the Options dialog box, Schematic Editor/General section. If **Zoom with mouse wheel** is set to false (unchecked), you can scroll vertically with the mouse wheel, and zoom with the mouse wheel by pressing the **Shift** key.

The **arrow keys** scroll the view up, down, left, or right in small increments. The page up and page down keys scroll the view up or down in larger increments. If your mouse has a scroll wheel, move the wheel up to scroll up and down to scroll down. If you scroll so far that no objects

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

are in the view, select **Fit Drawing** from the **View** drop-down on the schematic editor top menu bar (or press **Ctrl+D**) to re-center the entire design, resized to fill the window.

Setting Schematic Editor Options

These options are set under **Schematic Editor** in the **Options** window.

To set **Schematic Editor** options:

1. Click **Tools** > **Options** > **General Options**.
2. Click **Schematic Editor**.
3. Make your changes on the appropriate panels:
    - General
    - Fonts
    - Colors
    - Wiring
    - Multiple Placement
    - Symbol Editor
4. Click **OK**.

Schematic Editor Options: General

These options are set in the **General** panel under **Schematic Editor** in the **Options** window.

- **Symbol Graphics** – specifies the graphic symbol style (**IEEE** or **Traditional**) to be used on schematics. Each style contains two or more active levels for the graphic objects that comprise a symbol.
- **SubCircuit Pin Spacing** – specifies the pin spacing in grid units for pins on a subcircuit component symbol. The default value is 1 grid unit.
- **Show Pin Labels** – specifies that pin labels are to be displayed in the **Schematic Editor**.
- **Property Display Angle Follows Symbol Angle** – specifies that displayed text properties rotate when the associated component is rotated. Disabled by default.
- **Update Property Display on Definition Update** – specifies that when component symbol definitions are updated, the displayed properties of instances of the updated component on the schematic are automatically updated. Enabled by default. If not enabled, only the symbol graphics are updated.
- **Auto Scroll when close to edges** – specifies that the **Schematic Editor** display will scroll automatically when the cursor is placed close to the edge of the editor window.
- **Show advanced property data** - turns on the display of less often used property tabs, such as Symbol and General for components.
- **Zoom with mouse wheel** – controls the behavior of the mouse wheel on mice so equipped. When enabled, you can zoom while moving the mouse wheel up (zoom in) or

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

down (zoom out); and scroll vertically by holding down the **Shift** key while moving the mouse wheel or scroll horizontally by holding down the **Ctrl** key while moving the mouse wheel. When disabled, you can scroll vertically with the mouse wheel, and zoom with the mouse wheel by pressing the **Shift** key

- **Net name display** – sets the net name property display distance from the net.
- **Symbol Scaling Factor** – Third-party vendors often follow a different symbol dimensioning system than that used in Maxwell. Symbol sizes that look appropriate in a third-party application may not be appropriate for Maxwell. To mitigate such issues, the Symbol Scaling factor can be used during import to scale the incoming symbol graphics for symbol formats such as SVG by the specified amount. Similarly, when exporting, the symbol can be rescaled to the original dimensions.
- **Selection Colors** – sets the colors of the first and subsequent objects as they are selected on a schematic.

Schematic Editor Options: Fonts

These options are set in the **Fonts** panel under **Schematic Editor** in the **Options** window.

Set the **Font Name** (from a drop-down selection list) and **Size** for text used on schematics on the **Fonts** tab of the **Schematic Editor Options** dialog box. A **Sample Text** display window shows the appearance of the specified font and size. **Apply this font to all property displays in the active schematic** specifies that all displays in the schematic editor are to use the font style and size you selected.

Schematic Editor Options: Colors

These options are set in the **Color** panel under **Schematic Editor** in the **Options** window.

The **Schematic Objects** panel allows you to set the color for each **Object Type**.

- **Object Type** – a drop-down menu used to select the object type whose colors you want to modify. Types are: **Components**, **Interface Ports**, **Page Ports**, **Global Ports**, **Grounds**, **Graphic Items**, **Wires**, **Buses, Title Blocks, Page Borders,** and **Page Border Text**.
- **Set Color** – implements the color you have selected for the object type.
- **Clear Color** – resets the color of the selected object type to its default value.

The **Individual Definitions** panel lets you set the color for specified **Components** and **Wire Domains**.

- **Component Name** – is used to identify the component whose colors you want to modify. (Visible only when the **Components Object Type** is selected.)
- **Add** – allows you to add a component whose colors you want to define. (Visible only when the **Components Object Type** is selected.)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Wire Domain** – a drop-down menu used to select the wire domain whose color you want to modify. Each Wire Domain is assigned a unique color by default. For Maxwell, the Domain is: **Conservative - Electrical**.
- **Set Color** – implements the color you have selected for the specified component. All future instances of the component added to the active schematic window display will possess the selected color.
- **Clear Color** – resets the color of the component to its default value.
- **Apply these color settings to the current schematic** – applies the color settings for the selected **Object Type** to objects of that type in the active schematic window.

Schematic Editor Options: Wiring Tab

The following options are set on the **Wiring** panel under **Schematic Editor** in the **Options** window.

Connectivity Options:

- **Show Merge Wire dialog before combining wires** – displays the **Merge Wire** dialog box before combining wires to allow for changes to the merge configuration.
- **Show Split Wire dialog before separating wires** – displays the **Split Wire** dialog box before separating wires to allow for changes to the split configuration
- **When renaming a physically separate piece of wire**, there are two choices:
  - **Split into a different net**
  - **Keep connected (name all pieces the same)**
- **Show GlobalPort Disconnect dialog before separating** – displays the **GlobalPort Disconnect** dialog box before separating wires to allow for changes to the global port configuration.

Schematic Editor Options: Multiple Placement

Multiple placement allows you to place several objects of the same type in the schematic using repeated mouse clicks. These options are set in the **Multiple Placement** panel under **Schematic Editor** in the **Options** window.

Check the desired objects in the **Items for multiple placement** list:

- **Components**
- **Interface Ports**
- **Grounds**
- **Page Connectors**
- **Global Ports**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Schematic Editor Options: Symbol Editor

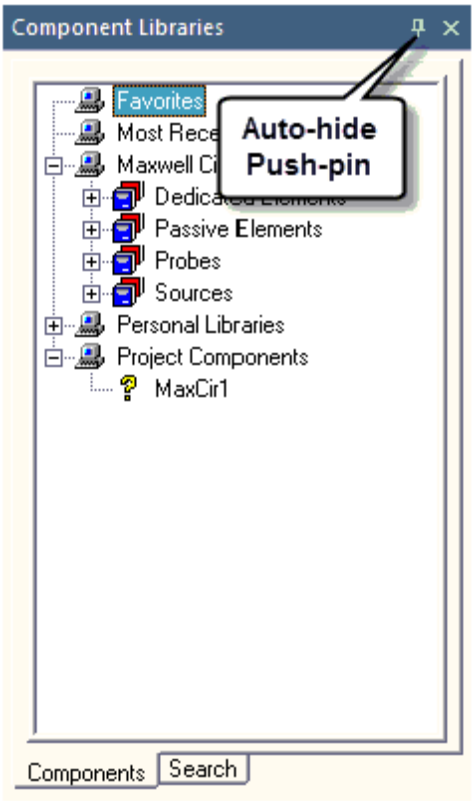
These options are set in the **Symbol Editor** panel under **Schematic Editor** in the **Options** window.

- **Adjust Pin Orientation on Drag** – enables symbol pin orientation to be adjusted automatically as pins are dragged to the desired side of a symbol in the symbol editor.

The Component Libraries Window

The Component Libraries window is a dockable window that allows you to select and search for schematic components that are available in the installed libraries. The Component Libraries window contains a Components tab and a Search tab.

A typical Component Libraries window is shown below. Your actual window contents will vary with the libraries installed and with use.

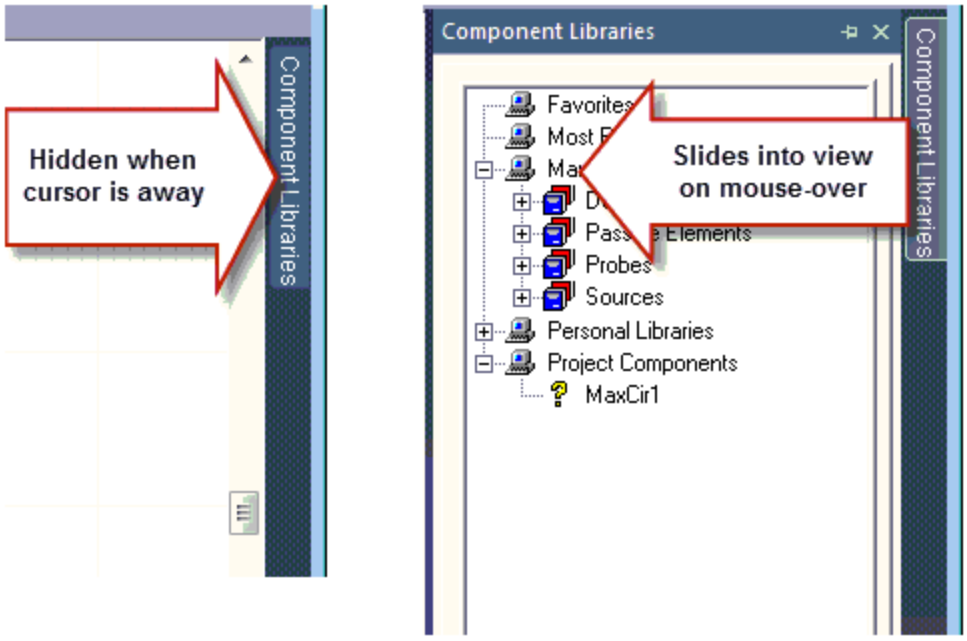


Auto-Hide Feature

Clicking the auto-hide push-pin in the upper right corner of the Component Libraries window toggles the auto-hide feature “on” and “off.” When auto-hide is enabled, the Component

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

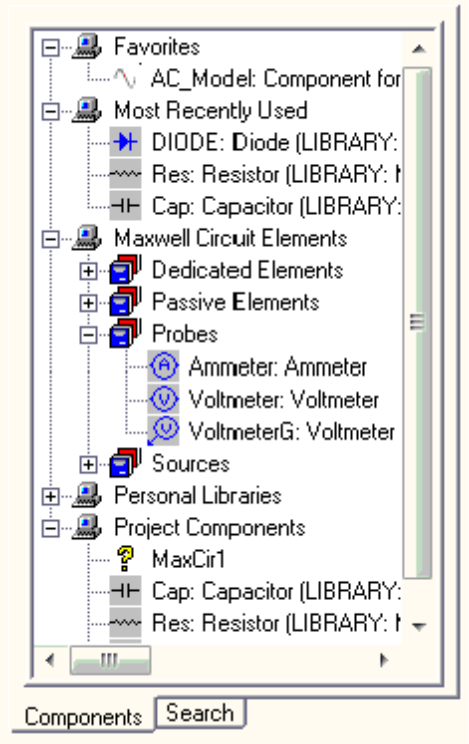
Libraries window slides out of sight and a Component Libraries “mouse-over button” appears on the edge of the main window. The Component Libraries window remains hidden until you move the mouse cursor over the mouse-over button. The Component Libraries window then slides into view for use; then hides again when the cursor moves out of the Component Libraries window.



The Components Tab

The **Components** tab in the **Component Libraries** window allows you to browse and select schematic components from the available elements libraries. The installed libraries are listed alphabetically in hierarchical folders grouped by component function. For example, the **Dedicated Elements** library expands as shown.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Individual elements are designated by icons next to the element abbreviation and name.

Right-clicking a component opens a shortcut menu where you can choose to:

- add the component to the Favorites folder
- place the component on a schematic
- view component help

The **Component** tab includes a **Favorites** folder and a **Most Recently Used** folder. These folders help provide quick access to components. Elements used in the current project are listed under **Project Components**.

Using the Favorites and Most Recently Used Folders

To add a library element as a favorite:

1. Select the desired element and right-click to display the shortcut menu.
2. Select **Add to Favorites** to place the component in the **Favorites** folder. When a component is in the **Favorites** folder, you can select, place, edit, view the component help, and load its example project (if one exists) from there.

To remove an element from the **Favorites** folder:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select an element and right-click to display the short cut menu.
2. Click **Remove from Favorites**.

The **Most Recently Used** folder contains a list of the most recently used elements. For convenience, these elements can be selected and placed from this folder. The number of elements in the folder is controlled by the **Tools** > **Options** > **General Options** window under the **Component Libraries Options** tab. The default number of elements is 10.

Help for Components

To launch help for a component from the **Components** tab of the **Component Libraries** window:

1. Right-click its icon.
2. Click **View Component Help**.

Placing Components on a Schematic

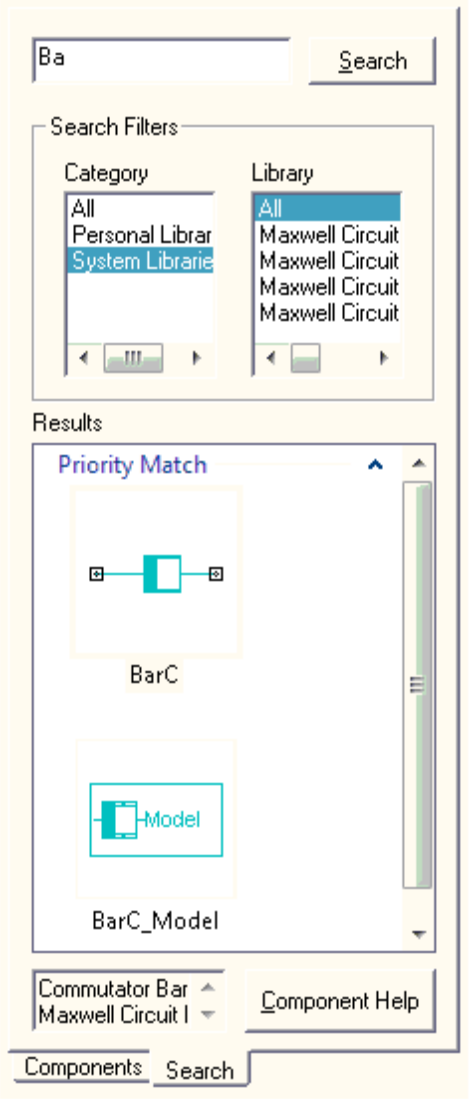
To place a component, do one of the following:

- Double-click its icon.
- Select, drag, and drop a component in the schematic editor.
- Right-click its icon, and then select **Place Component**.

The Search Tab

The **Search** tab allows you to enter full or partial words (case-insensitive) present in the names or descriptions of components in the search field. The search is done dynamically as you type. You can also restrict the search by select the categories and libraries you want to include in the search. Use the **Ctrl** key to select multiple categories or libraries.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For users running Microsoft® Windows, results appear in the **Results** panel as symbols representing the components matched by the search, and arranged in groups, depending on how well a search string matches the component. Generally, components which match the search string exactly appear in the **Best Match** group.

When you select a particular component you can see its description and library path in the text box at the bottom of the tab. You can either drag and drop the desired component from the **Results** panel onto the schematic, or double-click to attach it to the mouse. Use the **Component Help** button to view help information for components.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copying and Pasting Properties

You can copy and paste properties for primitive drawing elements (graphical objects) and components.

Primitive drawing elements (graphical objects)

You can copy and paste the common properties of the following primitive drawing elements (graphical objects):

- Arcs
- Circles
- Lines
- Rectangles
- Polygons
- Text
- Images

Components

Properties are copied only when property names match.

Parameters, and property displays are copied along with properties and their attributes.

> **Note:** Property displays are copied to the same location as the source if they are on the left, bottom, right, top or center. Custom location property displays are copied to the default bottom location.

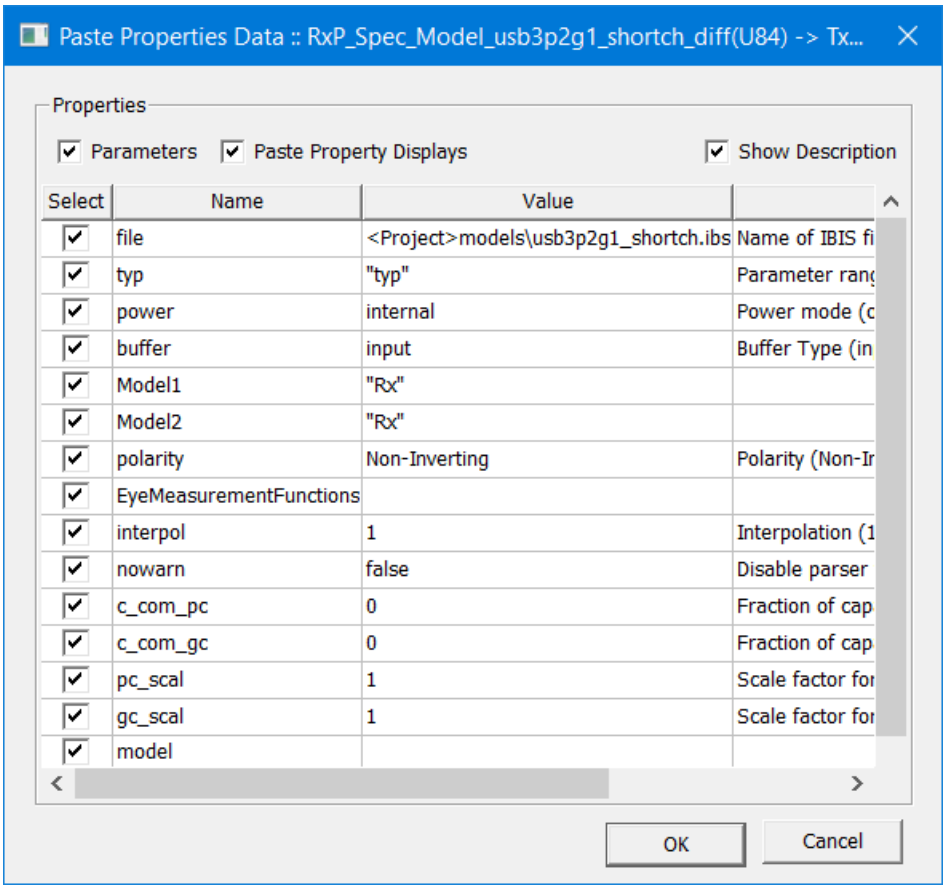
To Copy and Paste Selected Properties for Components

To copy and paste selected properties from one component to one or more components:

1. Select the component you want to copy from.
2. Use one of the following methods to access Copy Data:
    - Click **Edit** > **Copy Data**.
    - Right-click the object and select **Copy Data**.
    - Press **Ctrl+Shift+C**.
3. Select the components you want to paste to.
4. Paste the properties by using one of the following methods:
    - Select **Edit** > **Paste Data**.
    - Right-click the component and select **Paste Data.**
    - Press **Ctrl+Shift+V**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Paste Properties Data** window is displayed.



5. Do one of the following to select the properties to copy.
    - Select one or more of the following check boxes: Parameters, or Property Displays.
    - Click **Select** in the first column to toggle between selecting all and clearing all.
    - Click the **Select** check box for the property. If you do not want to include the property, clear the **Select** check box.
    - Click the **Show Description** check box to display the descriptions for each property.
6. Click **OK**. The properties are pasted into the component.

To Copy and Paste Common Properties for Primitive Drawing Elements

To copy and paste the common properties from one primitive drawing element (graphical object) to another:

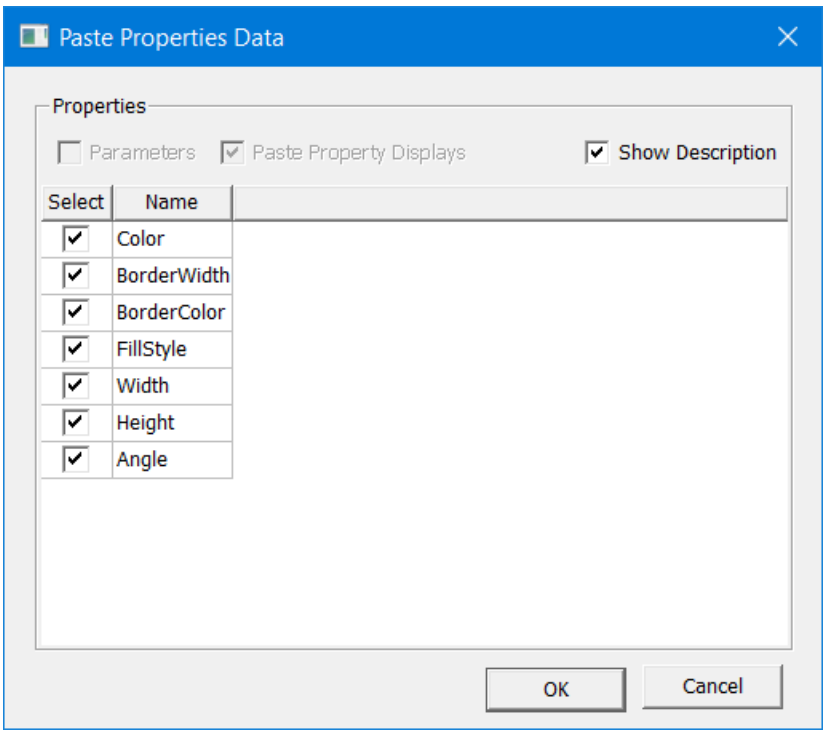
1. Select the object you want to copy from.
2. Use one of the following methods to access Copy Data:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click **Edit** > **Copy Data**.
- Right-click the object and select **Copy Data**.
- Press **Ctrl+Shift+C**.

3. Select the object you want to paste to.
4. Paste the properties by using one of the following methods:
   - Select **Edit** > **Paste Data**.
   - Right-click the component and select **Paste Data.**
   - Press **Ctrl+Shift+V**.

   The **Paste Properties Data** window is displayed.



5. In the **Properties** pane, keep the default selections or clear the check boxes for the properties you do not want to copy.

   Click the **Show Description** check box to display the description for each property.

6. Click **OK**. The properties are pasted into the object.

Related Properties

Copying and Pasting Properties

To Copy and Paste Selected Properties for Components

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Components

A number of components are available in the Circuit simulator.

To view and select components, click the **Components** tab in the **Component Libraries Window**. To expand a component subgroup, double-click its book icon. To read about a specific component, right-click its icon in the tree and select **View Component Help**.

The following types of elements are available in Maxwell Circuit Editor:

- Dedicated Elements
- Passive Elements
- Probes
- Sources

Once components are placed in the schematic for a project, they appear in the project tree beneath the **Project Components** branch. The most recently placed components also appear under the **Most Recently Used** branch in the project tree. You can also set **Favorites** that can be accessed from the project tree.

Dedicated Elements

The following dedicated elements are available in the Maxwell Circuit Editor project tree:

- AC_Model: Component for identifying windings with AC currents
- BarC: Commutator Bar
- BarC_Model: Model Data for Commutator Bar
- ECE3_Model: ECE 3-Phase Model
- ECE6_Model: ECE 6-phase Model
- ECEDCM_Model: Commutator Machine Model
- ECEF_Model: ECE Frequency Model
- ECEIM_Model: ECE Induction Motor Model
- ECEL_Model: ECE Linear Motion Model
- ECER_Model: ECE Rotation Model
- ECESRM_Model: ECE Switched Reluctance Motor Model
- ECET_Model: ECE Transformer Model
- ECEW_Model: ECE Winding Model
- Winding: Winding

The text before the colon (**:**) represents the component name and can be changed in the **Properties** window once the component is placed in the schematic.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AC_Model

The AC_model element is used to decrease the number of electric periods required to reach steady state when feeding windings of inductive loads with voltage sources in the external circuit editor. It assumes that the average current over one electric period in all windings defined in the AC_model is zero.

After each electric period, the initial current value of the windings is automatically manipulated to compensate for the offset current of the previous electric period. This allows you to modify currents for AC windings at the beginning of each period without interrupting the transient process. If the offset current of the last period is below a defined tolerance, no manipulation of the phase currents is triggered.

To change the name of the model placed on the sheet, click the symbol on the sheet, and change the name of the component in the property window (Value field in the DeviceName line, with the Param Values tab selected).

The AC_Model contains the following parameters:

- DeviceName, which identifies the name of device model.
- Windings, which is the AC winding list separated by commas. For example, PhA, PhB, PhC. Setting this parameter to “All” will include all windings.
- Frequency, which shows the frequency of the AC currents in Hz.
- Tolerance, which is the ratio of the DC component to the AC amplitude that defines the threshold value triggering a manipulation of the initial current for the subsequent electric period.

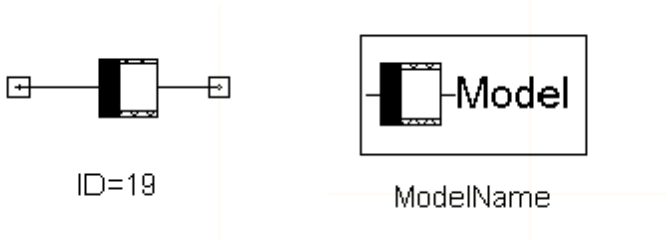
> **Note:** For Maxwell 2D/3D designs of AC machines, RMxprt automatically enables an alternating flux (AF) model in the voltage definition for each winding to reduce transient solution time to reach steady state when the Maxwell 2D or 3D model is created. Also refer to *Using the AF (alternating flux) model* for information on using the AF model to fast reach steady state for transient simulations whose excitation is the Sinusoidal Voltage Source component.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Commutator Bar and Commutator Bar Model

The commutator bar element is intended to be used for the motor model with a commutator. This element models the variable (periodic) contact resistance between the brush and the commutator bars, as well as the switching (commutation of the current) that occurs when the brush makes contact with the two adjacent commutator bars.

The element itself must always be used together with the corresponding commutator bar model. The commutator bar model can be dropped on the sheet anywhere and needs no connections. Only the commutator bar element itself should be connected as required by the application. The commutator bar elements need to reference the applicable commutator model.



Once the commutator bar element has been dropped on the sheet, you can double-click it to access the properties (make sure the **Parameter Values** tab is selected). Specify the applicable commutator bar model name in the MOD line and also the Lag parameter in degrees. Lag identifies the angle the commutator bar has to rotate from TIME = 0 in the chosen sense of rotation until it is perfectly aligned with the brush. By default, the element ID and lagging angle in degrees are displayed next to the element.

The commutator bar model needs to be dropped on the circuit sheet. It is unique for every commutator bar element. The commutator bar model contains the following parameters:

- Model name that has to be referenced by all the commutator bar elements;
- R, the full contact resistance between brush and commutator bar, regardless of which of the two is wider;
- WidB is the brush width in mechanical degrees;
- WidC is the commutator bar width in mechanical degrees (does not include the insulation between two adjacent bars);
- Period is the angular periodicity of the positive (or negative) brushes; use 360 for a two pole machine, 180 for a four pole machine with lap winding, etc.

Winding

The winding element is used in the Maxwell Circuit Editor to create the necessary connection between the finite element model (the type of solution that supports the concept of winding, such as the transient type of analysis, with or without motion) and the driving circuits. It is necessary

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

that the name(s) assigned for the winding(s) in the finite element model are matched exactly in the driving circuit created in Maxwell Circuit Editor. Windings can be placed on the design sheet at any moment while you are creating the circuitry to be used to drive the finite element windings.

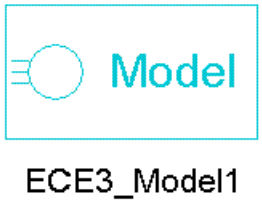
To change the name of the winding placed on the sheet, click the winding symbol on the sheet, and change the name of the component in the property window (Value field in the DeviceName line, with the Param Values tab selected).

> **Note:** The dot next to the winding symbol is used as the positive reference for the initial current (positive current is oriented from the "dotted" terminal toward to "undotted" terminal of the winding, through the winding).

ECE3 Model

The ECE3 model enables you to create an ECE model from Maxwell transient solutions. Compared with the procedure for creating ECE model from Maxwell magnetostatic solutions, creating an ECE model directly from Maxwell transient solutions takes advantage of winding and motion setups in the Maxwell transient solver.

The ECE3 model is used to set up the sweeping of currents in three-phase windings. To change the name of the model placed on the sheet, click the symbol on the sheet, and change the name of the component in the property window (Value field in the DeviceName line, with the Parameter Values tab selected).



The ECE3_Model contains the following parameters:

- DeviceName – the name of the device model with default name ECE3_Model1.
- Windings – specifies the comma-separated name list of three-phase windings whose currents are to be swept.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- CurrentSweeps – Specifies the current amplitude sweep list. For this, individual comma-separated values or range of values (linear step) can be specified.
    - For individual current values, the syntax is “v1, v2, v3” (e.g., 0 A, 5 A, 10 A). The values must be input in ascending order (i.e., v1<v2<v3,…)
    - For a value range with linear step, the syntax is (dv1, n1), where dv1 denotes the step and n1 the number of steps.
    - Value ranges always require an initial value, which is set to 0A by default.
    - For example, "(0.5 A, 5)" is equivalent to "0, 0.5 A, 1 A, 1.5 A, 2 A, 2.5 A"
    - Multiple value range inputs can be combined with each other. The later value range input will start with its first linear step at the end of the preceding value range.
    - For example, "(0.5 A, 5), (1 A,2)" is equivalent to "0, 0.5 A, 1 A, 1.5 A, 2 A, 2.5 A, 3.5 A, 4.5 A"
    - (Multiple) value range inputs can also be combined with individual values. Value range inputs always start from the last current step defined.
    - For example, "(0.5 A, 5), 3 A, (1 A, 2)" is equivalent to "0, 0.5 A, 1 A, 1.5 A, 2 A, 2.5 A, 3 A, 4 A, 5 A"
    - Make sure that the individual value inputs are larger than the last current value of value ranges preceding it. (e.g., 3 A > 2.5 A=5*0.5 A as in the example above.)
- PhAngIntervals – specifies the sweep types, or the number of phase-angle sweep intervals. For three-phase windings, only two independent variables are swept to derive three-phase currents, instead of directly sweeping three-phase currents. The two independent variables could be DQ currents (I_d, I_q), two-phase currents (I_α, I_β), or current amplitude and phase angle (I_m,β).

  PhAngIntervals = 0, 1, or 2, is defined for the DQ current sweep type, and 3 for the two-phase current sweep type.

    - When PhAngIntervals = 0, both I_d and I_q are swept based on the original setup of CurrentSweeps, and the look-up table is extended according to odd-symmetric conditions in a post-processing step.
    - When PhAngIntervals = 1, the I_d sweep is extended symmetrically to sample negative and positive current values. In contrast, the I_q sweep is based on the original setup of CurrentSweeps and extended in the look-up table according to odd-symmetric conditions in a post-processing step.
    - When PhAngIntervals = 2, both I_d and I_q sweeps are extended symmetrically from negative to positive, sampling positive and negative values for both, I_d and I_q. Accordingly, the computed look-up table does not need to be extended in post-processing.
    - When PhAngIntervals = 3, both I_α and I_β sweeps are extended symmetrically from negative to positive, and the computed look-up table is not extended in post-processing.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

    - When PhAngIntervals ≥ 12, the current amplitude and phase angle sweep type are defined. The current amplitude I_m is swept based on the setup of CurrentSweeps, and the phase angle is swept from 0 to 360 electrical degrees with the increment $\Delta\beta = 360°/(\text{PhAngIntervals})$.
- Res – phase winding resistance (available when Show Hidden is checked. Default value is 1 mΩ).
- IndE – phase end-leakage inductance (available when Show Hidden is checked. Default value is 1 µH).

An ECE3_Model can be used individually to create the ECE model of a three-phase inductor, or combined with the ECER_Model for three-phase PM machines. It can also be combined with the ECER_Model and ECEW_Model for winding excited three-phase synchronous machines. An ECE3_Model with DQ current sweep type must always be used with an ECER_Model.

For ECE model creation, the setup of Stop Time and Timestep are ignored. At Maxwell run time, the circuit simulator assigns currents for all windings step-by-step based on the lists of current sweeps until all sweeps are finished. An ECE model is then created automatically. You can import this ECE model in Twin Builder via the **Twin Builder** > **SubCircuit** > **Maxwell Component** > **Add Equivalent Circuit** menu command. For more information on coupling Maxwell designs to Twin Builder components, refer to the *Maxwell Equivalent Circuit Component* topic in the Twin Builder Help.

> **Note:** You can add an ECE3_Model to an existing external circuit without deleting the original circuit. In such cases, the original circuit is ignored and can be kept for future use. If you want to recover the original circuit for a transient simulation, instead of ECE model creation, you just need to deactivate each ECE related model by right-clicking the model and selecting Deactivate in the short-cut menu.

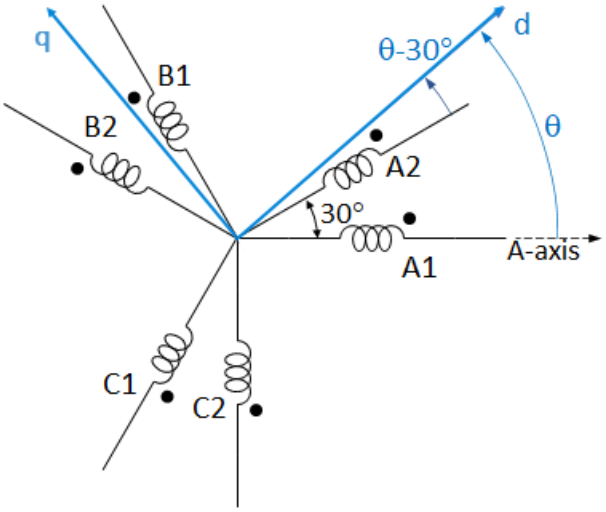
For more information on coupling Maxwell designs to Twin Builder components, refer to *Coupling Maxwell to a Twin Builder Component*.

ECE6 Model

The ECE6 model enables you to create an ECE model from Maxwell Transient solutions. Compared with the procedure for creating ECE model from Maxwell Magnetostatic solutions, creating an ECE model directly from Maxwell Transient solutions takes advantage of winding and motion setups in Maxwell Transient.

The ECE6 model is used to set up the sweeping of currents in a six-phase winding, which is described as dual three-phase windings. These two three-phase windings are shifted by 30 electrical degrees. The figure below shows the set-2 three-phase windings lagging behind the set-1 three-phase windings by 30 electrical degrees, and dq axes in rotor with the q-axis lagging behind the d-axis by 90 electrical degrees.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

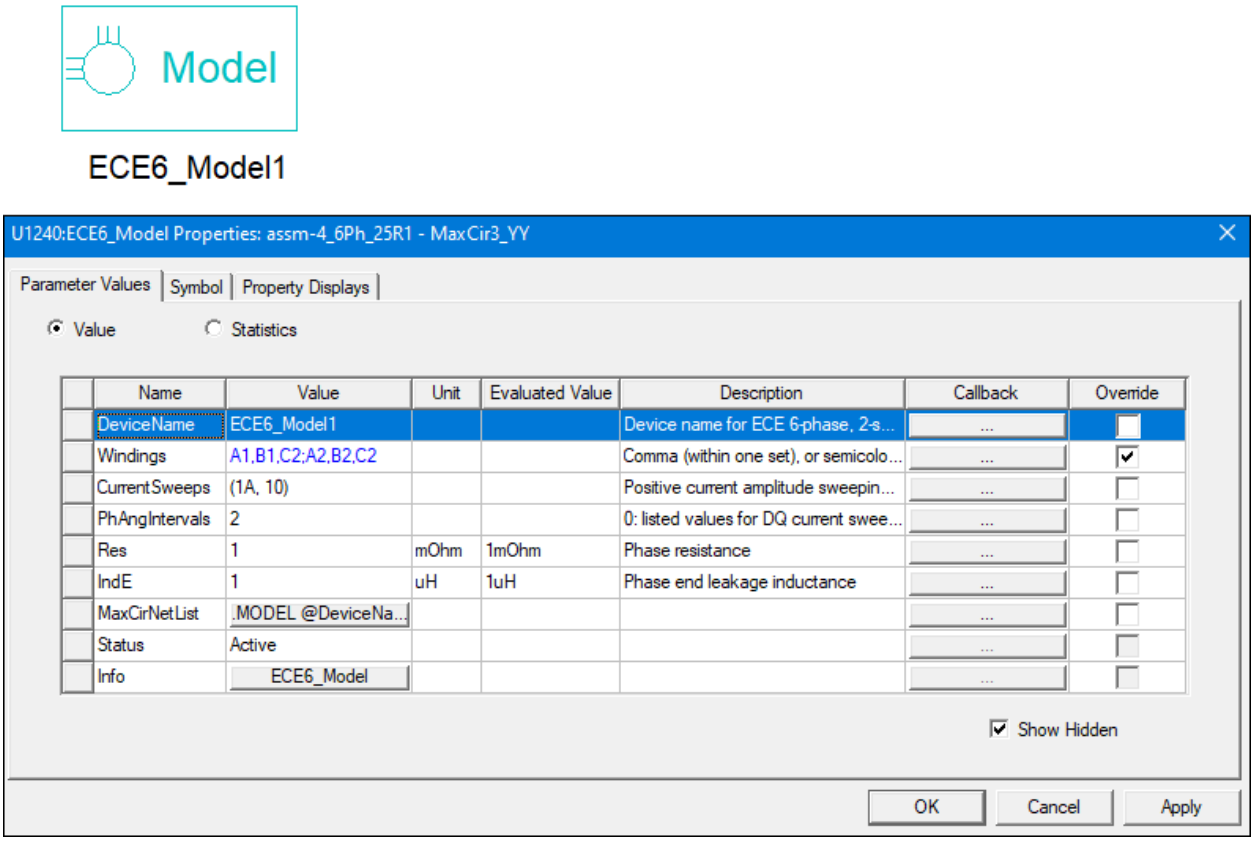


If the rotor initial position is defined with θ = 0 as the position where the rotor d-axis aligns with the axis of the phase A1 winding. That is, the phase A1 gets maximum flux linkage when θ = 0.

> **Note:** With Ansys Electronics Desktop release 25r1, Maxwell and RMxprt use the Park transformation (R. H. Park, "Two-reaction theory of synchronous machines", Transactions of the American Institute of Electrical Engineers, (IEEE)- Vol. 48, Iss: 3 (1929): 716-727), which defines the Q axis as being 90 electrical degrees behind the D axis in space. In previous releases, Maxwell and RMxprt used the Hancock transformation (N. N. Hancock, "*Matrix analysis of electrical machinery*, Pergamon Press, 1974), which defined the Q axis as being 90 degrees ahead the D axis.

To change the name of the model placed on the sheet, click the symbol on the sheet, and change the name of the component in the property window (Value field in the DeviceName line, with the Parameter Values tab selected).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The ECE6_Model contains the following parameters:

- **DeviceName** – the name of the device model with default name ECE6_Model1.
- **Windings** – Specifies winding names in the order of arrangement. Windings are separated by commas within each three-phase set, and by semicolon between the two three-phase sets. Notice that the winding list always starts with the leading set and ends with the lagging set. For the phase windings as shown in the figure above, the winding list is specified as: A1 B1, C1; A2, B2, C2
- **CurrentSweeps** – Specifies sweeping values for d- and q-axis currents, Id and Iq, respectively; you only need to list 0 and all positive sweeping currents. All negative currents will be swept, or extended, symmetrically from the specified positive sweeping values. For example: 0, 1, 10.
- **PhAngIntervals** – Specifies the type to extend the sweeping list and look-up table. Valid values are: 0, 1, or 2 only. Value 0 specifies the type that both Id and Iq are post-extended; Value 1 specifies the type that Id is pre-extended, and Iq is post-extended; Value 2 specifies the type that both Id and Iq are pre-extended. Here, “pre-extended” means the sweeping list for a current is symmetrically extended from the sweeping values specified in CurrentSweeps, and “post-extended” means the look-up table for a current, which is derived from the sweeping values specified in CurrentSweeps, is extended based on even

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

or odd symmetric conditions. In a post extension, the symmetric conditions are applicable only for fundamental fields. Therefore, it is not necessary to consider all harmonics in the look-up table for post-extended variables. In such a case, all harmonic components will be filtered by using the average values over various rotor positions for all dq0 flux linkages and torque to reduce the length of the look-up table.

- **Res** – phase winding resistance (available when **Show Hidden** is checked).
- **IndE** – phase end-leakage inductance (available when **Show Hidden** is checked).

> **Important:** An ECE6_Model must be used with an ECER_Model. When combined with ECE6_Model, the ECER_Model supports only "1. DQ0 flux output" for the Output field.

Transformation between the Six-Phase and dq0 Systems

The current transformation from dq0 system to set-1 ABC system is

$$\begin{bmatrix} I_{A1} \\ I_{B1} \\ I_{C1} \end{bmatrix} = \mathbf{C}(\theta) \cdot \begin{bmatrix} I_{d1} \\ I_{q1} \\ I_{01} \end{bmatrix}$$

where C(θ) is the Park transformation matrix, as expressed below:

$$\mathbf{C}(\theta) = \begin{bmatrix} \cos(\theta) & \text{-}\sin(\theta) & 1 \\ \cos\left(\theta - \frac{2\pi}{3}\right) & \text{-}\sin\left(\theta - \frac{2\pi}{3}\right) & 1 \\ \cos\left(\theta - \frac{4\pi}{3}\right) & \text{-}\sin\left(\theta - \frac{4\pi}{3}\right) & 1 \end{bmatrix}$$

Because the set-2 winding lags behind the set-1 winding by 30 electrical degrees, the current transformation from dq0 system to set-2 ABC system is

$$\begin{bmatrix} I_{A2} \\ I_{B2} \\ I_{C2} \end{bmatrix} = \mathbf{C}\left(\theta - \frac{\pi}{6}\right) \cdot \begin{bmatrix} I_{d2} \\ I_{q2} \\ I_{02} \end{bmatrix}$$

The contribution of 0-axis currents (I_{01}, I_{02}) to winding flux linkages can be considered by zero-inductance L_0, which is usually assumed to be constant. Therefore, it is not necessary to sweep 0-axis currents. In general, to cover all possible operation cases, the remaining four independent currents (I_{d1}, I_{q1}, I_{d2}, I_{q2}) need to be swept. However, in most applications, to get the benefits of 6-phase windings, these dual three-phase windings are operated with balanced currents, that is, in the winding set lagging behind by 30 electrical degrees in space, the currents lag behind by 30 degrees in time. In such cases, the dq currents on both winding sets are the same. Therefore, to save FEA simulation time, sweep only two independent currents (I_d, I_q) during FEA simulation, and assign

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\begin{cases} I_{d1} = I_{d2} = I_d \\ I_{q1} = I_{q2} = I_q \end{cases}$$

During circuit simulation, measure dual three-phase currents (I_{A1}, I_{B1}, I_{C1}) and (I_{A2}, I_{B2}, I_{C2}), and transform them to dq currents by

$$\begin{bmatrix} I_{d1} \\ I_{q1} \\ I_{01} \end{bmatrix} = \mathbf{C}^{-1}(\theta) \cdot \begin{bmatrix} I_{A1} \\ I_{B1} \\ I_{C1} \end{bmatrix}$$

and

$$\begin{bmatrix} I_{d2} \\ I_{q2} \\ I_{02} \end{bmatrix} = \mathbf{C}^{-1}\left(\theta - \frac{\pi}{6}\right) \cdot \begin{bmatrix} I_{A2} \\ I_{B2} \\ I_{C2} \end{bmatrix}$$

where

$$\mathbf{C}^{-1}(\theta) = \frac{2}{3}\begin{bmatrix} \cos(\theta) & \cos\left(\theta - \frac{2\pi}{3}\right) & \cos\left(\theta - \frac{4\pi}{3}\right) \\ \text{-sin}(\theta) & \text{-sin}\left(\theta - \frac{2\pi}{3}\right) & \text{-sin}\left(\theta - \frac{4\pi}{3}\right) \\ \frac{1}{2} & \frac{1}{2} & \frac{1}{2} \end{bmatrix}$$

The dq currents used in the look-up table are

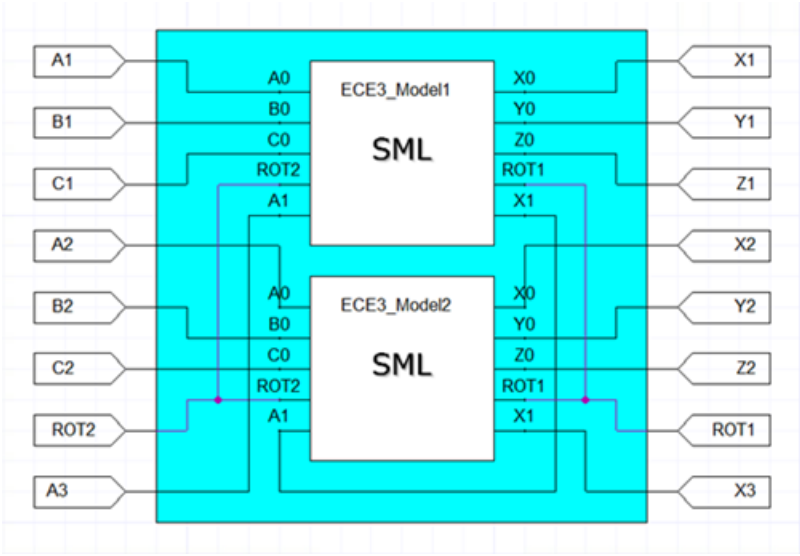
$$\begin{cases} I_d = (I_{d1} + I_{d2})/2 \\ I_q = (I_{q1} + I_{q2})/2 \end{cases}$$

Because the two sets of three-phase windings have the same turns, the fundamental field produced by the average dq currents will be the same as that produced by the real dq currents. Therefore, for unbalanced operations, replacing the real dq currents (I_{d1}, I_{q1}) and (I_{d2}, I_{q2}) in the two winding sets by the average dq currents (I_d, I_q) will not cause noticeable errors.

Circuit Model and Look-up Table

The circuit model for 6-phase machines consists of two sub-circuits of 3-phase machines with the dq currents of the other sub-circuit being the input arguments, as shown below, where the initial position, an input argument of the sub-circuit model, of ECE3_Model2 is set 30 electrical degrees lagging behind ECE3_Model1. The mechanical ports represented by (ROT1, ROT2) of the two sub-circuit are connected in parallel, and the torque values in the look-up table are a half of the FEA solutions for each sweep. The field winding, if it exists, represented by (A3, X3) in the circuit model, is obtained by series connection of the field windings, represented by (A1, X1), of the two subcircuits.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The flux linkage values in the look-up table of the field winding in the sub-circuit are a half of the FEA solutions. The input dq currents of the other subcircuit are used to get the average dq currents as the input of the look-up table. The following is an example of the data structure of the look-up table of a 6-phase machine with wound field.

```
B_BasicData
  Version   1.0
  Poles     4
  L1d  0.000888929
E_BasicData

B_PhaseImp 7
    PhaseA    1.0000000000e-03    1.0000000000e-06
    PhaseB    1.0000000000e-03    1.0000000000e-06
    PhaseC    1.0000000000e-03    1.0000000000e-06
   PhaseA1    1.0000000000e-03    1.0000000000e-06
   PhaseB1    1.0000000000e-03    1.0000000000e-06
   PhaseC1    1.0000000000e-03    1.0000000000e-06
     Field    1.0000000000e-03    1.0000000000e-06
E_PhaseImp

B_Sweepings
  Id_Iq     (7:  -10 -5 -1 0 1 5 10)
            (7:  -10 -5 -1 0 1 5 10)
  Iw        (5:  0 4 5 6 10)
  Rotate    (49:  0 1.25 2.5 ... 60)
E_Sweepings

B_OutputMatrix DQ0
        0   -2.0359505219e-01   -1.1958557789e-01    6.8140161295e-03   -3.7702990147e-01   -3.1186012134e+00
        1   -2.0314518063e-01   -1.1829390416e-01    7.9496955252e-03   -3.7825548839e-01   -3.1680201040e+00
    ......
    12004    2.5756484736e-01    8.1949917793e-02   -1.0324170975e-03    5.6583790148e-01   -5.4424987350e+00
E_OutputMatrix
```


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The subcircuit data include basic data, specified resistances and leakage inductances of all windings, value list for all controlling variables, and value list for all controlled variables of all sweeps.

The basic data are listed between “B_BasicData” and “E_BasicData”, which include the version number, the number of poles, and d- and q-axis leakage inductance between the 2-set 3-phase windings.

The data between “B_PhaseImp” and “E_PhaseImp” list the winding resistances and end leakage inductances, which are not included in 2D FEA results, for each winding. The winding resistances and end leakage inductances are obtained from specified parameters Res and IndE of the ECE6_Model, as well as ECEW_Model if it is included.

The data between “B_Sweeping” and “E_Sweeping” provide all sweeping variables, and their sweeping data lists. Each sweeping variable, corresponding to one ECE sweeping model, starts with a keyword and follows one or two sweeping data list. The sweeping data list is in the format of (n: v0 v1 …) representing the number of sweeps and sweeping values. Keyword “Id_Iq” means the dq current sweep, “Iw” represents wound field current sweep, and “Rotate” indicates the rotor position sweep.

The data between “B_OutputMaxtrix” and “E_OutputMatrix” lists solutions of all sweeps. The first column is the sweeping index from 0 to n-1, here n is the total number of sweeps. The total number of sweeps is 7x7x5x49 = 12005. The 2nd to 4th columns list the dq0 flux linkages of all sweeps for the present sub-circuit. If the field winding exists, the 5th column lists its flux linkage. The last column list torque values of all sweeps.

From FEA solutions, we get results of flux linkages of all windings. The dq0 flux linkages can be obtained from

$$\begin{bmatrix} \lambda_{d1} \\ \lambda_{q1} \\ \lambda_{01} \end{bmatrix} = \mathbf{C}^{-1}(\theta) \cdot \begin{bmatrix} \lambda_{A1} \\ \lambda_{B1} \\ \lambda_{C1} \end{bmatrix}$$

and

$$\begin{bmatrix} \lambda_{d2} \\ \lambda_{q2} \\ \lambda_{02} \end{bmatrix} = \mathbf{C}^{-1}\left(\theta - \tfrac{\pi}{6}\right) \cdot \begin{bmatrix} \lambda_{A2} \\ \lambda_{B2} \\ \lambda_{C2} \end{bmatrix}$$

Because set-2 windings lag behind set-1 windings by 30 electrical degrees, the ABC flux linkages of the set-2 windings will lag behind those of the set-1 windings by 30 degrees. As the result, the dq0 flux linkages satisfy

$$\begin{cases} \lambda_{d1}(\theta + 30^\circ) = \lambda_{d2}(\theta) \\ \lambda_{q1}(\theta + 30^\circ) = \lambda_{q2}(\theta) \\ \lambda_{01}(\theta + 30^\circ) = -\lambda_{02}(\theta) \end{cases}$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The repeatable period for the d- and q-axis flux linkages is 60 electrical degrees, but that for the 0-axis flux linkage is 120 electrical degrees. Therefore, all dq0 flux linkages must be extended to 120 electrical degrees by

$$\begin{cases} \lambda_{d1}\left(\theta+60^{\circ}\right)=\lambda_{d1}\left(\theta\right) \\ \lambda_{q1}\left(\theta+60^{\circ}\right)=\lambda_{q1}\left(\theta\right) \\ \lambda_{01}\left(\theta+60^{\circ}\right)=-\lambda_{01}\left(\theta\right) \end{cases}$$

The extended dq0 flux linkages in the final look-up table are based on the same dq currents in two winding sets. During the circuit simulation, if the dq currents in the two windings are not the same, the dq flux linkages obtained from the look-up table must be modified as described below. The modified d-axis flux linkage is

$$\lambda'_{d1}=\lambda_{Fd}+L_{d1}I_{d1}+M_{d12}I_{d2}$$

$$=\lambda_{d1}+L_{1d}\left(I_{d1}-I_{d2}\right)/2$$

where λ_{Fd} is the flux linkage produced by wound field winding current, or PM, and

$$L_{1d}=L_{d1}-M_{d12}$$

$$\lambda_{d1}=\lambda_{Fd}+L_{d1}I_{d}+M_{d12}I_{d}$$

which can be directly obtained from the look-up table. Similarly, the q-axis flux linkage is modified by

$$\lambda'_{q1}=\lambda_{q1}+L_{q1}\left(I_{q1}-I_{q2}\right)/2$$

where L_{1q} = L_{1d}, and λ_{q1} can be directly obtained from the look-up table. To filter the slot effects, the d- or q-axis leakage inductance L_{1d} can be computed from

$$L_{1d}=\frac{1}{n}\sum\nolimits_{i=1}^{n}0.5\left[\left(L_{d1}-M_{d12}\right)_{i}+\left(L_{q1}-M_{q12}\right)_{i}\right]$$

where *i* represents the sweeping index.

The d- and q-axis self-inductance and mutual inductance between 2 winding sets are derived from

$$\begin{bmatrix} L_{d1} & 0 & 0 \\ 0 & L_{q1} & 0 \\ 0 & 0 & L_{01} \end{bmatrix}=\mathbf{C}^{-1}(\theta)\cdot\begin{bmatrix} L_{A1} & M_{A1B1} & M_{A1C1} \\ M_{B1A1} & L_{B1} & M_{B1C1} \\ M_{C1A1} & M_{C1B1} & L_{C1} \end{bmatrix}\cdot\mathbf{C}(\theta)$$

and

$$\begin{bmatrix} M_{d12} & 0 & 0 \\ 0 & M_{q12} & 0 \\ 0 & 0 & M_{012} \end{bmatrix}=\mathbf{C}^{-1}(\theta)\cdot\begin{bmatrix} M_{A1A2} & M_{A1B2} & M_{A1C2} \\ M_{B1A2} & M_{B1B2} & M_{B1C2} \\ M_{C1A2} & M_{C1B2} & M_{C1C2} \end{bmatrix}\cdot\mathbf{C}\left(\theta-\frac{\pi}{6}\right)$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For ECE model creation, the setup of Stop Time and Timestep are ignored. At Maxwell run time, the circuit simulator assigns currents for all windings step-by-step based on the lists of current sweeps until all sweeps are finished. An ECE model is then created automatically. You can import this ECE model in Twin Builder via the **Twin Builder** > **SubCircuit** > **Maxwell Component** > **Add Equivalent Circuit** menu. For more information on coupling Maxwell designs to Twin Builder components, refer to the *Maxwell Equivalent Circuit Component* topic in the Twin Builder Help.

> **Note:**
>
> You can add an ECE6_Model to an existing external circuit without deleting the original circuit. In such cases, the original circuit is ignored and can be kept for future use. If you want to recover the original circuit for a transient simulation, instead of ECE model creation, you just need to deactivate each ECE related model by right-clicking the model and selecting Deactivate in the short-cut menu.

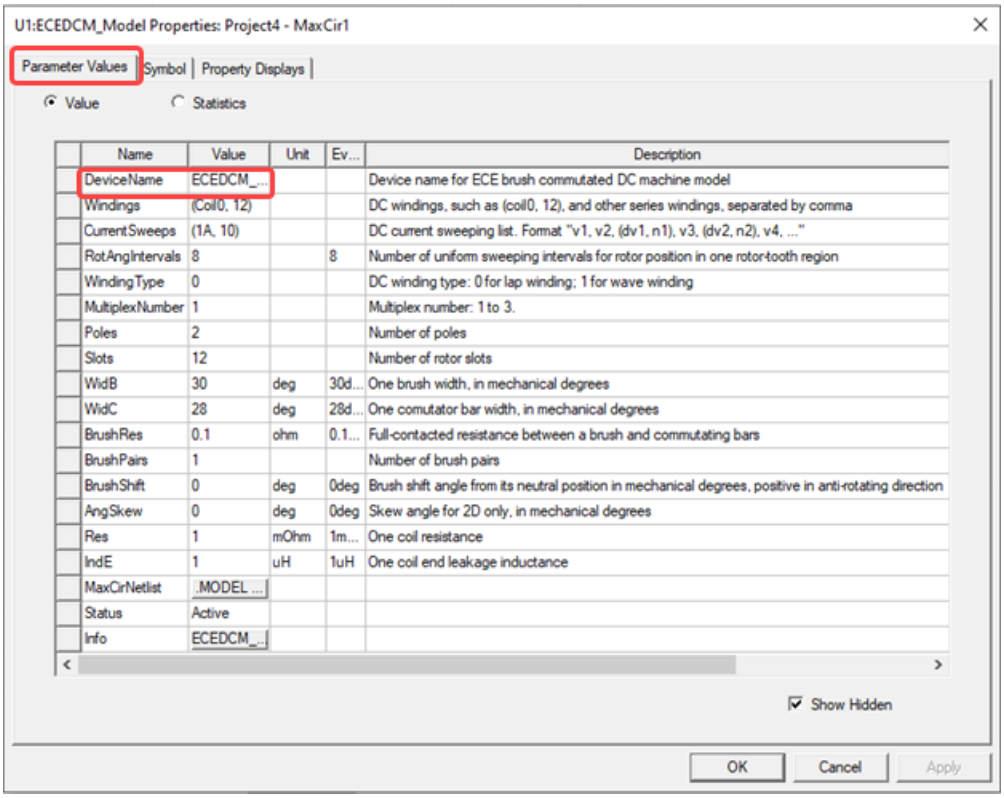
For more information on coupling Maxwell designs to Twin Builder components, refer to Coupling Maxwell to a Twin Builder Component.

ECEDCM Model

The ECEDCM model enables you to create a reduced-order model (ROM) of a commutator machine, such as a permanent magnet DC (PMDC) machine, an electrically excited DC machine, or a universal motor, from Maxwell 2D and 3D transient solutions. The symbol of an ECEDCM model element is shown below:

To change the name of the model placed in the Schematic Editor window, double-click the symbol to open its Properties window, and from the **Parameter Values** tab, enter the new name in the Value field for the DeviceName parameter:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The ECEDCM model contains the following parameters:

- **DeviceName** – specifies the name of the device model; the default name is *ECEDCM_Model1*.
- **Windings** – specifies an armature winding, as well as all other windings connected in series with the armature winding. The armature winding is specified by two parameters inside a pair of parentheses. The first parameter specifies the base coil name, which consists of a non-digit string and an index number of 0 or 1, such as *Coil0*, or *Coil1*. The second parameter specifies the number of armature coils.
  Other series windings are directly specified with winding names, separated by a comma "," between any two windings. For example, if a PMDC machine has an armature winding consisting of twelve coils with the base coil name of *Coil0*, then the armature winding is specified as *(Coil0, 12)*. Windings of a universal motor having eighteen armature coils with base coil name of *Coil1* can be specified as *(Coil1, 18), Field*; here *Field* is the field winding name.
- **CurrentSweeps** – specifies input current sweeps for the armature winding and all windings connected in series with the armature winding; see CurrentSweeps for the ECEW model for more details. For a PMDC motor, you need to list 0 and all positive sweeping currents. For a universal motor, you need to list all negative, 0, and positive

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

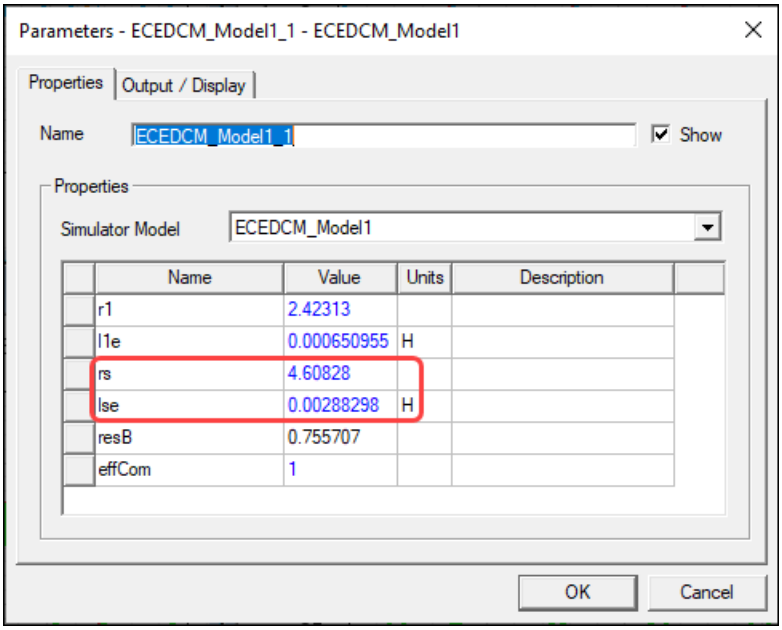
sweeping currents.

- **RotAngIntervals** – specifies the number of uniform sweeping intervals for rotor position in one rotor-slot region. For an eighteen-slot rotor, one rotor-slot region is twenty mechanical degrees. If RotAngIntervals = 8, the sweeping step for rotor position will be 2.5 mechanical degrees.
- **WindingType** – specifies armature winding type: 0 for lap winding and 1 for wave winding.
- **MultiplexNumber** – specifies the multiplex number: 1 for simplex winding, 2 for duplex winding, and 3 for triplex winding.
- **Poles** – specifies the number of poles of the machine.
- **Slots** – specifies the number of rotor slots.
- **WidB** – specifies one brush width in mechanical degrees.
- **WidC** – specifies one commutator bar width in mechanical degrees.
- **BrushRes** – specifies full-contacted resistance between a brush and commutator bars.
- **BrushPairs** – specifies the number of brush pairs. For a lap winding, the number of brush pairs is always the same as the number of pole pairs. For a wave winding, it could be 1 or the number of pole pairs required to have a periodic circuit.
- **BrushShift** – specifies the brush shift angle from its neutral position in mechanical degrees; it is positive in the anti-rotating direction.
- **Res** – specifies the resistance of one armature coil.
- **IndE** – specifies the end-leakage inductance of one armature coil for 2D.

An ECEDCM model can be used individually to create a ROM of a PMDC machine, a series-excited DC motor, or a universal motor (series-excited AC motor). Combined with an ECEW model that sets up the shunt winding current sweep, this model can also be used to create a ROM of a separate excitation DC machine.

When the winding list in the ECEDCM model includes an armature winding and one or more series windings, the total resistance and the end leakage inductance of all other series windings can be specified in the created Simplorer circuit model via arguments **rs** and **lse**, respectively, as shown in the following figure:

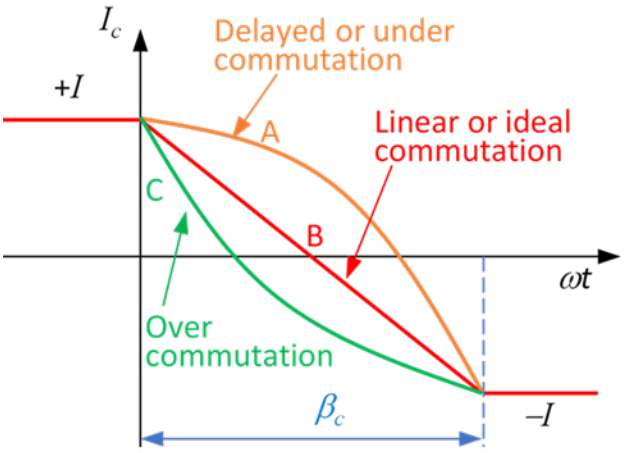
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Simplorer circuit model also includes the **effCom** argument, which is used to specify the weight of the commutating current effect. The commutating current effect is described below.

During a Maxwell simulation, armature coil currents are determined from the input armature sweeping current based on linear commutation. However, the assumed linear commutation (see curve B in the following figure) is an ideal case. In real applications, the current shape of the commutating coil depends on the total induced voltage of the coil, which includes inductance- and rotation-induced voltage components. When the inductance-induced voltage is larger than the rotation-induced voltage, the current commutation will lag curve B, as shown by curve A. This case is called delayed or under-commutation. When the rotation-induced voltage is larger than the inductance-induced voltage, the current will commutate as it approaches curve B, as shown by curve C. This case is called over-commutation. The difference between the real and the linear commutation currents is called commutating current.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



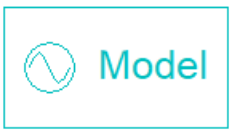
The commutating current will change the main magnetic field because the axis of the commutating coil aligns with the axis of the main field. As a result, it will affect the values of flux linkages of all coils and the rotor torque. To improve model accuracy, we can include the effect of commutating current on the main field. Therefore, in addition to sweeping the input armature current, we also sweep the commutating current. In this way, we can consider the commutating current effects on the main field. However, the resultant commutating current effect is estimated based on one or two coils rather than the commutating current effects of one coil by one coil, so it is not highly accurate.

Therefore, you can scale the weight of the commutating current effect using the **effCom** argument. When **effCom** = 0, the commutating current effects are not considered. When **effCom** = 1, the commutating current effects are 100% considered.

ECEF Model

The ECEF model enables you to create an ECE model from Maxwell AC Magnetic solutions.

The ECEF model is used to set up a frequency sweep. To change the name of the model placed on the sheet, click the symbol on the sheet, and change the name of the component in the property window (Value field in the DeviceName line, with the Parameter Values tab selected).



ECEF_Model1

The ECEF_Model contains the following parameters:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- DeviceName –the name of the device model with default name ECEF_Model1.
- FreqSweeps – specifies the frequency sweep list. The format is: "v1, v2, (dv1, n1), v3, …", where v1, v2, v3 specify individual sweeping values, and (dv1, n1) specify a sweep range with dv1 being the increment, and n1 the number of increments. For example, a list of "50Hz, (10Hz, 5)" is equivalent to "50 Hz, 60 Hz, 70 Hz, 80 Hz, 90 Hz, 100 Hz".

An ECEF_Model cannot be used individually. It should be used combined with an ECEIM_ Model for induction motors.

At Maxwell run time, the circuit simulator assigns current, slip, as well as frequency if ECEF_ Model is added, to Maxwell AC Magnetic solver until all sweeps are finished. An ECE model is created automatically. You can import this ECE model in Twin Builder via the **Twin Builder** > **SubCircuit** > **Maxwell Component** > **Add Equivalent Circuit** menu command. For more information on coupling Maxwell designs to Twin Builder components, refer to the *Maxwell Equivalent Circuit Component* topic in the Twin Builder Help.

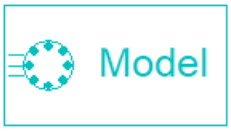
> **Note:** You can add an ECEF_Model to an existing external circuit without deleting the original circuit. In such cases, the original circuit is ignored and can be kept for future use. If you want to recover the original circuit for a simulation, instead of ECE model creation, you just need to deactivate each ECE related model by right-clicking the model and selecting Deactivate in the short-cut menu.

For more information on coupling Maxwell designs to Twin Builder components, refer to the *Coupling Maxwell to a Twin Builder Component* topic in the Maxwell Help.

ECEIM Model

The ECEIM model enables you to create an ECE model of a three-phase induction motor from Maxwell AC Magnetic solutions.

To change the name of the model placed on the sheet, click the symbol on the sheet, and change the name of the component in the property window (Value field in the DeviceName line, with the Parameter Values tab selected).



ECEIM_Model1

The ECEIM_Model contains the following parameters:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- DeviceName – the name of the device model with default name ECEIM_Model1.
- Windings – specifies the comma-separated name list of three-phase windings whose currents are to be swept.
- CurrentSweeps – specifies lists of sweep currents. The format is: “v1, v2, (dv1, n1), v3, …”, where v1, v2, v3 specify individual sweep values, and (dv1, n1) specify a sweeping range with dv1 being the increment, and n1 the number of increments. For example, a list of “-2.5 A, (0.5 A, 10)” is equivalent to "-2.5 A, -2 A, -1.5 A, -1 A, -0.5 A, 0, 0.5 A, 1 A, 1.5 A, 2 A, 2.5 A", and a list of "(0.5 A, 5)" is equivalent to “0, 0.5 A, 1 A, 1.5 A, 2 A, 2.5 A”.
- SkewAng – specifies the skew angle (in mechanical degrees) of the stator or rotor core for 2D only (available when Show Hidden is checked).
- Res – phase winding resistance (available when Show Hidden is checked).
- IndE – phase end-leakage inductance (available when Show Hidden is checked).

The ECEIM model is used to set up the sweeping of currents in three-phase windings. A slip sweep is added together with the current sweep. The slip sweep is fixed (not user-configurable) with a list of the following 13 values: 0.01, 0.02, 0.04, 0.1, 0.2, 0.3, 0.4, 0.5, 0.6, 0.7, 0.8, 0.9, 1.0. To simulate the real operation of induction motors at a specified slip with AC Magnetic solvers, the rotating rotor is treated as a locked rotor with equivalent rotor resistances and inductances.

An ECEIM_Model can be used individually to create the ECE model of a three-phase induction machine, or combined with the ECEF_Model for sweeping frequency.

At Maxwell run time, the circuit simulator assigns current, slip, as well as frequency if ECEF_ Model is added, to Maxwell AC Magnetic solver until all sweeps are finished. An ECE model is then created automatically. You can import this ECE model in Twin Builder via the **Twin Builder** > **SubCircuit** > **Maxwell Component** > **Add Equivalent Circuit** menu command. For more information on coupling Maxwell designs to Twin Builder components, refer to the *Maxwell Equivalent Circuit Component* topic in the Twin Builder Help.

> **Note:** You can add an ECEIM_Model to an existing external circuit without deleting the original circuit. In such cases, the original circuit is ignored and can be kept for future use. If you want to recover the original circuit for a simulation, instead of ECE model creation, you just need to deactivate each ECE related model by right-clicking the model and selecting Deactivate in the short-cut menu.

For more information on coupling Maxwell designs to Twin Builder components, refer to Coupling Maxwell to a Twin Builder Component.

Circuit and Parameters

Due to saturation and skin effects, the stator leakage inductance might be nonlinear, and the rotor leakage inductance will be nonlinear and slip dependent. The rotor leakage inductance can be decoupled into two parts, nonlinear only and slip dependent only, as explained below.

The stator and rotor nonlinear can be expressed as

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\begin{cases} L_1(I_1) = L_{10} + L_{1m}/k_{s1}(I_1) \\ L_2(I_1) = L_{20} + L_{2m}/k_{s2}(I_1) \end{cases}$$

where

$$\begin{cases} k_{s1}(I_1) = (I_1/I_{T1})/\left(1 - e^{-I_1/I_{T1}}\right) \\ k_{s2}(I_1) = (I_1/I_{T2})/\left(1 - e^{-I_1/I_{T2}}\right) \end{cases}$$

The slip-dependent rotor resistance and leakage inductance are

$$\begin{cases} R_{2s} = k_r R_{2s0} \\ L_{2s} = k_x L_{2s0} \end{cases}$$

where

$$\begin{cases} k_r(\xi) = \xi \frac{\sinh(2\xi)+\sin(2\xi)}{\cosh(2\xi)-\cos(2\xi)} \\ k_x(\xi) = \frac{3}{2\xi} \frac{\sinh(2\xi)-\sin(2\xi)}{\cosh(2\xi)-\cos(2\xi)} \end{cases}$$

with

$$\xi = \xi_0 \sqrt{s}$$

In the above equations, I_1 is the stator phase current and s is the rotor slip, L_{10}, L_{1m}, L_{20}, L_{2m}, I_{T1}, I_{T2}, R_{2s0}, L_{2s0} and ξ_0, as well as R_2 in the circuit, are ten constant parameters identified from the eddy-current FEA solutions at different stator currents and rotor slips. If the stator or rotor leakage inductance is constant, L_{1m} or L_{2m} is zero, and I_{1T} or L_{T2} is not used in such a case.

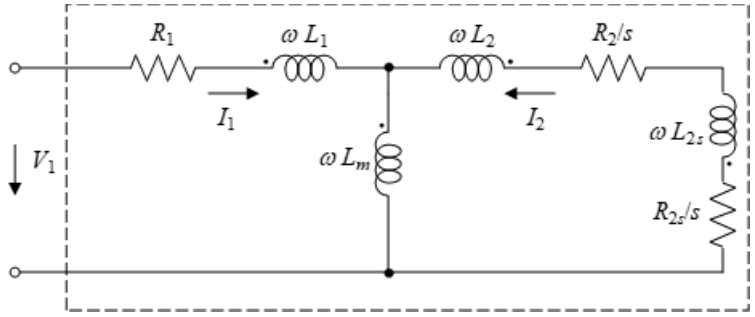
> **Note:** For information on how the constant parameters are calculated, see D. Lin, P. Zhou and Z. Zhang, "Reduced Order Modeling and Parameter Identification of Induction Motors Based on FEA Solutions," in IEEE Transactions on Energy Conversion, vol. 38, no. 2, pp. 1108-1117, June 2023.

The nonlinear main inductance in the circuit is expressed as

$$L_m = L_{m0}/k_{sm}$$

where, L_{m0} is a constant parameter, and k_{sm} is the saturation factor of main field which is expressed as a look-up table. Both L_{m0} and k_{sm} are identified from the AC Magnetic FEA solutions with different stator currents at no-load operation (with rotor slip being near zero).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Because R_{2s} and L_{2s} are slip dependent, the above circuit can be implemented in the frequency domain only. If the rotor bar is divided into two sections with top section ratio of k_1, and the bottom section ratio of k_2 =1 – k_1, the resistance and self-inductance of the top section will be

$$\begin{cases} R_{2t} = R_{2s0}/k_1 \\ L_{2t} = k_1 L_{2s0} \end{cases}$$

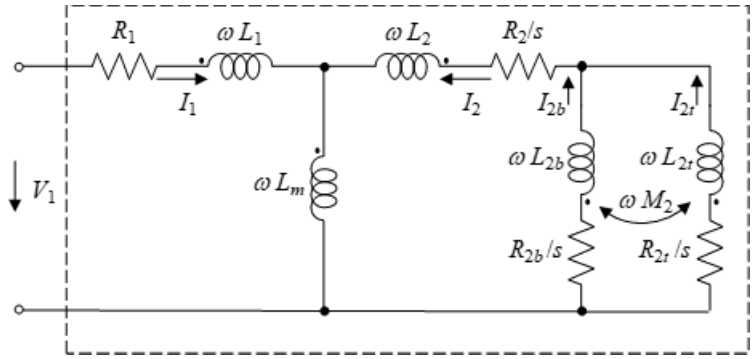
and the resistance and self-inductance of the bottom section are

$$\begin{cases} R_{2b} = R_{2s0}/k_2 \\ L_{2b} = k_2 L_{2s0} + 3k_1 L_{2s0} \end{cases}$$

The mutual inductance between the top and bottom sections is

$$M_2 = \frac{3}{2} k_1 L_{2s0}$$

The circuit with distributed rotor sections is shown in the following figure:



After all parameters in the first circuit are identified, the top section ratio k_1 is identified by minimizing the error of the impedance between the two circuits. All identified parameters are saved in file **ece_table.txt**, as shown in the figure below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
B_GeneralData
 Poles   4
 Phases 3
 Frequency        50
E_GeneralData

B_UnsatParam
 r1       4.27905
 l1e      0.00414949
 l10      0.00653594
 l1m      0
 it1      10
 r20      1.06698
 l20      0.0199702
 l2m      0.347714
 it2      0.310086
 r2s0     0.272791
 l2s0     0.00449169
 kxi0     2.78555
 k1       0.309297
 lm0      0.632328
 l00      0.0397237
 skew     0
 err      0.0319919
 use_tanh         0
E_UnsatParam

B_SatFactorForXm0_im_ksm 33
 0        1
 0.15625          1
 0.3125           1.00013
 ......
E_SatFactorForXm0_im_ksm
```

In above figure, r_1, $l1_e$, and *skew* are directly obtained from ECEIM_Model inputs Res, IndE, and SkewAng, respectively, 10 parameters, l10, l1m, it1, r20, l20, l2m, it2, r2s0, l2s0, kxi0 (ξ0), and lm0, as well as the look-up table between B_SatFactorForXm0_im_ksm and E_ SatFactorForXm0_im_ksm, are identified based on the first circuit, and k1 is identified based on the second circuit. Parameter l00 is the zero-sequence inductance. The err value represents the relative error of impedances between the FEA solutions and the parameter-derived results. If the value of use_tanh is 1, the saturation factors for the stator and rotor leakage inductances will be expressed as

$$\begin{cases} k_{s1}\left(I_1\right) = \left(I_1/I_{T1}\right)/tanh(I_1/I_{T1}) \\ k_{s2}\left(I_1\right) = \left(I_1/I_{T2}\right)/tanh(I_1/I_{T1}) \end{cases}$$

to achieve smaller identification error, where tanh() is a hyperbolic tangent function.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Core Loss Parameters

Core losses can be computed for each sweep in eddy-current solver. To include core loss computation, users must check the **Core Loss** setting in the **Excitations/Set Core Loss** panel. Since core losses are computed in a post-process, including core loss computation will not affect the parameter identification of the induction machine circuit.

The stator and/or rotor core losses can be obtained from lamination steel or power ferrite material. For lamination steel material, the stator core loss can be computed from the phase circuit variables and expressed as

$$P_{fe1} = k_{hs}{E_m}^2/f + k_{cs}{E_m}^2 + k_{es}{E_m}^{1.5}$$

where k_{hs}, k_{cs}, and k_{es} are to-be-determined parameters, E_m is the phase induced voltage magnitude, and f is the frequency. For power ferrite material, core loss is expressed as

$$P_{fe1} = C_m{E_m}^A f^B$$

where C_m, A, and B are to-be-determined parameters. The three parameters for either material can be identified at different induced voltage magnitudes and different frequencies by minimizing the error of core losses between the FEA solutions and the results obtained from the above equations. One of the above two equations will be automatically selected according to their identified errors. If the identified error based on the first equation is smaller than that based on the second equation, the first equation for lamination steel material will be selected. Otherwise, the second equation for power ferrite material will be selected. The core loss resistance is derived from core loss, and for lamination steel material, it is given by

$$R_{fe1} = \frac{{E_m}^2/2}{P_{fe1}/3} = \frac{1.5}{k_{hs}/f + k_{cs} + k_{es}/{E_m}^{0.5}}$$

For power ferrite material, core loss resistance is given by

$$R_{fe1} = \frac{{E_m}^2/2}{P_{fe1}/3} = \frac{1.5}{C_m{E_m}^{A-2}f^B}$$

Replacing E_m by sE_m and f by sf, where s is the rotor slip, we can calculate rotor core loss in the same way as the stator core loss. Then the rotor core loss resistance for lamination steel material is

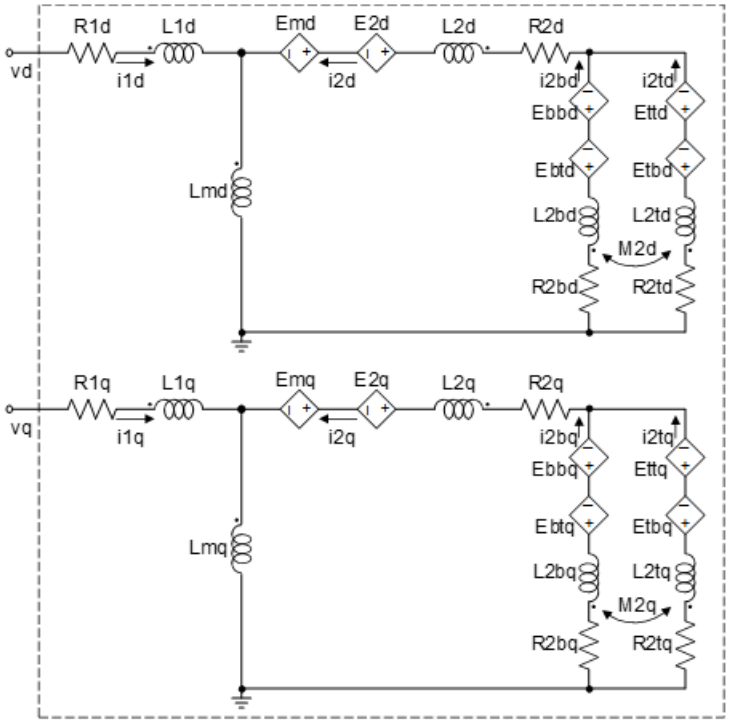
$$R_{fe2} = \frac{{E_m}^2/2}{P_{fe2}/3} = \frac{1.5}{sk_{hs}/f + s^2k_{cs} + s^{1.5}k_{es}/{E_m}^{0.5}}$$

and the rotor core loss resistance for power ferrite material is

$$R_{fe2} = \frac{{E_m}^2/2}{P_{fe2}/3} = \frac{1.5}{C_m{E_m}^{A-2}f^B s^{A+B}}$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Core loss resistance is frequency-dependent. In a time-domain circuit, as shown in the following figure, what we can measure are time-instant values. In order to use frequency-dependent resistance in a time-domain circuit, we need to calculate the frequency based on measurable time-instant values.



By measuring the induced voltages across Lmd and Lmq at time t, noted as Vmd and Vmq, respectively, we can compute the magnitude of induced voltage as

$$V_m = \sqrt{V_{md}{}^2 + V_{md}{}^2}$$

Similarly, by measuring the flus linkages of Lmd and Lmq at time t, noted as λmd and λmq, respectively, we can compute the magnitude of flux linkage as

$$\lambda_m = \sqrt{\lambda_{md}{}^2 + \lambda_{mq}{}^2}$$

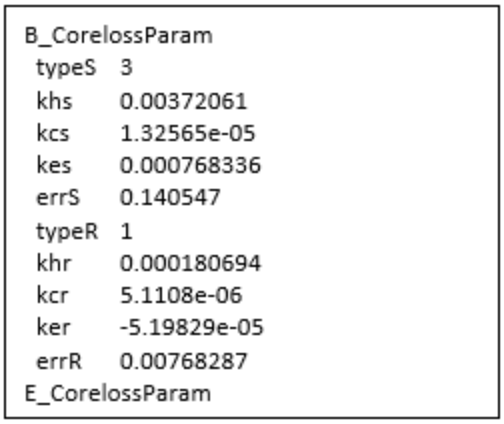
The frequency then can be obtained from

$$f = \frac{V_m}{2\pi\lambda_m}$$

and the induced voltage magnitude in three-phase side E_m for core loss resistance computation can be obtained by V_m with a factor of two-phase to three-phase transformation. The slip s for the computation of rotor core loss resistance can be obtained from the number of poles, rotor speed, and frequency.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The identified core loss parameters are listed between B_CorelossParam and E_ CorelossParam in **ece_table.txt** file, as shown below.



In the above figure, parameters typeS, khs, kcs, kes, and errS are for stator core loss. When typeS is 0, the circuit model will not include the stator core loss. When typeS is 1 or 3, the parameters are lamination-steel-based, and when it is 2 or 4, the parameters khs, kcs, and kes represent Cm, A, and B, respectively, for power ferrite material. When typeS is 1 or 2, two stator core loss resistances are connected in parallel with Lmd and Lmq, otherwise, when it is 3 or 4, two core loss resistances are connected from the input nodes of L1d and L1q to the ground. The errS value is the regression error for the stator core loss. Similarly, parameters typeR, khr, kcr, ker, and errR are for rotor core loss with the same definition as the parameters for stator core loss.

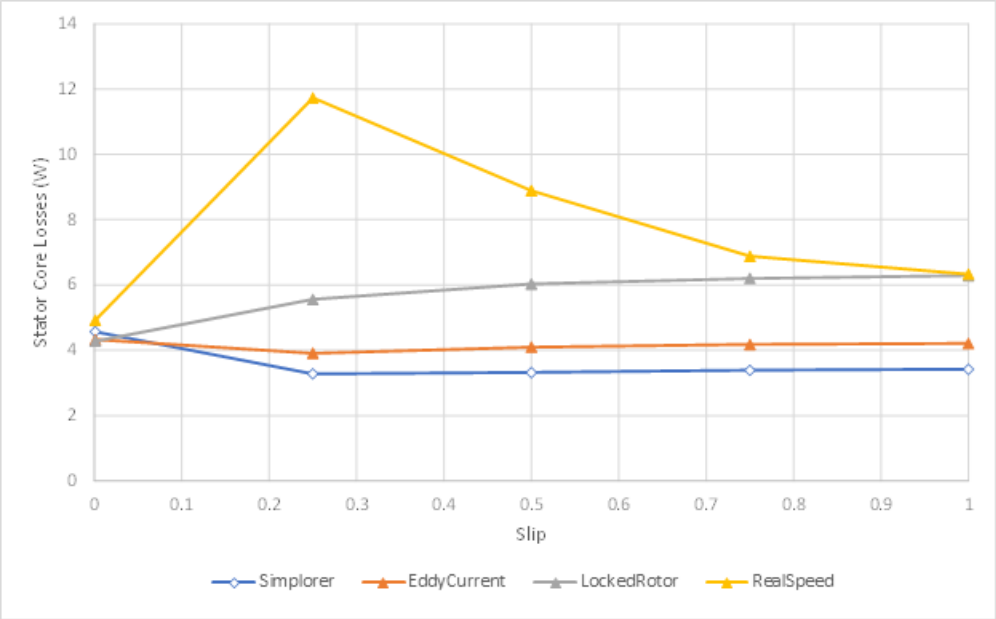
Because core loss parameters are identified from Maxwell eddy-current solutions at no-load operation based on the method referring the rotor frequency from the real operation condition to a locked-rotor operation (called locked-rotor method), the predicted core loss from the identified parameters will be under-estimated due to the following approximation errors:

1. The error caused by field difference between no-load and load operations. At no-load operation, the field in an induction machine is mainly the main flux which links both the stator and rotor windings, while at load operation, the main fluxes created by the stator and rotor currents will mostly offset each other, remaining large leakage fluxes which will go through the stator and rotor tooth tips. In the stator, the leakage flux will add to the main flux, causing the stator core loss increasing. In the rotor, the leakage flux will partially cancel the main flux, causing the rotor core loss decreasing a little.
2. The error caused by difference between sinusoidal (eddy-current solver) and non-sinusoidal (transient solver) waveforms. In eddy-current solver, core loss is computed based on pure sinusoidal waveforms, but in transient solver, core loss is computed based on real non-sinusoidal waveforms due to nonlinearity. The harmonic field components will cause additional core losses in both stator and rotor cores in transient analysis.

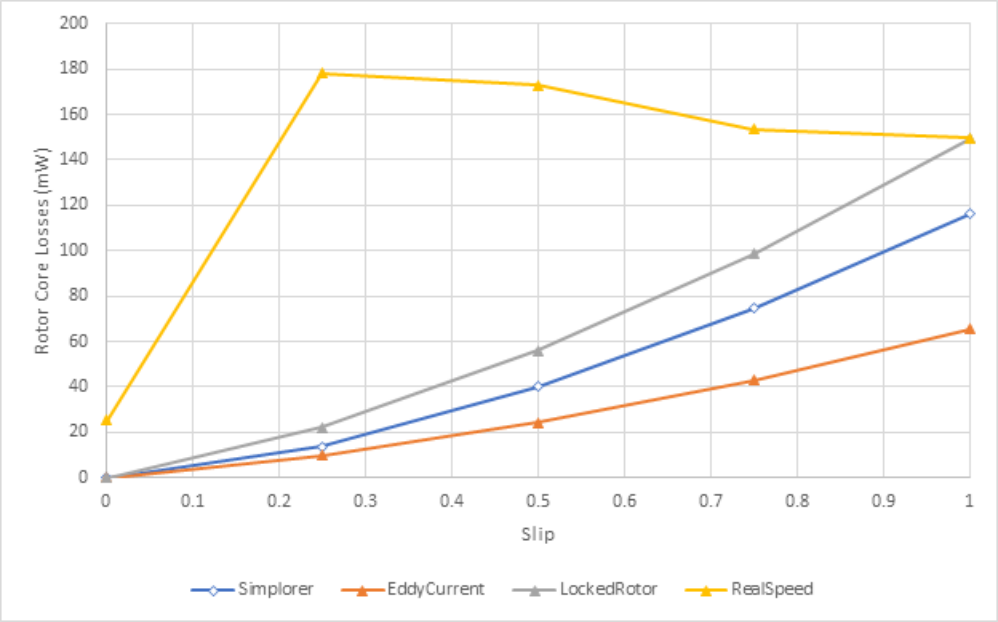
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. The error caused by difference between locked-rotor and rotating operations. When rotor is rotating, the slot effects will cause additional core losses at the surfaces of the stator and rotor cores known as surface losses, as well as additional core loss at teeth of the stator and rotor cores known as pulsation losses.

The above three loss errors can be illustrated by the following two figures for stator and rotor core losses, respectively. In the figures, curves labeled with “Simplorer” are obtained from the system simulation based on the extracted core loss parameters, curves labeled with “EddyCurrent” are obtained from the eddy-current simulation based on the locked-rotor method, curves labeled with “LockedRotor” are obtained from the transient simulation based on the locked-rotor method, and curves labeled with “RealSpeed” are obtained from the transient simulation with real rotor speed. The loss difference between curves “EddyCurrent” and “Simplorer” represents the error described in (1), the loss difference between curves “LockedRotor” and “EddyCurrent” indicates the error described in (2), and the loss difference between curves “RealSpeed” and “LockedRotor” is the error described in (3), which will increase as the stator currents increase, and will decrease as the rotor speed decreases.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Therefore, compare to Maxwell transient analysis, the core loss obtained from the circuit simulation will be under-estimated, especially at load operation. To adjust the simulated core loss, we introduce an argument KPfe in SIMPLORER (Twin Builder) circuit model to scale core losses for both stator and rotor. The default value of KPfe is 1.0. The reasonable value of KPfe at full-load operation may range from 1.5 to 2.5.

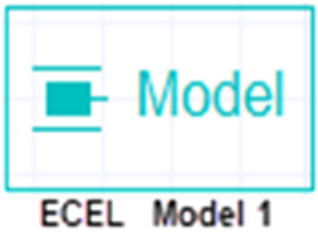
Since the stator and rotor core loss resistances are nonlinear and frequency dependent, we allow users to consider or not consider core loss effects on circuit variables to avoid possible divergence during circuit simulation. When we input a positive value for KPfe, we will consider core loss effects by connecting the core loss resistances with derived values. When the input value for KPfe is negative, the core loss resistances are disconnected.

No matter core loss effects are considered or not, the stator and/or rotor core losses are output via circuit arguments Pfe1 and/or Pfe2 in the SIMPLORER (Twin Builder) circuit model.

ECEL Model

The ECEL model is used to set up the sweeping of mover position of a translational motion. To change the name of the model placed on the sheet, click the symbol on the sheet, and change the name of the component in the property window (Value field in the DeviceName line, with the **Parameter Values** tab selected).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The ECEL_Model contains the following parameters:

- DeviceName − the name of the device model with the default name as ECEL_Model1;
- MotionMax − the maximum motion range for sweeping;
- MotionIntervals − the number of uniform sweeping intervals for motion range;
- MotionType − the motion type: 0 for limited motion; 1: for periodic motion with full sweeping; 2 or more: for periodic motion with half or less sweeping.
- MotionDir − the motion direction in (x, y, z);

The initial position set in Maxwell designs is always referred to as 0 position in ECE model. When MotionType ≥ 1 (for periodic motions), the number of sweepings is equal to MotionIntervals. In such a case, the last point at MotionMax is not swept and is automatically extended from the first sweeping point at zero position based on the periodic conditions. When MotionType = 0 (for limited motion), the number of sweepings is equal to MotionIntervals + 1. In such a case, the motion limit is always ranged from 0 to MotionMax in the ECE model.

An ECEL_Model cannot be used individually. It should be used combined with an ECEW_Model and/or an ECE3_Model for rotational electrical machines.

For ECE model creation, the setup of Stop Time and Timestep are ignored. At Maxwell run time, the circuit simulator assigns currents for all windings step-by-step based on the lists of current and position sweeps until all sweeps are finished. An ECE model is created automatically. You can import this ECE model in Twin Builder via the **Twin Builder** > **SubCircuit** > **/Maxwell Component** > **Add Equivalent Circuit** menu command. For more information on coupling Maxwell designs to Twin Builder components, refer to the *Maxwell Equivalent Circuit Component* topic in the Twin Builder Help.

> **Note:** You can add an ECEL_Model to an existing external circuit without deleting the original circuit. In such cases, the original circuit is ignored and can be kept for future use. If you want to recover the original circuit for a transient simulation, instead of ECE model creation, you just need to deactivate each ECE related model by right-clicking the model and selecting Deactivate in the short-cut menu.

For more information on coupling Maxwell designs to Twin Builder components, refer to Coupling Maxwell to a Twin Builder Component.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ECER Model

The ECER model enables you to create an ECE model with rotational motion from Maxwell Transient solutions. Compared with the procedure for creating ECE model from Maxwell Magnetostatic solutions, creating an ECE model directly from Maxwell Transient solutions takes advantage of winding and motion setups in the Maxwell Transient setup.

The ECER model is used to set up the sweeping of rotor position. To change the name of the model placed on the sheet, click the symbol on the sheet, and change the name of the component in the property window (Value field in the DeviceName line, with the Parameter Values tab selected).

The ECER_Model contains the following parameters:

- **DeviceName** – the name of the device model with default name ECER_Model1.
- **RotAngMax** – specifies the maximum rotor angle (by default, in mechanical degrees) for sweeping the rotor position.

  > **Note:**
  >
  > When combining the ECER component with an ECE3 model, the extraction method for ECE models can automatically take advantage of the windings' symmetry. The lookup tables will automatically be extended during the ECE extraction process.
  >
  > For most cases, RotAngMax can be defined as 120° (in electrical degrees) or 120°/(Number of Pole Pairs) (in mechanical degrees). Depending on the machine design, a larger value might be required or a further reduction can be applied as discussed in Rosu *et al*, Chapter 2.5.2[1].

- **InElecDeg** – check to define **RotAngMax** in electrical degrees; uncheck to define it in mechanical degrees.
- **RotAngIntervals** – specifies the number of uniform sweep intervals for rotating angle.
- **SkewAng** – specifies the skew angle (in mechanical degrees) of the stator or rotor core. This input only affects 2D models and is ignored otherwise. By specifying this input, the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

lookup tables derived from the FE model are manipulated analytically in order to consider the effect of skewing on flux and torque.

Considering skewing of an arbitrary look-up table quantity *f(θ)*, the look-up table output can be calculated from the corresponding value *f '(θ)* for the unskewed case as

$$f(\theta) = \frac{1}{\delta}\int_{\theta-\frac{\delta}{2}}^{\theta+\frac{\delta}{2}} f'(\theta)d\theta$$

where δ is the specified skew angle, and θ denotes the rotor position dimension in the look-up-table.

Strictly speaking, this analytical manipulation only applies to continuous skewing. When using this option, the Maxwell 2D transient model must not have **Use Skew Model** activated in the Design Settings window (*i.e.*, the FEA model of the machine defines an unskewed machine).

- **Poles** – specifies the number of poles of the machine.
- **Slots** – specifies the number of stator slots to determine cogging torque period (available when **Show Hidden** is checked). If this input is provided, the cogging torque period of the machine will be calculated (in mechanical degrees) as 360°/LeastCommonMultiple(Poles, Slots). Then, for the combination id=iq=0, the input **RotAngIntervals** will be overwritten by a value corresponding to the cogging torque period divided by 18.
- **Output** – specifies the type of lookup table outputs for the winding flux linkages: **0** for flux linkages in ABC (*i.e.*, per phase flux linkages); **1** for flux linkages in DQ0 (*i.e.*, d-, q- and 0 components of flux linkages in a rotating reference frame); and **2** for lookup tables of d- and q- axis inductances(**Ld**, **Lq**) and flux linkage caused by rotor excitation (**Fm**) (available when **Show Hidden** is checked).
  If **Output**=2, the generated ECE model will be a "fundamental flux model." The values of **Ld**, **Lq**, and **Fm** in the lookup tables represent the average value computed across the swept rotor angle.

An ECER_Model cannot be used individually. It should be used combined with an ECEW_Model and/or an ECE3_Model for rotational electrical machines. When DQFluxOutput is checked, it must be combined with an ECE3_Model.

For ECE model creation, the setup of Stop Time and Timestep are ignored. At Maxwell run time, the circuit simulator assigns currents for all windings step-by-step based on the lists of current sweeps until all sweeps are finished. An ECE model is created automatically. You can import this ECE model in Twin Builder via the **Twin Builder** > **SubCircuit** > **Maxwell Component** > **Add Equivalent Circuit** menu command. For more information on coupling Maxwell designs to Twin Builder components, refer to the *Maxwell Equivalent Circuit Component* topic in the Twin Builder Help.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: You can add an ECER_Model to an existing external circuit without deleting the original circuit. In such cases, the original circuit is ignored and can be kept for future use. If you want to recover the original circuit for a transient simulation, instead of ECE model creation, you just need to deactivate each ECE related model by right-clicking the model and selecting **Deactivate** in the short-cut menu.

For more information on coupling Maxwell designs to Twin Builder components, refer to Coupling Maxwell to a Twin Builder Component.

---

1. *Multiphysics Simulation by Design for Electrical Machines, Power Electronics, and Drives*, First Edition. Marius Rosu, Ping Zhou, Dinsheng Lin, Dan Ionel, Mircea Popescu, Frede Blaabjerg, Vandana Rallabandi, and David Staton. Copyright 2017, Institute of Electrical and Electronics Engineers, Inc. Published 2017 by John Wiley and Sons, Inc.

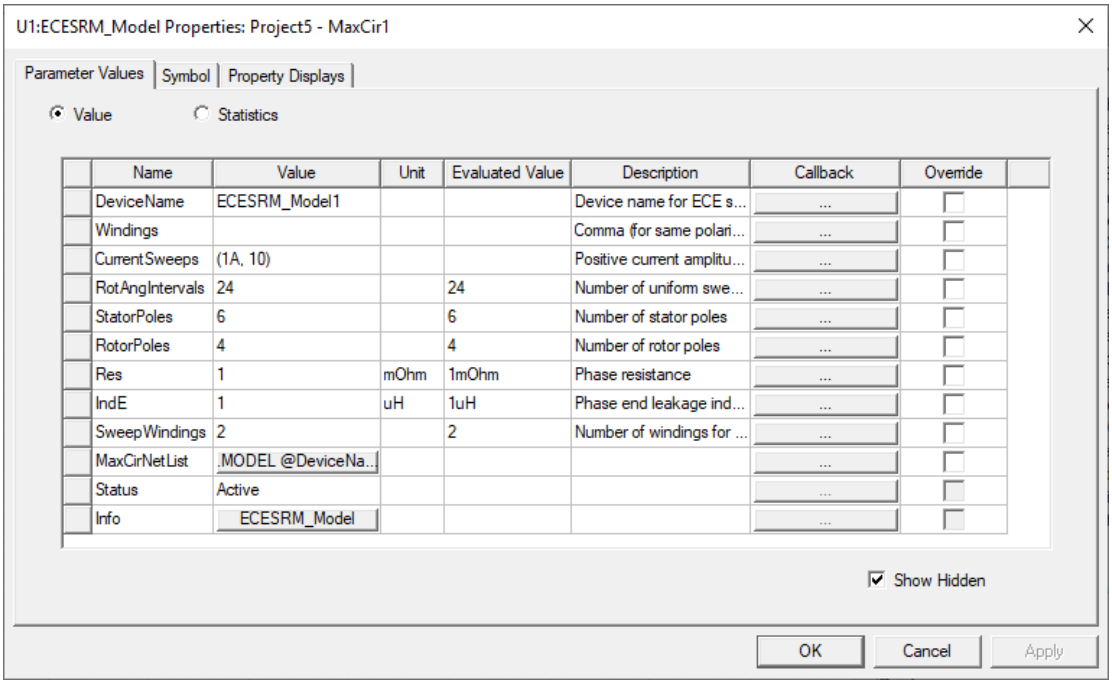
ECESRM Model

The ECESRM model enables you to create an ECE model of a switched reluctance motor from Maxwell 2D and 3D Transient solutions.

To change the name of the model placed on the sheet, click the symbol on the sheet, and change the name of the component in the property window (Value field in the DeviceName line, with the Parameter Values tab selected).

ECESRM_Model1

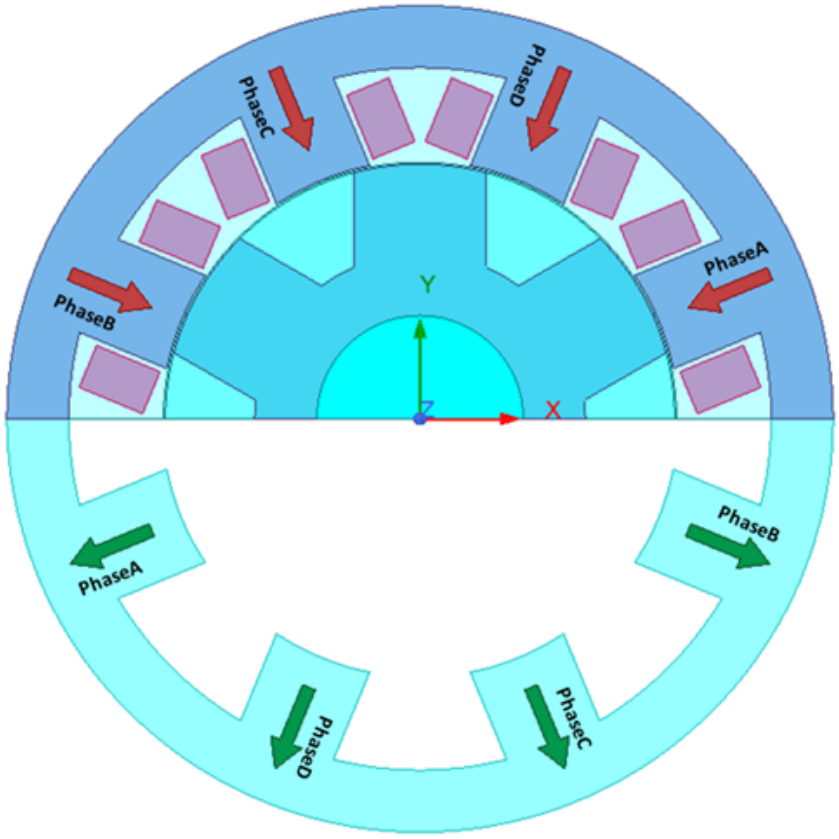
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The ECESRM_Model contains the following parameters:

- DeviceName – the name of the device model with default name ECESRM_Model1.
- Windings – specifies winding names in the order of arrangement. Windings are separated by commas or semicolons. If two adjacent windings are defined with the same polarity, they will be separated by a comma “,”, otherwise by a semicolon “;”. For example, for an 8/6 pole SRM, as shown below, windings for phase A, B, C and D are arranged in clockwise direction. The adjacent two windings of phase A and phase B have different polarities, and all other two adjacent windings have the same polarities. Therefore, windings can be specified in ECESRM_Model as:“PhaseA; PhaseB, PhaseC, PhaseD”where the separator between PhaseA and PhaseB is “;”, indicating different polarities between them. The last separator between PhaseD and PhaseA is not required because the total number of semicolons should be an odd number, and the last separator can be automatically determined.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- CurrentSweeps – specifies current sweeps in a winding. The first winding in the winding list is always excited with sweeping current. You just need to list 0 and all positive sweeping currents. All negative currents will be swept, or extended, symmetrically from the positive sweeping currents.
- RotAngIntervals – specifies the number of uniform sweeping intervals for rotor position in one rotor-pole region. For a 6-pole rotor, one rotor-pole region will be 60 mechanical degrees. If RotAngIntervals = 24, the sweeping step for rotor position will be 2.5 mechanical degrees.
- StatorPoles and RotorPoles – specifies the numbers of stator and rotor poles.
- Res and IndE – specifies winding resistance and end-leakage inductance for 2D.
- SweepWindings – specifies the number of windings to be excited. The value of SweepWindings could be 0, 1, or 2. Value 0 is for the case that only one winding current is swept, and all mutual effects of other winding currents are ignored; value 1 is for the case that only one winding current is swept, and the mutual effects on winding flux linkage are considered, but the mutual effects of other windings currents on field saturation and torque are ignored; value 2 is for the case that two winding currents are swept, and all mutual effects are considered.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

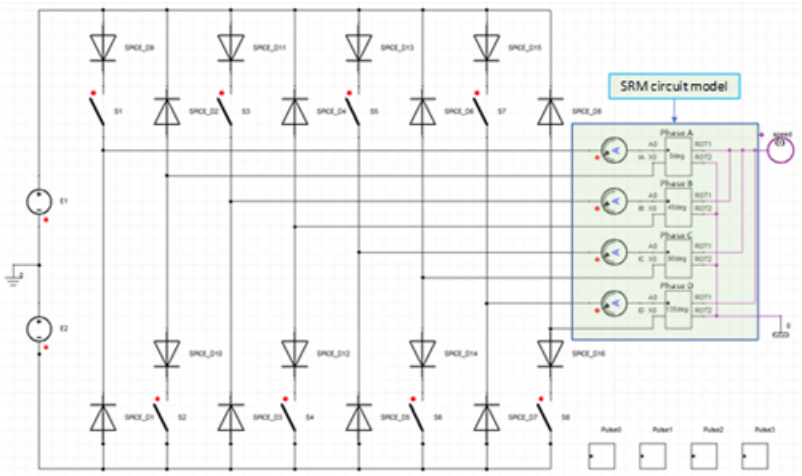
Look-Up Table Processing

For an SRM, if a winding is triggered after the previously triggered winding has finished the free-wheeling process, there will be no mutual effects between the currently triggered winding and the previously triggered winding. In such a case, only one winding is excited with current at any time. In many cases, in order to increase power capacity, a winding could be triggered when the previous triggered winding is still working. In this situation, two windings could be excited with currents in some time periods. Therefore, we will provide different sweep types for different applications, which is specified by parameter SweepWindings, as described below.

- **SweepWindings = 0**

  SweepWindings = 0 means only one winding is excited, and all mutual effects between windings are ignored. The first listed winding PhaseA is excited with sweeping currents in the range of [0, Im], and the look-up table for the negative currents is automatically extended based on symmetry conditions. The final circuit model derived from this setting is equivalent to the circuit which is manually created by:

  i. Obtaining a single-phase circuit submodel using ECEW_Model and ECER_Model.
  ii. connecting single-phase circuit submodels for all phases with suitable IniPos settings, as shown in the circuit inside the shaded rectangle below.



This sweep type has the smallest look-up table, which includes controlling variables of one winding current and rotor position, and controlled variables of flux linkage and torque. For this sweep type, it is not necessary to solve the field for negative sweeping currents. The look-up table for the negative currents can be automatically extended based on symmetry conditions, as given below:

$$\begin{cases} F_{this}\left(-I_{this},\theta\right) = -F_{this}\left(I_{this},\theta\right) \\ T\left(-I_{this},\theta\right) = T\left(I_{this},\theta\right) \end{cases}$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where $F_{this}(I_{this}, \theta)$ and $T(I_{this}, \theta)$, as outputs of look-up table, are the flux linkage of the excited winding and rotor torque, respectively, and the controlling variables I_{this} and θ are the winding current and rotor position, respectively.

- **SweepWindings = 1**

  SweepWindings = 1 means only one winding is excited, and the mutual effects on winding flux linkage are considered, but the mutual effects on field saturation and torque are ignored. In order to easily illustrate windings for look-up table creation, we call the excited winding, PhaseA, as *this* winding, PhaseB as *next* winding, and PhaseD as *previous* winding. In this way, in the first single-phase submodel as shown in the figure above, this/next/previous windings represent phase A/B/D windings. In the second single-phase submodel, they represent phase B/C/A windings, and so on. Only *this* winding has a circuit inside a single-phase submodel. The mutual effects of the next/previous currents on this-winding flux linkage are considered by

$$F = F_{this}\left(I_{this}, \theta\right) \pm I_{next} \cdot M_{next}\left(I_{this}, \theta\right) \pm I_{pre} \cdot M_{pre}\left(I_{this}, \theta\right)$$

  where $F_{this}(I_{this}, \theta)$, $M_{next}(I_{this}, \theta)$, $M_{pre}(I_{this}, \theta)$, as outputs of look-up table, are flux linkage, mutual inductance between *this* and *next* windings, and mutual inductance between *this* and *pre*vious windings, respectively. Controlling variables I_{this} and θ represent *this*-winding current and rotor position, respectively. The *next* and *previous* winding currents, $I_{next}$and I_{pre}, respectively, are delivered via parameters of single-phase submodels. The signs before the *next* and *pre*vious winding currents will be automatically determined in the final SRM model based on the separators for *pre*vious, *this* and *next* windings. Like the case of SweepWindings = 0, the look-up table for the negative currents can be automatically extended based on symmetry conditions, as given below:

$$\begin{cases} F_{this}\left(-I_{this}, \theta\right) = -F_{this}\left(I_{this}, \theta\right) \\ M_{next}\left(-I_{this}, \theta\right) = M_{next}\left(I_{this}, \theta\right) \\ M_{pre}\left(-I_{this}, \theta\right) = M_{pre}\left(I_{this}, \theta\right) \\ T\left(-I_{this}, \theta\right) = T\left(I_{this}, \theta\right) \end{cases}$$

  Comparing this with the sweep type of SweepWindings = 0, this sweep type has the same controlling variables with the same sweeping points, but it has a larger-size lookup table because it has more controlled variables.

- **SweepWindings = 2**

  SweepWindings = 2 means two windings, *this* and *next* windings, are excited, therefore all mutual effects on winding flux linkage, field saturation and torque are considered. If CurrentSweep in the ECESRM_Model is specified in the range of [0, Im], the *this*-winding current will be swept in the range of [-Im, Im], and the *next*-winding current will be swept in the range of [0, Im]. After field simulations for all sweeps are completed, we obtain a clean look-up table. This look-up table has two controlled variables (*this*-winding flux linkage F_{this} and torque T); and three controlling variables (*this*-winding current I_{this}, *next*-winding

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

current I_{next}, and rotor position θ). In this clean look-up table, T is the torque produced by *this*-winding current but considered the mutual effects of *next*-winding current. In field analysis, a torque can be derived based on field solutions and be expressed, by numerical integration over all related mesh elements, as

$$T_m = \sum_{i=1}^{n} k_{Ti} \left(\mathbf{B}_i \cdot \mathbf{B}_i \right)$$

where $\mathbf{B_i}$ is the flux-density vector created by all winding currents, and k_{Ti} is a coefficient, in mesh element *i*. If we freeze the permeabilities of all nonlinear materials, the field becomes linear. Then, the flux density vector in any mesh elements can be expressed as

$$\mathbf{B}_i = \mathbf{B}_{1i} + \mathbf{B}_{2i}$$

where B_{1i} and B_{2i} are the flux-density vectors created by this and next winding currents, respectively. Hence, the torque will be

$$T_m = \sum_{i=1}^{n} k_{Ti} \left(\mathbf{B}_{1i} + \mathbf{B}_{2i} \right) \cdot \left(\mathbf{B}_{1i} + \mathbf{B}_{2i} \right) = T_{11} + 2T_{12} + T_{22}$$

with

$$\begin{cases} T_{11} = \sum_{i=1}^{n} k_{Ti} \left(\mathbf{B}_{1i} \cdot \mathbf{B}_{1i} \right) \\ T_{12} = \sum_{i=1}^{n} k_{Ti} \left(\mathbf{B}_{1i} \cdot \mathbf{B}_{2i} \right) \\ T_{22} = \sum_{i=1}^{n} k_{Ti} \left(\mathbf{B}_{2i} \cdot \mathbf{B}_{2i} \right) \end{cases}$$

In field analysis, we can get T_{11} by injecting current in this winding only, and we get T_{22} by infecting current in next winding only. Then we get

$$T_{12} = (T_m - T_{11} - T_{22})/2$$

Finally, we get torque in the clean look-up table as $T = T_{11} + T_{12}$.

The clean look-up table can be extended to a full look-up table based on even and odd symmetry conditions, as given below:

$$\begin{cases} F_{this} \left(I_{this}, -I_{next}, \theta \right) = -F_{this} \left(-I_{this}, I_{next}, \theta \right) \\ T \left(I_{this}, -I_{next}, \theta \right) = T \left(-I_{this}, I_{next}, \theta \right) \end{cases}$$

The flux linkage and torque in the look-up table are derived based on this-winding current with considering the mutual effects of next-winding current. In real operation with two excited windings, we need to get the flux linkage and torque produced by each winding current. For example, in the circuit diagram above, if phases A and B are excited simultaneously, the flux linkage and torque, produced by phase-A current with mutual effects of phase-B current, can be directly obtained from the look-up table because phase-

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A current is *this*-winding current and phase-B current is *next*-winding current, as given below:

$$\begin{cases} F_1 = F_{this}\left(I_{this}, \pm I_{next}, \theta\right) \\ T_1 = T\left(I_{this}, \pm I_{next}, \theta\right) \end{cases}$$

where the sign before I_{next} is positive if the separator between *this* and *next* windings is the same as that between the first and second windings listed in the Windings field in the properties dialog box above, or negative otherwise. Because the lookup table includes the mutual-effect information, it is important to transfer the *next*-winding current to make sure the polarity relationship between *this* and *next* windings in a single-phase sub-model is the same as that for lookup table creation. For example, for the phase-A sub-model in the circuit diagram above, this sign is positive because the separator between *this* and *next* windings, or between PhaseA and PhaseB, is ";", which is the same as that between PhaseA and PhaseB for lookup table creation. However, for the phase-B sub-model, the sign is negative because the separator between *this* and *next* windings, or between PhaseB and PhaseC, is ",", which is different from the separator ";" between PhaseA and PhaseB for look-up table creation.

When we consider the flux linkage and torque produced by phase-B current with mutual effects of phase-A current, we cannot obtain them directly from the look-up table because in this situation phase-B current is *this*-wingding current and phase-A current is *pre*-winding current, but we can still obtain them from the look-up table based on symmetry conditions. In the winding polarities diagram above, if we mirror the geometry about the center line of the phase-B pole, then phase-A winding aligns with phase-C winding, and rotor position becomes negative. Based on this symmetry condition, we have

$$\begin{cases} F_2 = F_{this}\left(I_{this}, \pm I_{pre}, -\theta\right) \\ T_2 = -T\left(I_{this}, \pm I_{pre}, -\theta\right) \end{cases}$$

where the sign before I_{pre} is positive if the separator between *this* and previous windings is the same as that between the first and second windings listed in the Windings field in the properties dialog box above, or negative otherwise.

In the final SRM model, the signs before *Inext* and Ipre for all single-phase sub-models will be automatically determined based on all separators listed in the Windings field, which, together with I_{next} and I_{pre}, are forwarded to submodels via input parameters.

You can import this ECE model in Twin Builder via the **Twin Builder** > **SubCircuit** > **Maxwell Component** > **Add Equivalent Circuit** menu command. For more information on coupling Maxwell designs to Twin Builder components, refer to the *Maxwell Equivalent Circuit Component* topic in the Twin Builder Help.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** You can add an ECESRM_Model to an existing external circuit without deleting the original circuit. In such cases, the original circuit is ignored and can be kept for future use. If you want to recover the original circuit for a simulation, instead of ECE model creation, you just need to deactivate each ECE related model by right-clicking the model and selecting Deactivate in the short-cut menu.

For more information on coupling Maxwell designs to Twin Builder components, refer to the *Coupling Maxwell to a Twin Builder Component* topic in the Maxwell Help.

ECET Model

The ECET model is used to set up the sweeping of the magnetizing current for each phase of a single- or poly-phase transformer. To change the name of the model placed on the sheet, click the symbol on the sheet, and change the name of the component in the property window (Value field in the DeviceName line, with the Parameter Values tab selected).

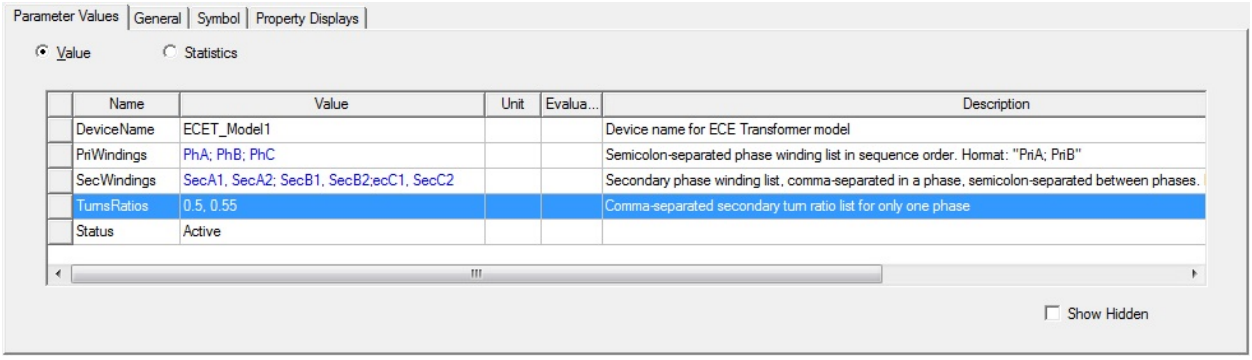
ECET_Model1

The ECET_Model contains the following parameters:

- **DeviceName** − the name of the device model with the default name as ECET_Model1;
- **PriWindings** − the name list of all primary windings with semicolons for separators between phase;
- **SecWindings** − the name list of all secondary windings with semicolons for separators between phases;
- **TurnsRatios** − the ratio of the secondary turns to the primary turns for only one phase.

If a transformer has two or more secondary windings for one phase, the secondary winding name list is separated by commas between winding names for the same phase, and TurnsRatios is a value list, instead of a single value, which is separated by commas between values for one phase. For example, for a three-phase transformer with two secondary windings for each phase, the parameters for ECET_Model are set as

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



An ECET_Model cannot be used individually. It should be used combined with an ECEW_Model and/or an ECE3_Model.

In an ECE model look-up table, only the magnetizing inductance for each phase is assumed to be nonlinear, and all leakage inductances for all primary and secondary windings are assumed to be constant. Therefore, only the magnetizing current of the primary winding for each phase is required to be swept, which reduces the number of sweeping points significantly.

For ECE model creation, the setup of Stop Time and Time step are ignored. At Maxwell run time, the circuit simulator assigns currents for all windings step-by-step based on the list of current sweeps until all sweeps are finished. An ECE model is created automatically.

You can import this ECE model in Twin Builder via the **Twin Builder** > **SubCircuit** > **Maxwell Component** > **Add Equivalent Circuit** menu command. When the ECET_model is created and then imported into Twin Builder the following additional parameters are made available for use in Twin Builder:

- **r1*_[WindingName]*** is winding resistance (default = 0)
- **l1*_[WindingName]*** is end leakage inductance (default = 1 mOhm, for 3D always = 0)
- **i0*_[WindingName]*** is initial current (default = 0)

For more information on coupling Maxwell equivalent circuit designs to Twin Builder components, refer to the *Maxwell Equivalent Circuit Component* topic in the Twin Builder Help.

> **Note:** You can add an ECET_Model to an existing external circuit without deleting the original circuit. In such cases, the original circuit is ignored and can be kept for future use. If you want to recover the original circuit for a transient simulation, instead of ECE model creation, you just need to deactivate each ECE related model by right-clicking the model and selecting Deactivate in the short-cut menu.

For more information on coupling Maxwell designs to Twin Builder components, refer to Coupling Maxwell to a Twin Builder Component.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ECEW Model

The ECEW model enables you to create an ECE model from Maxwell Transient solutions. Compared with the procedure for creating ECE model from Maxwell Magnetostatic solutions, creating an ECE model directly from Maxwell Transient solutions takes advantage of winding and motion setups in Maxwell Transient.

The ECEW model is used to set up the sweeping of one winding current. To change the name of the model placed on the sheet, click the symbol on the sheet, and change the name of the component in the property window (Value field in the DeviceName line, with the Parameter Values tab selected).

ECEW_Model1

The ECEW_Model contains the following parameters:

- **DeviceName** – the name of the device model with default name ECEW_Model1.
- **WindingName** – specifies the name of winding whose current is to be swept.
- **CurrentSweeps** – Specifies the current amplitude sweep list. For this, individual comma-separated values or range of values (linear step) can be specified.
    - For individual current values, the syntax is “v1, v2, v3” (for example, 0 A, 5 A, 10 A). The values must be input in ascending order (i.e., v1<v2<v3,…)
    - For a value range with linear step, the syntax is (dv1, n1), where dv1 denotes the step and n1 the number of steps.

      Value ranges always require an initial value, which is set to 0 A by default.

      For example, "(0.5 A, 5)" is equivalent to "0, 0.5 A, 1 A, 1.5 A, 2 A, 2.5 A"
    - Multiple value range inputs can be combined with each other. The later value range input will start with its first linear step at the end of the preceding value range.

      For example, "(0.5 A, 5), (1 A, 2)" is equivalent to "0, 0.5 A, 1 A, 1.5 A, 2 A, 2.5 A, 3.5 A, 4.5 A"
    - (Multiple) value range inputs can also be combined with individual values. Value range inputs always start from the last current step defined.

      For example, "(0.5 A, 5), 3 A, (1 A, 2)" is equivalent to "0, 0.5 A, 1 A, 1.5 A, 2 A, 2.5 A, 3 A, 4 A, 5 A"

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Make sure that the individual value inputs are larger than the last current value of value ranges preceding it. (*e.g.*, 3 A > 2.5 A=5*0.5 A as in the example above.)

- **Res** – phase winding resistance (available when Show Hidden is checked).
- **IndE** – phase end-leakage inductance (available when Show Hidden is checked).

An ECEW model can be used individually to create the model of a single-phase inductor, or combined with other ECEW_Models to setup two or more windings for poly-phase inductors. It can also be combined with an ECER model, as well as an ECE3 model, for rotational electrical machines.

For ECE model creation, the setup of Stop Time and Timestep are ignored. At Maxwell run time, the circuit simulator assigns currents for all windings step-by-step based on the lists of current sweeps until all sweeps are finished. Then an ECE model is created automatically. You can import this ECE model in Twin Builder via the **Twin Builder** > **SubCircuit** > **Maxwell Component** > **Add Equivalent Circuit** menu command. For more information on coupling Maxwell designs to Twin Builder components, refer to the *Maxwell Equivalent Circuit Component* topic in the Twin Builder Help.

> **Note:** You can add an ECEW_Model to an existing external circuit without deleting the original circuit. In such cases, the original circuit is ignored and can be kept for future use. If you want to recover the original circuit for a transient simulation, instead of ECE model creation, you just need to deactivate each ECE related model by right-clicking the model and selecting Deactivate in the short-cut menu.

For more information on coupling Maxwell designs to Twin Builder components, refer to Coupling Maxwell to a Twin Builder Component.

Passive Elements

Thirteen passive elements are available in the Maxwell Circuit Editor project tree:

- Cap: Capacitor
- DIODE: Diode
- DIODE_Model: Diode Model Data
- Ind: Inductor
- IndM: Mutual Inductance
- Res: Resistor
- SW_I: Current Controlled Switch
- SW_I4: Current Controlled Switch with Controlling Port
- SW_IModel: Model Data for Current Controlled Switches
- SW_V: Voltage Controlled Switch

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- SW_V4: Voltage Controlled Switch with Controlling Port
- SW_VModel: Model Data for Voltage Controlled Switches
- Transformer: Ideal Transformer

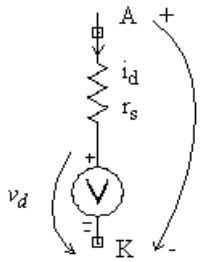
The text before the colon (**:**) represents the component name and can be changed in the **Properties** window once the component is placed in the schematic.

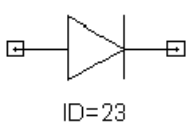
Diode & Diode Model

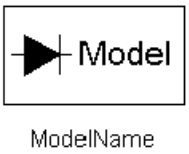
A Diode element must always be used together with a diode model. One diode model element can be used as reference for multiple diodes, as needed. Thus, once a diode is placed on the design sheet, a corresponding model must also be present on the sheet. Once both needed elements (diode and diode model) have been placed on the sheet, right mouse click the diode model and specify the parameters as required by the application. Then, right mouse click the diode and create the reference to the corresponding model by entering the name of the model in the MOD line (Parameter Values tab should be selected). If you select the **Show Hidden** check box, the AREA diode parameter (used below in the model definition) becomes visible. The default value of the AREA parameter is 1.

The diode model used by Maxwell Circuit Editor is a static model as described by the following equation:

$$i_d = i_s \cdot \left(e^{v_d/v_t} - 1\right) - \left(i_{bv} \cdot e^{-(v_d+BV)v_t}\right)$$







$v_t = N \cdot (kT)/q$

$q = 1.6022 \cdot 10^{-19}\ C$

$k = 1.3807 \cdot 10^{-23} (J/K)$

T is temperature in K, fixed at 300 K.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$i_s = IS \cdot AREA \cdot e^{\left(\frac{T/T_0 - 1}{kT - 1}\right) \cdot (EG)/v_t} \cdot \left(\frac{T}{T_0}\right)^{\frac{XTI}{N}}$$

$$i_{bv} = IBV \cdot AREA$$

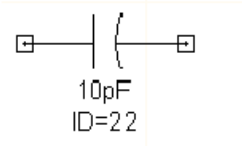
$$r_s = \frac{RS}{AREA}$$

The model parameters are as follows:

- *IS* is the saturation current in Amps.
- *RS* is contact resistance in Ohms.
- *N* is the emission coefficient.
- *EG* is the barrier height at 0 K, in volts.
- *XTI* is the diode saturation current temperature coefficient.
- *BV* is the magnitude of the reverse breakdown voltage in volts.
- *IBV* is the magnitude of the reverse breakdown current in amps.
- *TNOM* is the reference temperature in Celsius.

Capacitor (CAP)

A capacitor is assumed to be ideal (without losses or inductance) and is defined by the value of its capacitance (in the unit chosen by the user) and the corresponding initial condition (initial voltage). The straight bar of the capacitor symbol is used as the positive reference, and the curved bar of the capacitor symbol is used as a negative reference for the initial voltage (expressed in volts). The default value of the initial voltage for all capacitors is zero. By default, the capacitance and element ID are displayed next to the component.

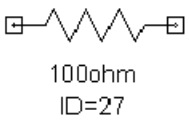


Resistor (Res)

Resistor is assumed to be ideal (without inductive or capacitive effects) and is defined by the value of its resistance (in the unit chosen by the user, Ohm by default.

By default the resistance and element ID are displayed next to the component.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

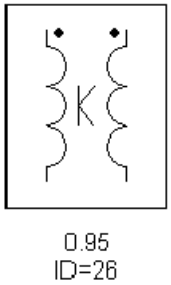


Inductor (IND)

Inductor is assumed to be ideal (without resistive or capacitive effects) and is defined by the value of its inductance (in the unit chosen by the user) and the corresponding initial condition (initial current). Note that the dot next to the inductor symbol is used as the positive reference for the initial current (positive current is oriented from the "dotted" terminal toward to "undotted" terminal of the inductor, through the inductor). The dot is also used to specify the mutual inductance between two or more inductors and thus determines the "polarized" terminals of the inductors. The default value of the initial current for all inductors is zero. By default the inductance and element ID are displayed next to the component.

Mutual Inductance (IndM)

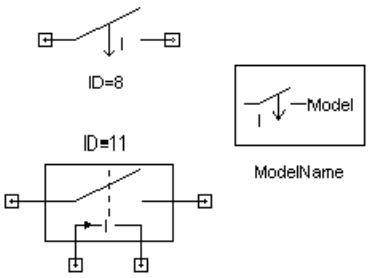
Mutual inductance is used to specify the inductive coupling between inductors. It is defined by specifying the IDs of the coupled inductors (always in pairs) and the coupling coefficient, a number between -1 and 1. The default value of the coupling coefficient is 0.95.



Current Controlled Switch (SW_I)

The current controlled switch comes in two flavors: with controlling port and without a controlling port. In either case a model data for the current controlled switch needs to be specified, similarly to the case of the diode.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Right-click the current controlled switch model and select **Properties**. With the **Parameter Values** tab selected, specify the switch model name (in the MOD line) as well as the ID of the controlling element: either an ammeter or a voltage source. In the later case the controlling quantity is the current through the voltage source. (Note that an ammeter is a voltage source with zero voltage, that is, a short circuit).

The current controlled switch with controlling port allows for the controlling quantity to be wired directly using connections with wires. In this case a reference arrow in the controlling port is displayed and is internally used as current reference (positive current flow as indicated by the arrow).

In the model for the current controlled switch the following parameters are used:

- R_{on} is the resistance of the switch in the **on** state (0.001 ohms default value).
- R_{off} is the resistance of the switch in the **off** state (1,000,000 ohms default value).
- I_{on} is the "**on**" value of the controlling current in amps. **If I > I_{on}, then R = R_{on}.**
- I_{off} is the "**off**" value of the controlling current in amps. **If I < I_{off}, then R = R_{off}.**

Note: Setting **R_{off} = 0** in the voltage or current controlled switch model data changes the behavior of the device into a **controlled conductance**, according to the following equation:

$$G(P) = \frac{1}{R_{on}} \cdot f_{Control}\left(P\right)$$

where $f_{control}(P)$ is a function describing the controlling signal, which is a time, position, or speed dependent current source or voltage source. Function ***G(P)*** clearly shows that the magnitude of the conductance is dictated by both the value of R_{on} and the magnitude of the control signal, while the time / position / speed dependency is dictated by the control signal itself.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

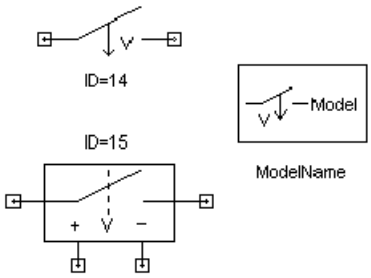
Note: Setting $\mathbf{R_{on} = 0}$ in the voltage or current controlled switch model data changes the behavior of the device into a **controlled resistance**, according to the following equation:

$$R(P) = R_{off} \cdot f_{Control}(P)$$

where $f_{control}(P)$ is a function describing the controlling signal, which is a time, position, or speed dependent current source or voltage source. Function **R(P)** clearly shows that the magnitude of the conductance is dictated by both the value of $\mathbf{R_{off}}$ and the magnitude of the control signal, while the time / position / speed dependency is dictated by the control signal itself.

Voltage Controlled Switch (SW_V)

The voltage controlled switch comes in two flavors: with controlling port and without a controlling port. In either case a model data for the voltage controlled switch needs to be specified, similarly to the case of the diode and current controlled switches.



Right-click the voltage controlled switch model and select **Properties**. With the **Parameter Values** tab selected, specify the switch model name (in the MOD line) as well as the ID of the controlling element: either a voltmeter or a current source. In the later case the controlling quantity is the voltage across the current source. (Note that a voltmeter is a current source with zero current, i.e. an open circuit).

The voltage controlled switch with controlling port allows for the controlling quantity to be wired directly using connections with wires. In this case the reference for the voltage across the controlling port is displayed by "+" and "-" symbols and the two symbols are internally used as voltage reference.

In the model for the voltage controlled switch the following parameters are used:

- R_{on} is the resistance of the switch in the **on** state (0.001 ohms default value).
- R_{off} is the resistance of the switch in the **off** state (1,000,000 ohms default value).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** Refer to **Additional Considerations** in the Sinusoidal Current Source topic for information on the effects of load resistance values in current calculations.

- V_{on} is the "**on**" value of the controlling voltage in volts. **If V > V_{on}, then R = R_{on}.**
- V_{off} is the "**off**" value of the controlling voltage in volts. **If V < V_{off}, then R = R_{off}.**

Note: Setting **R_{off} = 0** in the voltage or current controlled switch model data changes the behavior of the device into a **controlled conductance**, according to the following equation:

$$G(P) = \frac{1}{R_{on}} \cdot f_{control}\left(P\right)$$

where $f_{control}(P)$ is a function describing the controlling signal, which is a time, position, or speed dependent current source or voltage source. Function ***G(P)*** clearly shows that the magnitude of the conductance is dictated by both the value of R_{on} and the magnitude of the control signal, while the time / position / speed dependency is dictated by the control signal itself.

> **Important:** The minimum limit for G(P) value is 1e-8 siemens.

Note: Setting **R_{on} = 0** in the voltage or current controlled switch model data changes the behavior of the device into a **controlled resistance**, according to the following equation:

$$R(P) = R_{off} \cdot f_{control}(P)$$

where $f_{control}(P)$ is a function describing the controlling signal, which is a time, position, or speed-dependent current source or voltage source. Function **R(P)** clearly shows that the magnitude of the conductance is dictated by both the value of R_{off} and the magnitude of the control signal, while the time / position / speed dependency is dictated by the control signal itself.

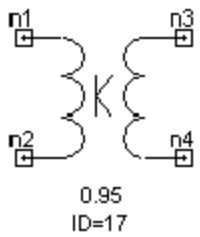
> **Important:** The minimum limit for R(P) value is 1e-8 ohm.

Transformer

The transformer is an ideal element of infinite power without resistive or capacitive effects and linear. It is defined the values of the primary and secondary inductances and the coupling

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

coefficient between the two windings.



Probes

Three types of probes are available in the Maxwell Circuit Elements component library:

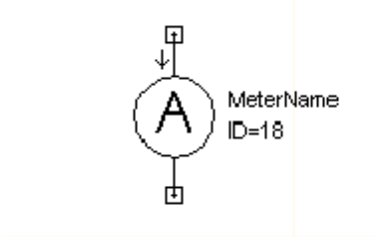
- Ammeter: Ammeter
- Voltmeter: Voltmeter
- VoltmeterG: Voltmeter with One Pin Grounded

The text before the colon (**:**) represents the component name and can be changed in the **Properties** window once the component is placed in the schematic.

Ammeter

The ammeter is an ideal element (equivalent with an ideal voltage source with zero voltage). An arrow is attached to the symbol so that a positive current measured by the ammeter flows as indicated by the arrow.

No numerical value is needed, just the element ID and name may by altered by the user.

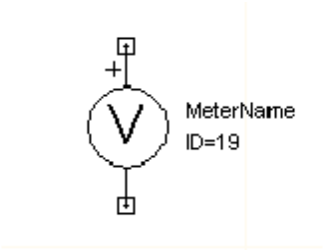


Voltmeter

The voltmeter is an ideal element with two pins (equivalent with an ideal current source with zero current). A plus sign is attached to the voltmeter symbol so that a reference for the voltage measured by the voltmeter is possible.

No numerical value is needed, just the element ID and name may by altered by the user.

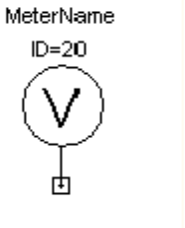
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Voltmeter with One Pin Grounded (VoltmeterG)

The voltmeter with one pin grounded is an ideal element with one pin, equivalent with an ideal current source with zero current. A plus sign is not necessary since it is assumed that the grounded pin (not available for connection in the circuit) is the negative one while the pin available for connection is the positive one. Thus, the reference for the voltage measured by the voltmeter is possible.

No numerical value is needed, just the element ID and name may by altered by the user.



Current and Voltage Sources

Sources available in the Maxwell Circuit Editor can be defined such that in general the dependency of the current or voltage can be made function of time, position, or speed. The type of dependency is part of the properties of the respective source and is always user-selectable. Thus, in the equation defining the behavior of the source (when specified), the variable "t" can mean TIME or POSITION or SPEED as selected by the user for each application.

The default dependency type is TIME.

The following types of sources are available in the Maxwell Circuit Elements component library:

- IDC: DC Current Source
- IExp: Exponential Current Source
- IPulse: Pulse Current Source

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- IPWL: Piecewise Linear Current Source
- IPWM: PWM Current Source
- ISffm: Frequency Modulated Sinusoidal Current Source
- ISin: Sinusoidal Current Source
- VDC: DC Voltage Source
- VExp: Exponential Voltage Source
- VPulse: Pulse Voltage Source
- VPWL: Piecewise Linear Voltage Source
- VPWM: PWM Voltage Source
- VSffm: Frequency Modulated Sinusoidal Voltage Source
- VSin: Sinusoidal Voltage Source

The text before the colon (**:**) represents the component name and can be changed in the **Properties** window once the component is placed in the schematic.

DC Current Source

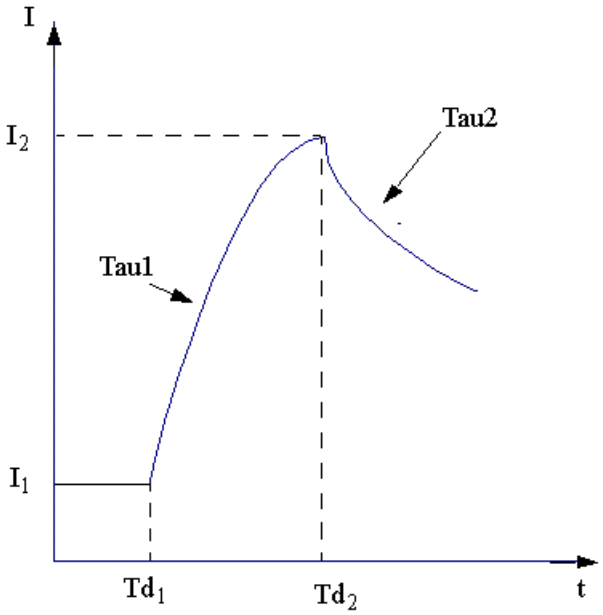
This type of source specifies the DC current flowing in a conductor. You can set either the total current or the current density flowing in the object.

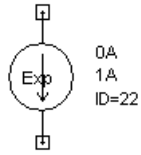
- If total current is specified, the current density is assumed to be uniformly distributed throughout the object.
- If current density is specified, you may define a uniform current density or one that varies as a function of position.

Exponential Current Source

This is an independent current source with an exponential waveform of the current as function of time as shown in the following figure. The arrow symbol shows the direction of the positive current flow through the current source.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Thus, the parameters of this exponential source are

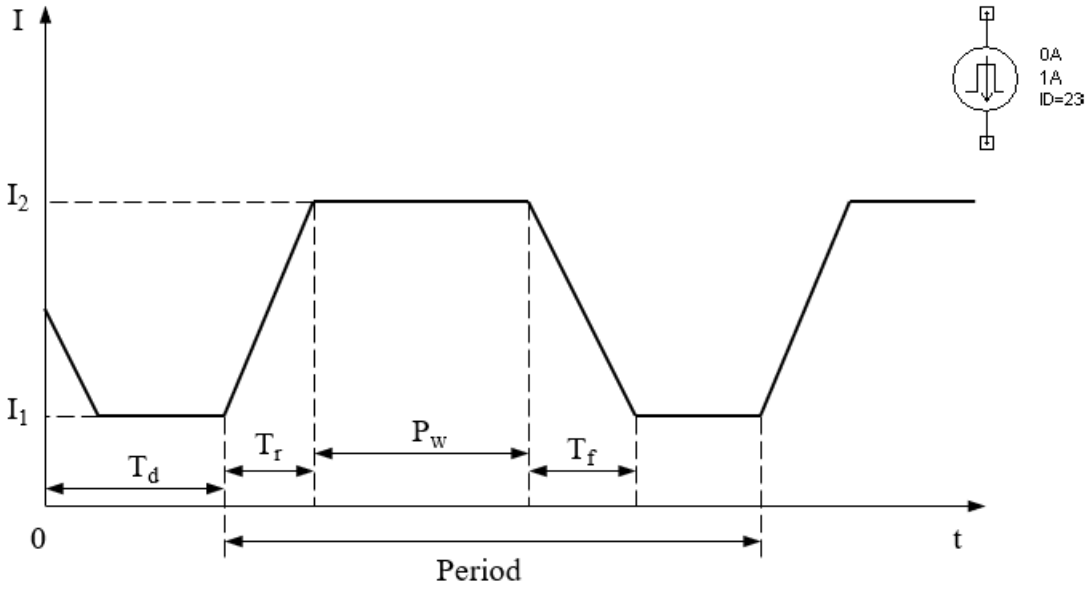
- Initial current in Amps, I1;
- Peak current in Amps, I2;
- Rise time delay, Td1:
  - In seconds if type is TIME
  - In degrees if type is POSITION and type of motion is rotational;
  - In geometry units if type is POSITION and type of motion is translational;
  - In rpm if type is SPEED and type of motion is rotational;
  - In geometry units per second if type is SPEED and type of motion is translational;
- Rise time constant, Tau1:
  - In seconds if type is TIME
  - In degrees if type is POSITION and type of motion is rotational;
  - In geometry units if type is POSITION and type of motion is translational;
  - In rpm if type is SPEED and type of motion is rotational;
  - In geometry units per second if type is SPEED and type of motion is translational;
- Fall time delay, Td2:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - In seconds if type is TIME
  - In degrees if type is POSITION and type of motion is rotational;
  - In geometry units if type is POSITION and type of motion is translational;
  - In rpm if type is SPEED and type of motion is rotational;
  - In geometry units per second if type is SPEED and type of motion is translational;
- Fall time constant, Tau2:
  - In seconds if type is TIME
  - In degrees if type is POSITION and type of motion is rotational;
  - In geometry units if type is POSITION and type of motion is translational;
  - In rpm if type is SPEED and type of motion is rotational;
  - In geometry units per second if type is SPEED and type of motion is translational;

Pulse Current Source

This is an independent current source with a trapezoidal waveform of the current as a function of time. The arrow symbol shows the direction of the positive current flow through the current source.



The parameters of a pulse current source are the following:

- Initial current in Amps, I_1;
- Peak current in Amps, I_2;
- Initial delay time shift, T_d:
  - The waveform is repeated during $t < T_d$ based on the period.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - In seconds if type is TIME
  - In degrees if type is POSITION and type of motion is rotational
  - In geometry units if type is POSITION and type of motion is translational
  - In rpm if type is SPEED and type of motion is rotational
  - In geometry units per second if type is SPEED and type of motion is translational
- Rise time, T_r:
  - In seconds if type is TIME
  - In degrees if type is POSITION and type of motion is rotational
  - In geometry units if type is POSITION and type of motion is translational
  - In rpm if type is SPEED and type of motion is rotational
  - In geometry units per second if type is SPEED and type of motion is translational
- Fall time T_f:
  - In seconds if type is TIME
  - In degrees if type is POSITION and type of motion is rotational
  - In geometry units if type is POSITION and type of motion is translational
  - In rpm if type is SPEED and type of motion is rotational
  - In geometry units per second if type is SPEED and type of motion is translational
- Pulse width, P_w:
  - In seconds if type is TIME
  - In degrees if type is POSITION and type of motion is rotational
  - In geometry units if type is POSITION and type of motion is translaional
  - In rpm if type is SPEED and type of motion is rotational
  - In geometry units per second if type is SPEED and type of motion is translational
- Pulse period, Period:
  - In seconds if type is TIME
  - In degrees if type is POSITION and type of motion is rotational
  - In geometry units if type is POSITION and type of motion is translational
  - In rpm if type is SPEED and type of motion is rotational
  - In geometry units per second if type is SPEED and type of motion is translational

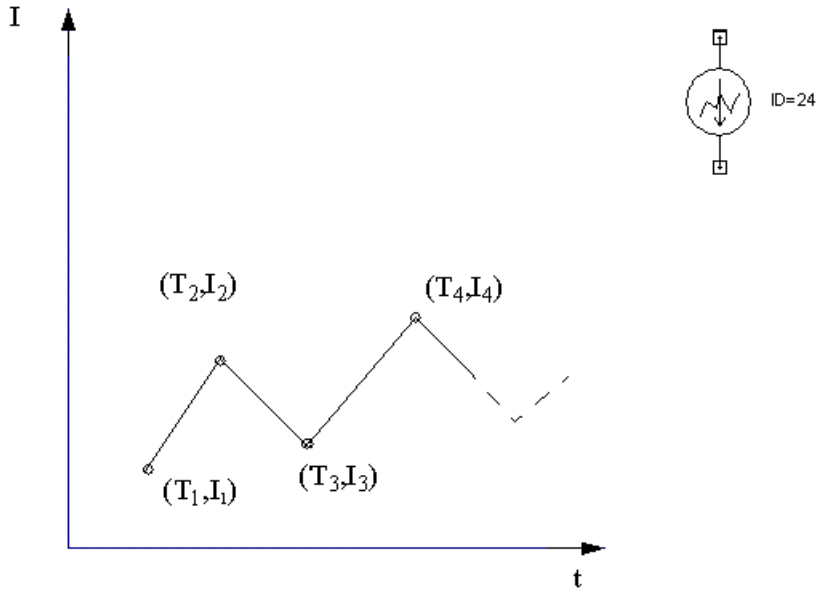
> **Warning:** The simulation will only output the correct waveform when $T_d + T_r + T_f + P_w \leq$ Period.

Piecewise Linear Current Source

The Piecewise Linear Current Source is an independent current source with a piecewise linear waveform of the current as a function of time. The arrow symbol shows the direction of the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

positive current flow through the current source.

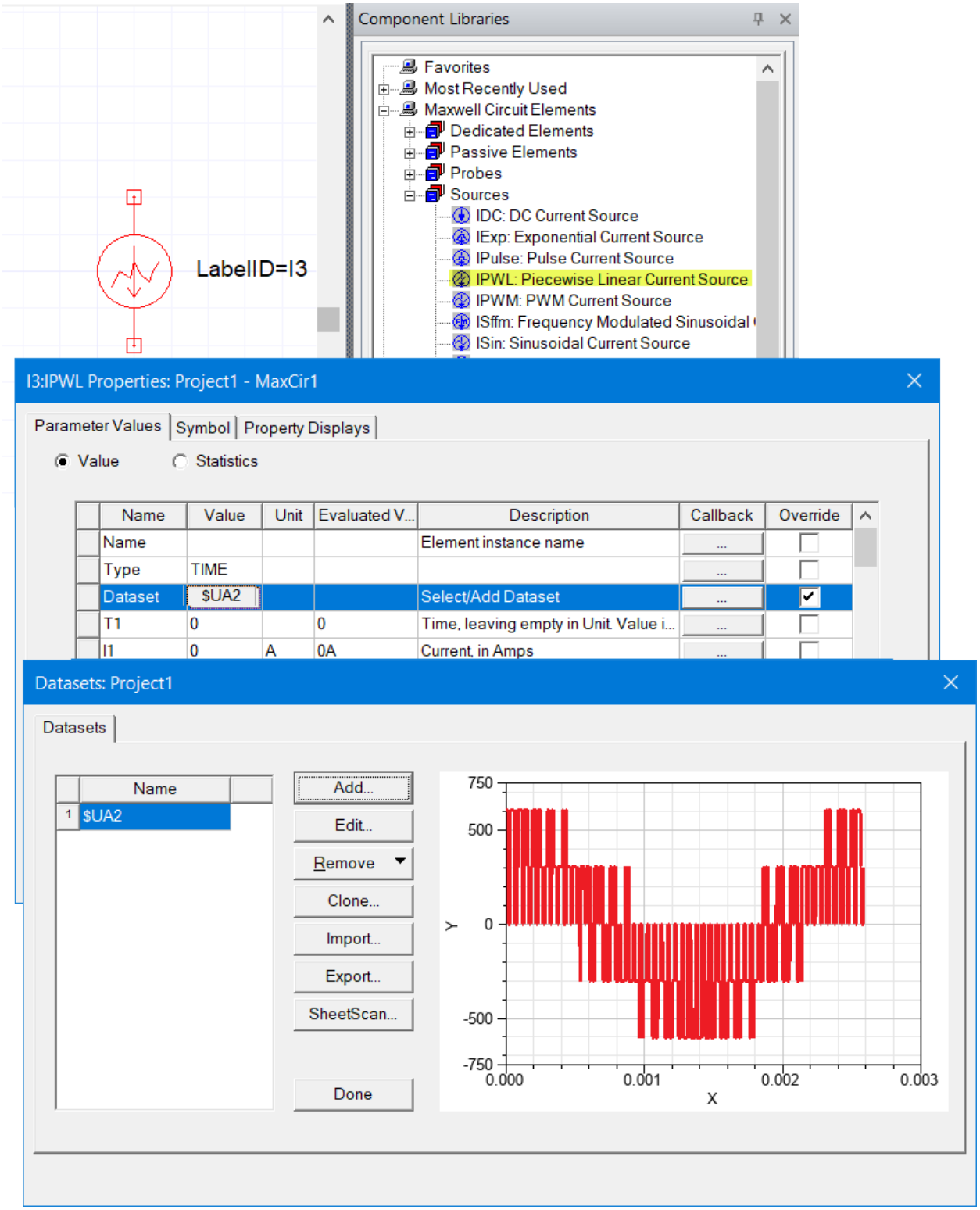


A piecewise linear current source is described by up to 20 pairs (T_i, I_i), where every pair of values specifies the value I_i in Amps of the current at time T_i in the following units:

- In seconds if type is TIME
- In degrees if type is POSITION and type of motion is rotational
- In geometry units if type is POSITION and type of motion is translational
- In rpm if type is SPEED and type of motion is rotational
- In geometry units per second if type is SPEED and type of motion is translational

Alternatively, instead of specifying 20-pair data to define a PWL waveform, you can specify a project-level dataset. In this case, the dataset needs to be specified in the properties of the source and any defined point pairs are ignored. Only project-level datasets can be used for this purpose as design-level datasets do not exist for external circuit designs. For more details on using datasets, see Adding Datasets.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



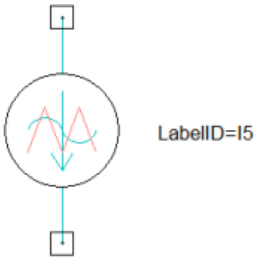
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PWM Current Source

The PWM current source component shown below provides PWM control signals used to support TDM parallel computation. (Refer to Time Decomposition Method for Maxwell Transient Designs in the Maxwell help.) The arrow symbol shows the direction of positive current flow through the current source.

Because all timesteps in one subdivision along the time axis are solved together when TDM is used, all controlling signals in a control circuit cannot depend on solutions such as currents/voltages of the previous timestep. Therefore, all diodes and switches which are controlled by independent voltages and currents are not supported.

In PWM control, the controlling signals are purely time and/or position dependent, therefore, PWM control is supported in TDM. In a PWM control circuit, since the upper and lower switches belong to the same phase are turned on and off at exactly same time, the free-wheeling diodes are not necessary. In order for users to directly use existing PWM control circuit, PWM current and voltage source components have been introduced to replace original controlling sub-circuit which includes a triangle current/voltage source connected in series with a sinusoidal current/voltage source. The circuit solver will detect all switches which are controlled by PWM current/voltage sources and automatically filter all related diodes, which means during the simulation, diodes are automatically excluded in PWM control circuit.



> **Note:** Prior to the introduction of the PWM current source (IPWM) component, PWM control signals used to support TDM parallel computation were obtained from the difference of two current sources: a carrier signal with triangular waveform (IPulse), and a control signal with sinusoidal waveform (ISin)

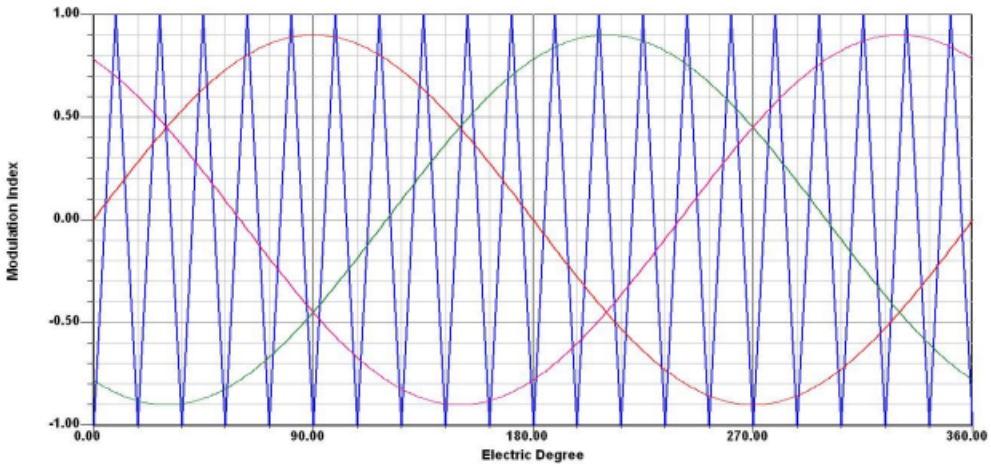
An **IPWM** current source is defined by the following parameters:

- **ModuIndex** is the modulation index, the ratio of control-signal amplitude to carrier-signal amplitude.
- **FreqTime** is the ratio of carrier-signal frequency to control-signal frequency.
- **IFreq** is the frequency of the control signal, if **Type** is **TIME**.
- **Phase** is the phase delay of the control signal, in electrical degrees.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

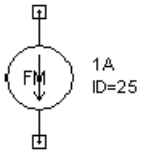
- **Type** can be chosen as either **TIME** or **POS**, defining the output signal as a function of time or position.

The following figure shows typical three-phase PWM control signals:



Frequency-Modulated Sinusoidal Current Source

This is an independent current source with a single frequency modulated sinusoidal waveform of the current as a function of time. The arrow symbol shows the direction of the positive current flow through the current source.



The equation describing the waveform is

$I(t) = I_O + I_a \cdot \sin[(2\pi \cdot F_C \cdot t) + M_{di} \cdot \sin(2\pi \cdot F_S \cdot t)]$

where

- I_O is Offset current in Amps.
- I_a is the peak amplitude in Amps.
- F_C is the carrier frequency if type is TIME.
- M_{di} is the modulation index.
- F_S is the signal frequency if type is TIME.

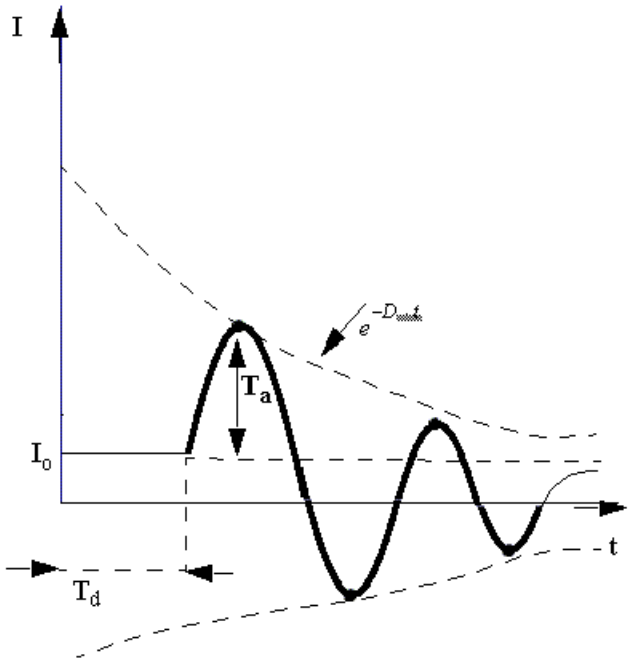
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

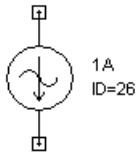
If the type is POSITION, the frequency should be calculated based on the respective spatial periodicity, taking into account the fact that *t* in the above equation is measured in degrees for rotational type of motion and in the user-defined geometry units for translational type of motion.

If the type is SPEED, the frequency should be calculated based on the respective speed periodicity, taking into account the fact that *t* in the above equation is measured in rpm for rotational type of motion and in the user-defined geometry units per second for translational type of motion.

Sinusoidal Current Source

This is an independent current source with an exponentially damped sinusoidal waveform of the current as a function of time. The arrow symbol shows the direction of the positive current flow through the current source.





The equation describing the waveform is

$$I(t) = I_0 + I_a \cdot e^{-Df(t-T_d)} \cdot \sin\left[2 \cdot \pi \cdot IFreq \cdot (t - T_d) - Phase\right]$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where

- I_0 is offset current in Amps.
- I_a is the peak amplitude in Amps.
- *IFreq* is the signal frequency if type is TIME.
- T_d is the delay time in seconds if type is TIME.
- *Phase* is the signal phase delay if type is TIME.
- *Df* is the damping factor in 1/seconds if type is TIME.

If the type is POSITION, the frequency should be calculated based on the respective spatial periodicity, taking into account the fact that *t* in the above equation is measured in degrees for rotational type of motion and in the user-defined geometry units for translational type of motion. The delay and damping factor should also be interpreted accordingly.

If the type is SPEED, the frequency should be calculated based on the respective speed periodicity, taking into account the fact that *t* in the above equation is measured in rpm for rotational type of motion and in the user-defined geometry units per second for translational type of motion. The delay and damping factor should also be interpreted accordingly.

Additional Considerations

To simplify the coupling equations for the field solver, Ansys uses loop analysis method in the circuit solver. That is, the circuit solver will forward a loop matrix and a loop voltage vector to the field solver, and the field solver will return solved loop currents to the circuit solver. When one or more windings are excited by ideal current sources in an external circuit, the loop matrix forwarded to the field solver will be singular. In such a case, the coupling algorithm is changed to directly forward current values from the circuit solver to the field solver. This will cause large errors when some windings are excited by current sources while some windings are excited by voltage sources because the currents of all voltage-excited windings are estimated based on the parameters of the previous timestep.

To avoid errors in current estimation, the solver transforms all ideal current sources to real current sources with a large internal parallel resistance by connecting a large parallel resistance. The reasonable value of such large resistance is 1e08 ohm. If the value is too large, all normal component resistance in the same loop with this large resistance will be "eaten" by the resistance which makes the loop matrix singular. If it is too small, the bypass current through the large resistance will cause inaccurate current through a component connected in series with the current source. Like a real voltage source with a small internal series resistance in which the terminal voltage will change with the load resistance, the load current of a real current source with a large internal parallel resistance will change with the load resistance.

The error between the load current and the source current depends on load resistance. Since the parallel resistance of the current source is internally fixed as 1e08 ohm, the error will be

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

within one percent if the load resistance is limited with 1e06 ohm. This limit is due to the transformation of current source, instead of the switch.

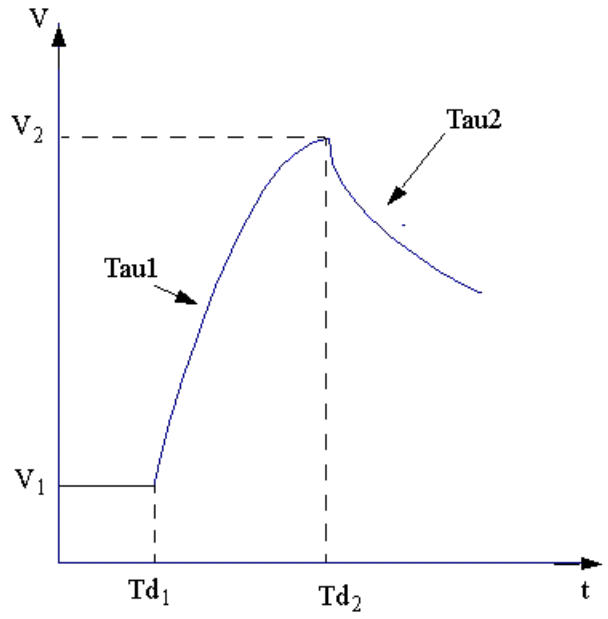
DC Voltage Source

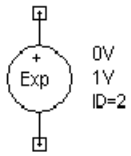
This type of source specifies the total DC voltage (electric potential) on a conductor. Voltages can be defined as constants or as math functions; however, the potential on a conductor is constant over the entire conductor. Note that conductors that touch should be set to the same voltage or defined as a single voltage source, since their potentials are identical.

DC voltage may also be specified on an edge or edges. Voltages can be defined as constant or as functions of position (for instance, to model a specific distribution of potential on the surface of a dielectric).

Exponential Voltage Source

This is an independent voltage source with an exponential waveform of the voltage as a function of time. The "+" and "-" symbols are used to mark the polarity of the source.





The parameters of an exponential voltage source are the following:

- Initial voltage in Volts, V1
- Peak voltage in Volts, V2
- Rise time delay, Td1:
  - In seconds if type is TIME
  - In degrees if type is POSITION and type of motion is rotational

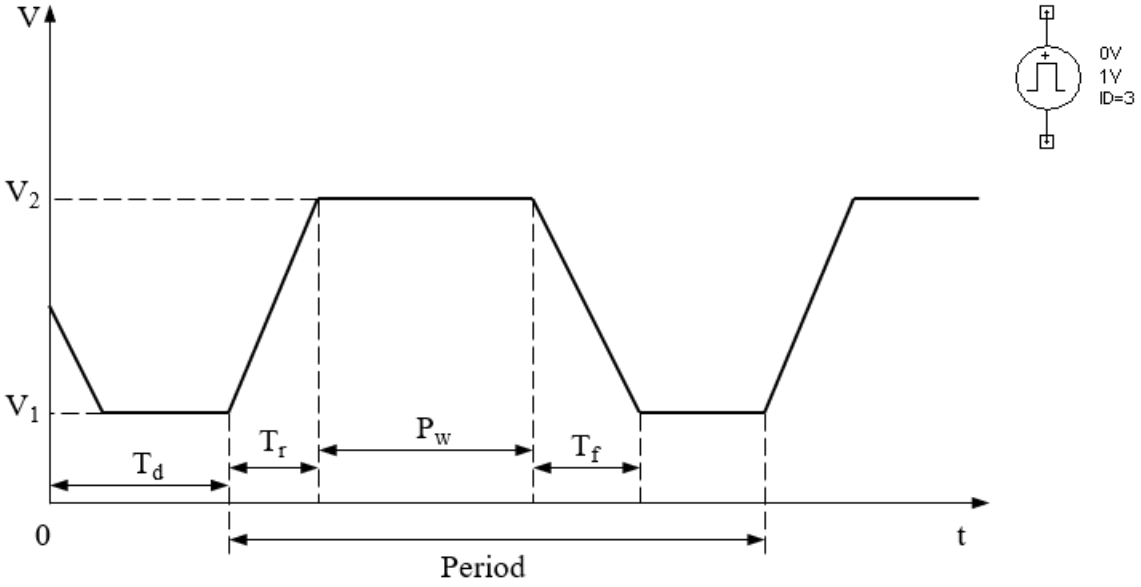
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - In geometry units if type is POSITION and type of motion is translational
  - In rpm if type is SPEED and type of motion is rotational
  - In geometry units per second if type is SPEED and type of motion is translational
- Rise time constant, Tau1:
  - In seconds if type is TIME
  - In degrees if type is POSITION and type of motion is rotational
  - In geometry units if type is POSITION and type of motion is translational
  - In rpm if type is SPEED and type of motion is rotational
  - In geometry units per second if type is SPEED and type of motion is translational
- Fall time delay, Td2:
  - In seconds if type is TIME
  - In degrees if type is POSITION and type of motion is rotational
  - In geometry units if type is POSITION and type of motion is translational
  - In rpm if type is SPEED and type of motion is rotational
  - In geometry units per second if type is SPEED and type of motion is translational
- Fall time constant, Tau2:
  - In seconds if type is TIME
  - In degrees if type is POSITION and type of motion is rotational
  - In geometry units if type is POSITION and type of motion is translational
  - In rpm if type is SPEED and type of motion is rotational
  - In geometry units per second if type is SPEED and type of motion is translational

Pulse Voltage Source

This is an independent voltage source with a trapezoidal waveform of the voltage as a function of time. The "+" and "-" symbols are used to mark the polarity of the source.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The parameters of a voltage pulse source are the following:

- Initial voltage in Volts, V_1
- Peak voltage in Volts, V_2
- Initial delay time shift, T_d:
  - The waveform is repeated during $t < T_d$ based on the period
  - In seconds if type is TIME
  - In degrees if type is POSITION and type of motion is rotational
  - In geometry units if type is POSITION and type of motion is translational
  - In rpm if type is SPEED and type of motion is rotational
  - In geometry units per second if type is SPEED and type of motion is translational
- Rise time, T_r:
  - In seconds if type is TIME
  - In degrees if type is POSITION and type of motion is rotational
  - In geometry units if type is POSITION and type of motion is translational
  - In rpm if type is SPEED and type of motion is rotational
  - In geometry units per second if type is SPEED and type of motion is translational
- Fall time T_f:
  - In seconds if type is TIME
  - In degrees if type is POSITION and type of motion is rotational
  - In geometry units if type is POSITION and type of motion is translational

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

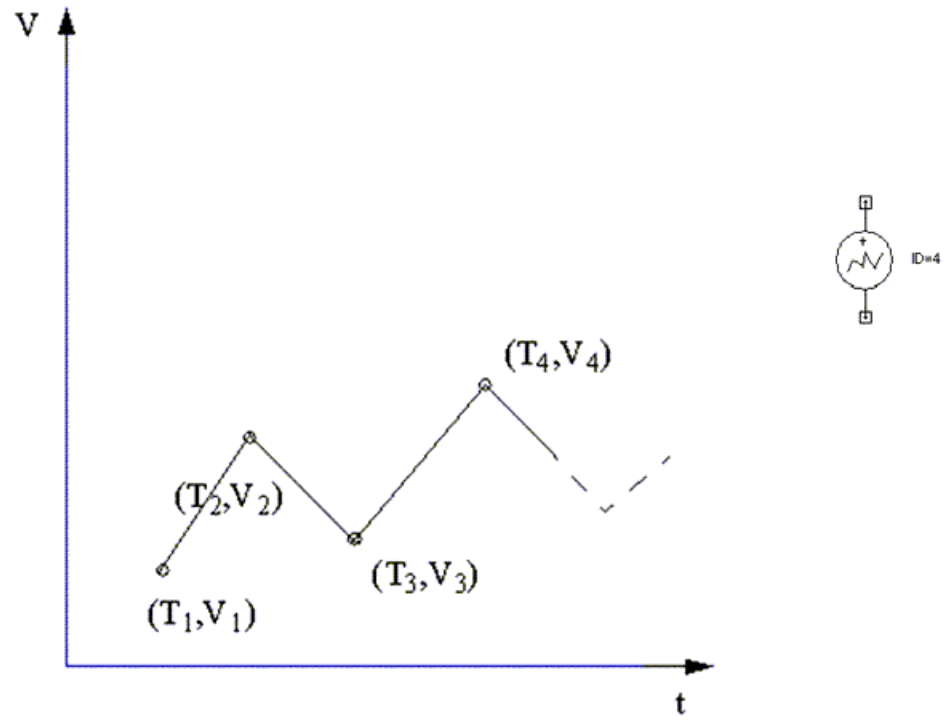
    - In rpm if type is SPEED and type of motion is rotational
    - In geometry units per second if type is SPEED and type of motion is translational
- Pulse width, P_w:
    - In seconds if type is TIME
    - In degrees if type is POSITION and type of motion is rotational
    - In geometry units if type is POSITION and type of motion is translational
    - In rpm if type is SPEED and type of motion is rotational
    - In geometry units per second if type is SPEED and type of motion is translational
- Pulse period, Period:
    - In seconds if type is TIME
    - In degrees if type is POSITION and type of motion is rotational
    - In geometry units if type is POSITION and type of motion is translational
    - In rpm if type is SPEED and type of motion is rotational
    - In geometry units per second if type is SPEED and type of motion is translational

> **Warning:** The simulation will only output the correct waveform when $T_d + T_r + T_f + P_w \leq$ Period.

Piecewise Linear Voltage Source

This is an independent voltage source with a piecewise linear waveform of the voltage as a function of time. The "+" and "-" symbols are used to mark the polarity of the source.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

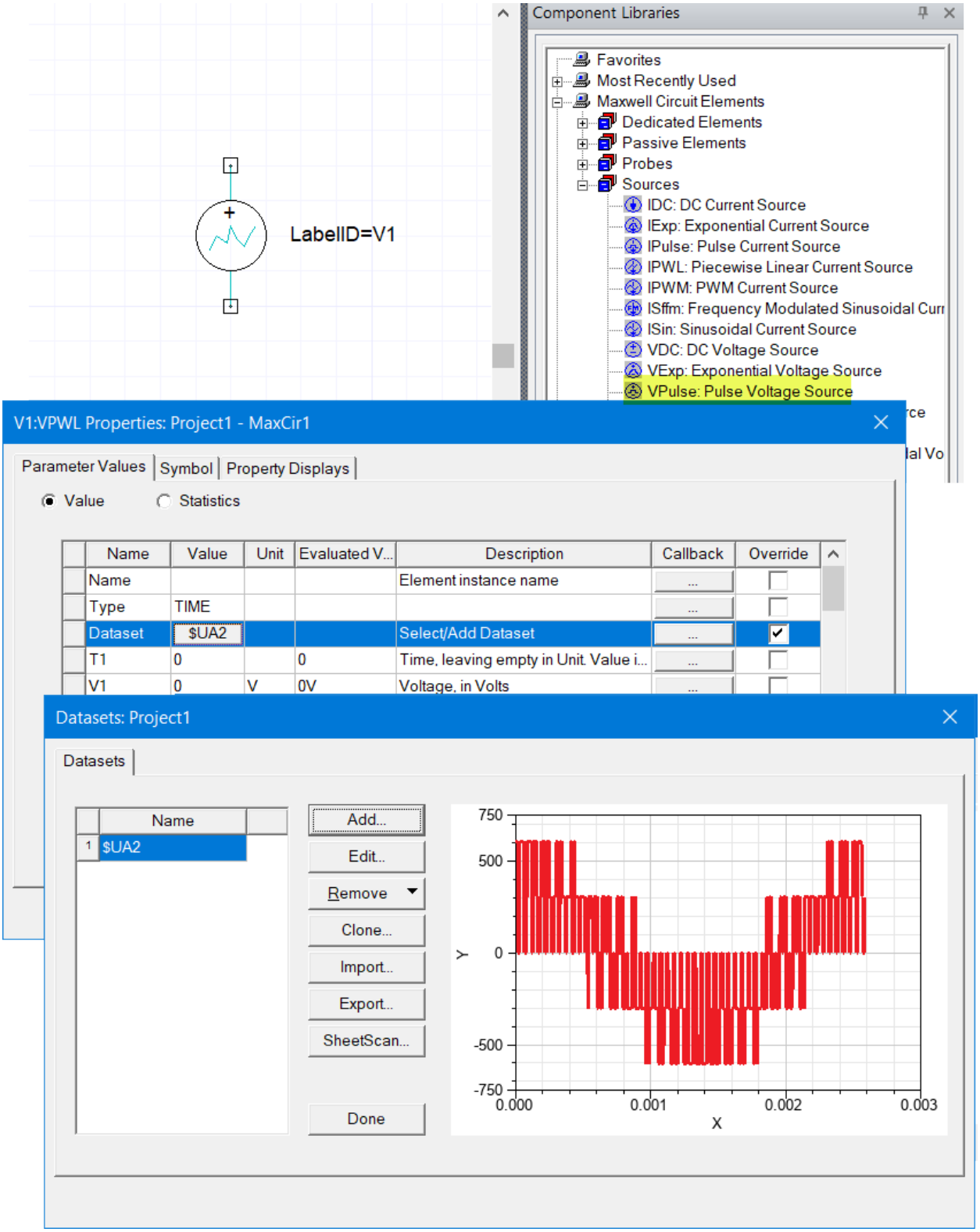


A piecewise linear voltage source is described by up to 20 pairs (T_i, V_i), where every pair of values specifies the value V_i in Volts of the voltage at time T_i in the following units:

- In seconds if type is TIME
- In degrees if type is POSITION and type of motion is rotational
- In geometry units if type is POSITION and type of motion is translational
- In rpm if type is SPEED and type of motion is rotational
- In geometry units per second if type is SPEED and type of motion is translational

Alternatively, instead of specifying 20-pair data to define a PWL waveform, you can specify a project-level dataset. In this case, the dataset needs to be specified in the properties of the source and any defined point pairs are ignored. Only project-level datasets can be used for this purpose as design-level datasets do not exist for external circuit designs. For more details on using datasets, see Adding Datasets.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



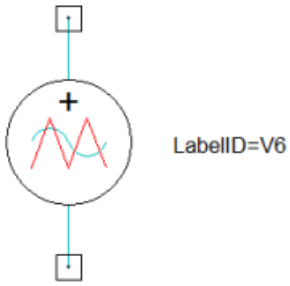
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PWM Voltage Source

The PWM voltage source component shown below provides PWM control signals used to support TDM parallel computation. (Refer to *Time Decomposition Method for Maxwell Transient Designs* in the Maxwell help.) The "+" symbol is used to mark the polarity of the source.

Because all timesteps in one subdivision along the time axis are solved together when TDM is used, all controlling signals in a control circuit cannot depend on solutions such as currents/voltages of the previous timestep. Therefore, all diodes and switches which are controlled by independent voltages and currents are not supported.

In PWM control, the controlling signals are purely time and/or position dependent, therefore, PWM control is supported in TDM. In a PWM control circuit, since the upper and lower switches belong to the same phase are turned on and off at exactly same time, the free-wheeling diodes are not necessary. In order for users to directly use existing PWM control circuit, PWM current and voltage source components have been introduced to replace original controlling sub-circuit which includes a triangle current/voltage source connected in series with a sinusoidal current/voltage source. The circuit solver will detect all switches which are controlled by PWM current/voltage sources and automatically filter all related diodes, which means during the simulation, diodes are automatically excluded in PWM control circuit.



> **Note:** Prior to the introduction of the PWM voltage source (VPWM) component, PWM control signals used to support TDM parallel computation were obtained from the difference of two voltage sources: a carrier signal with triangular waveform (VPulse), and a control signal with sinusoidal waveform (VSin)

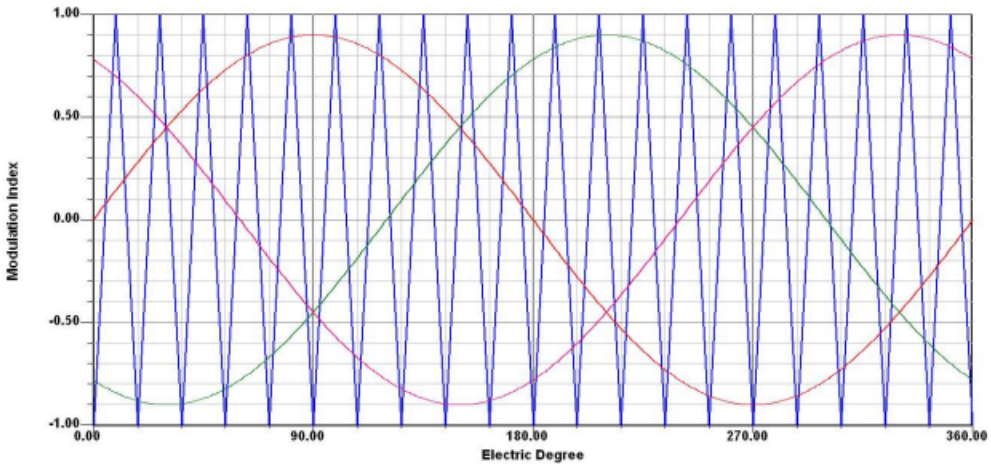
A **VPWM** source is defined by the following parameters:

- **ModuIndex** is the modulation index, the ratio of control-signal amplitude to carrier-signal amplitude.
- **FreqTime** is the ratio of carrier-signal frequency to control-signal frequency.
- **VFreq** is the frequency of the control signal, if **Type** is **TIME**.
- **Phase** is the phase delay of the control signal, in electrical degrees.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

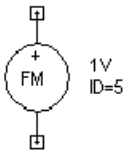
- **Type** can be chosen as either **TIME** or **POS**, defining the output signal as a function of time or position.

The following figure shows typical three-phase PWM control signals.



Frequency-Modulated Sinusoidal Voltage Source

This is an independent voltage source with a single frequency modulated sinusoidal waveform of the voltage as a function of time. The "+" and "-" symbols are used to mark the polarity of the source.



The equation describing the waveform is

$$V(t) = V_0 + V_a \cdot \sin[(2\pi \cdot F_C \cdot t) + M_{di} \cdot \sin(2\pi \cdot F_S \cdot t)]$$

where

- V_0 is Offset voltage in Volts.
- V_a is the peak amplitude in Volts.
- F_C is the carrier frequency if type is TIME.
- M_{di} is the modulation index.
- F_S is the signal frequency if type is TIME.

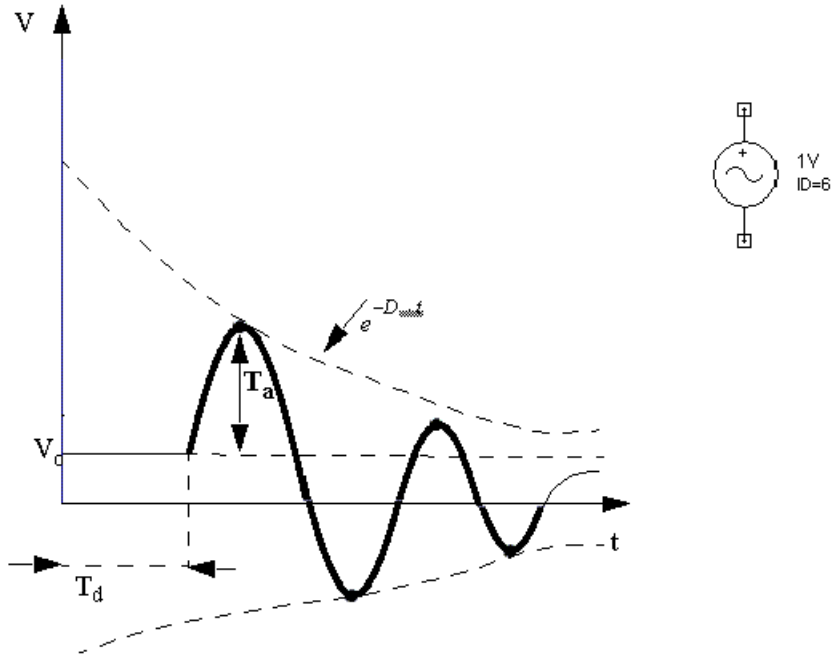
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the type is POSITION, the frequency should be calculated based on the respective spatial periodicity. taking into account the fact that "t" in the above equation is measured in degrees for rotational type of motion and in the user-defined geometry units for translational type of motion.

If the type is SPEED, the frequency should be calculated based on the respective speed periodicity, taking into account the fact that "t" in the above equation is measured in rpm for rotational type of motion and in the user-defined geometry units per second for translational type of motion.

Sinusoidal Voltage Source

This is an independent voltage source with an exponentially damped sinusoidal waveform of the voltage as a function of time. The "+" and "-" symbols are used to mark the polarity of the source.



The equation describing the waveform is

$$V(t) = V_0 + V_a \cdot e^{-Df(t-T_d)} \cdot \sin\left[2 \cdot \pi \cdot VFreq \cdot (t - T_d) - Phase\right]$$

where

- V_0 is Offset voltage in Volts.
- V_a is the peak amplitude in Volts.
- *VFreq* is the signal frequency if type is TIME.
- T_d is the delay time in seconds if type is TIME.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

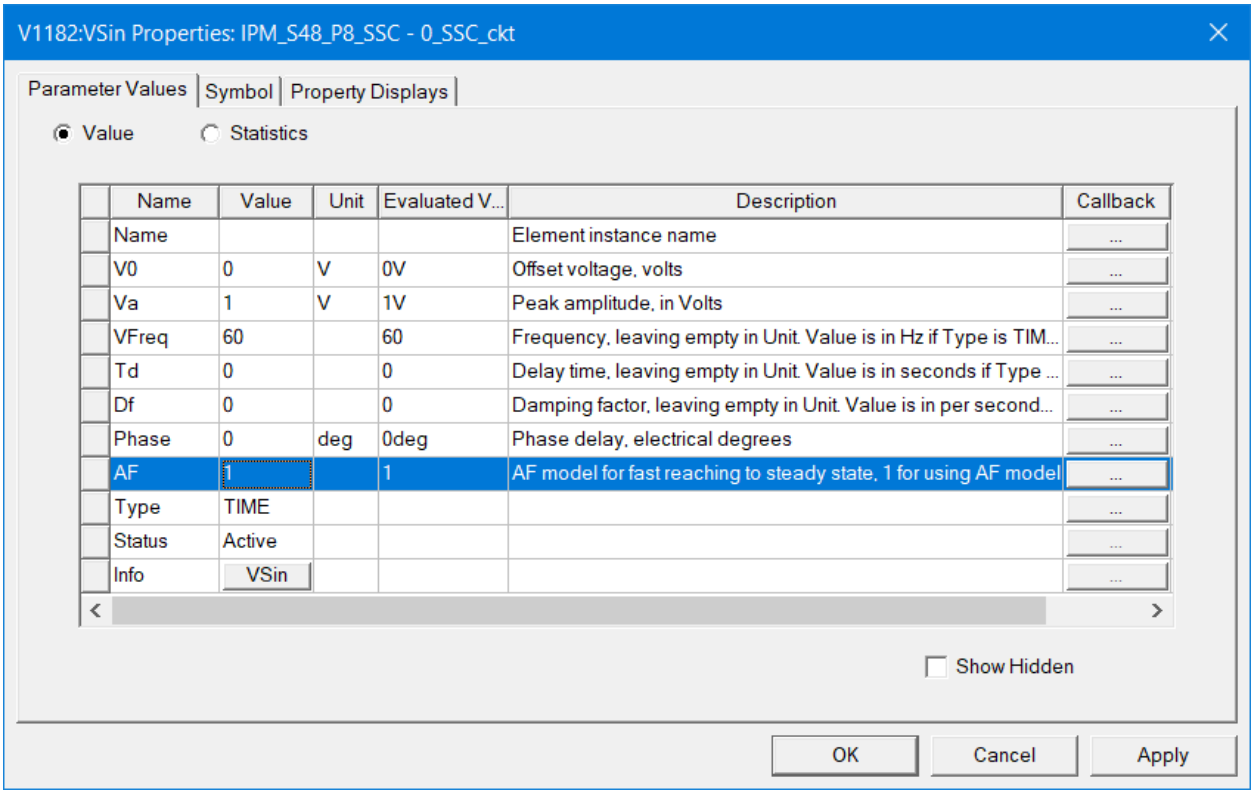
- *Phase* is the signal phase delay if type is TIME.
- Df is the damping factor in 1/seconds if type is TIME.

If the type is POSITION, the frequency should be calculated based on the respective spatial periodicity, taking into account the fact that "t" in the above equation is measured in degrees for rotational type of motion and in the user-defined geometry units for translational type of motion. The delay and damping factor should also be interpreted accordingly.

If the type is SPEED, the frequency should be calculated based on the respective speed periodicity, taking into account the fact that "t" in the above equation is measured in rpm for rotational type of motion and in the user-defined geometry units per second for translational type of motion. The delay and damping factor should also be interpreted accordingly.

Using the Alternating Flux (AF) Model

For transient simulations whose excitation is the sinusoidal voltage source, the DC flux linkage component may take a very long time to decay, especially for a device with a large time constant. Setting AF=1 enables the AF model to be used by the Maxwell transient solver.



The AF model algorithm adds an additional voltage component to the original voltage definition during the first half-cycle to quickly eliminate the DC flux linkage, thus speeding up simulation solutions.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** RMxprt automatically enables the AF model in the voltage definition for each winding when the Maxwell 2D or 3D model and associated Maxwell circuit are created. This speed-up capability is only applicable to voltage sources whose integration of voltage quantity over one period is zero, so the DC component of all voltage sources must also be zero.

Placing Components in the Maxwell Circuit Editor Schematic

To place a circuit component:

1. Click the **Components** tab in the **Component Libraries Window**.
2. In the tree, expand the **Maxwell Circuit Elements** branch.
3. Expand the branch containing the component you want to place. The choices are **Dedicated Elements**, **Passive Elements**, **Probes**, and **Sources**.
4. Click to select the name of the component you want to place.
5. Drag the component to the Schematic window.

   A diagram of the component appears connected to the mouse pointer.
6. Release the mouse button to place the component in the location you prefer.
7. To place a second component of the same type, move the mouse pointer to another location, and release it again.
8. To exit from placement mode, do one of the following:
   - To place the component a final time before exiting, press **ENTER**.
   - To exit without placing the component again, press the Space bar.

   > **Tip:** To access these commands, you can also right-click and select one of the following from the shortcut menu:
   > - Place and Finish
   > - Finish

Assigning Component Properties in Maxwell Circuit Editor

Once a component has been placed in the Schematic window, do the following to edit its properties:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Double-click the component.
2. In the **Properties** window that opens, make the desired edits on the following four tabs:
    - Parameter Values
    - General
    - Symbol
    - Property Displays

    > **Note:** Variables may be assigned to parameter values in the circuit by entering a variable name in the parameter value field and assigning a value in the **Add Variable** dialog box. Variables assigned in **Maxwell Circuit Editor** will be exported with the circuit and are available for modifying when the circuit is imported into another **Ansys** application.

3. Optionally, you can select or clear the **Show Hidden** check box on any of the **Properties** window tabs.
4. Click **OK**.
    - Parameter Values
    - General
    - Symbol
    - Property Displays

    > **Note:** Variables may be assigned to parameter values in the circuit by entering a variable name in the parameter value field and assigning a value in the **Add Variable** dialog box. Variables assigned in **Maxwell Circuit Editor** will be exported with the circuit and are available for modifying when the circuit is imported into another **Ansys** application.

5. Optionally, you can select or clear the **Show Hidden** check box on any of the **Properties** window tabs.
6. Click **OK**.

The specific properties differ per component, as seen in the help topics describing each circuit component.

Each tab of the **Properties** window contains a list of parameters (each individual row) and properties you need to set for each parameter (each column is a property).

Callback Scripting Using PropHost Object

Callback scripts are scripts that can be set in the Property window for individual properties by clicking the button in the Callback column and choosing a script that is saved with the project.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

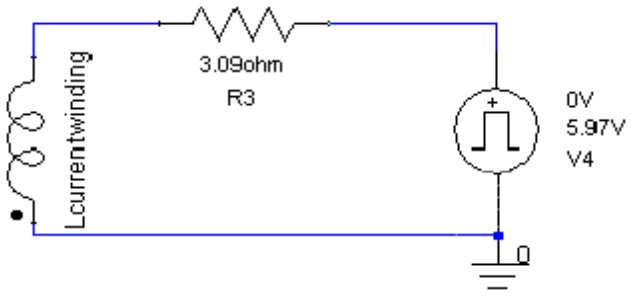
Callback scripts can contain any legal script commands including general Ansys script function calls (e.g., GetApplication()). In addition, they can call functions on a special object named PropHost. The PropHost represents the PropServer (owner of properties) that contains the Property that is calling the Callback script. Therefore, the Callback script can use the PropHost's functions to query or set other properties in the same PropServer. Refer to the Property Level scripting commands for more information.

Setting Up an External Circuit Using the Maxwell Circuit Editor

> **Note:** One use for an external circuit can be to supply an excitation to a coil terminal, rather than using a voltage type of excitation.

> **Note:** External circuits intended for use with the Time Decomposition Method (TDM) can only support basic circuit elements that are solution-independent. This is because, for TDM, all time steps (or all time steps in a subdivision) are solved simultaneously - and thus the circuit topology and its parameters must be known before solving. Therefore some circuit elements, such as solution-dependent switches, diodes, capacitors, and inductors which are not directly connected to windings in series, are not supported.

The driving circuit for the winding in this design consists of a voltage source in series with a resistor and with the winding. When complete, the circuit should look similar to the following figure.



To set up the external circuit, follow this general procedure:

1. Add the circuit elements.
2. Connect the circuit elements in series.
3. Export the netlist.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Save the Maxwell Circuit Editor project.
5. Assign the external circuit.

Add the Circuit Elements

To add the circuit elements in Maxwell Circuit Editor:

1. Click **Project** > **Insert Maxwell Circuit Design**.
   The circuit sheet appears.
2. Place the winding circuit element on the sheet:
   a. In the **Component Libraries**, under **Maxwell Circuit Elements** > **Dedicated Elements**, select the **Winding** element.
   b. Drag it onto the sheet.
   c. Right-click, and select **Finish** to place the component.
   d. To view the properties, double-click the component in the Schematic window.
   The **Properties** window appears.
   e. Change the **Name** to **currentwinding**, the same name you used when defining the winding in the Maxwell design.
   f. Click **OK**.
   g. Click **Draw** > **Rotate**, and position the winding vertically.
3. Place a resistor on the sheet:
   a. In the project tree, under **Passive Elements**, select **Resistor**.
   b. Drag the resistor onto the sheet.
   c. Right-click, and select **Finish** to place it where desired.
   d. Double-click the symbol of the resistor, change the value of the resistor, **R**, to **3.09**, keep the **Unit** value set to **ohm**, and click **OK**. The default is 100 Ohms.
4. Place a voltage pulse on the sheet:
   a. In the project tree, under **Sources** select a **VPulse** element (Pulse Voltage Source).
   b. Drag it to the sheet, and then right-click and select **Finish** to place it onto the sheet.
   c. Double-click the source element symbol on the sheet, and then specify the following source characteristics:

Parameter	Value	Description
V1	0	Initial voltage
V2	5.97	Peak voltage
Tr	0.001	Rise time
Tf	0.001	Fall time
Pw	1	Pulse width
Period	2	

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. Leave the other fields set to the default values, and click **OK**.

Connect the Circuit Elements in Series

To connect the circuit elements in series:

1. From within the Maxwell Circuit Editor, click **Draw** > **Wire**.
2. Click each terminal.
3. When done, place the Ground symbol: Click **Draw** > **Ground** (or click the **Ground** symbol on the toolbar), place the **Ground** symbol on the sheet, right-click, and select **Finish** to place the symbol.
4. Connect the ground to the circuit: Click **Draw** > **Wire**, and draw the final wire.

Export the Netlist

To export the netlist:

1. From within the Maxwell Circuit Editor, click **Maxwell Circuit** > **Export Netlist**.

   > **Note:** To view the netlist before exporting it, click **Maxwell Circuit** > **Browse Netlist**.

2. In the **Netlist Export** dialog box that appears, select the folder where you want to save the external circuit file.
3. Type a name for the circuit in the **File name** box.
4. Click **Save**.

The **Netlist Export** dialog box closes and the Maxwell Circuit Editor reappears.

Save the Maxwell Circuit Editor Project

To save the project and exit Maxwell Circuit Editor:

1. Click **File** >**Save**, type a name for the project, and click **Save** to save the Maxwell Circuit Editor project.
2. Click **File** > **Exit** to close the Maxwell Circuit Editor program.

Assign the External Circuit

To assign the circuit in the Maxwell design:

1. Click **Maxwell 2D** (or **3D**) > **Excitations** > **External Circuit** > **Edit External Circuit**.
2. In the **Edit External Circuit** dialog box that appears, click **Import Circuit Netlist**.
3. In the **Select File** dialog box that appears, select **Designer Net List Files (*.sph)** from the **Files of type** drop-down menu.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Browse to the location where you saved the circuit, select it, and click **Open** to import it.
5. Click **OK** to close the **Edit External Circuit** dialog box.

Renaming a Source in Maxwell Circuit Editor

To rename a source that is drawn in the Schematic:

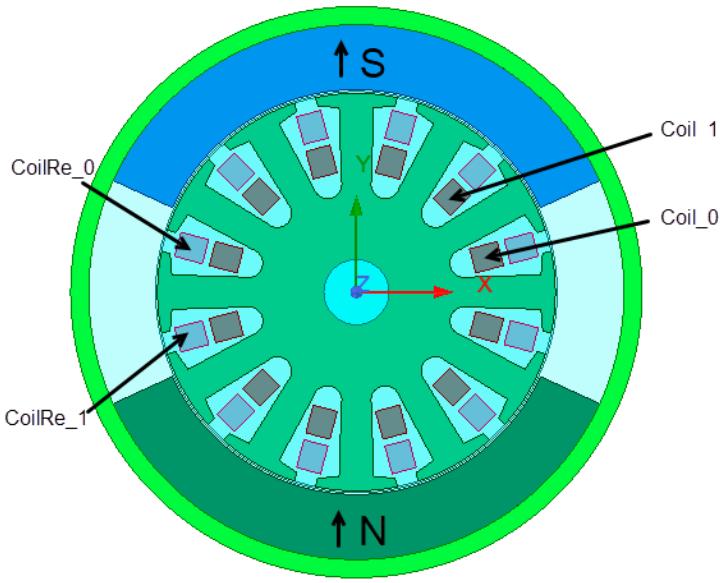
1. Double-click the source.
   The **Properties** window appears.
2. Click the **Parameter Values** tab (which should be the default tab visible).
3. Change the text value in the **Name** row.
4. Click **OK**.
   The new name appears for the source label in the Schematic.

Applying the Commutating Bar Element

This topic describes how to apply the commutating bar element to simulate the commutating process of brush-type DC machines. A two-pole 12-slot PMDC motor (shown below) is used as an example.

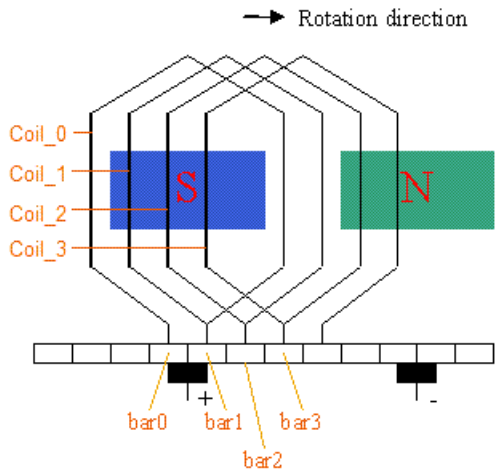
> **Note:** The example used here is based on the **stndj-1.aedt** project found in the <*install_dir*>/Examples/RMxprt/pmdc folder.

The flux direction inside the S pole permanent magnet is from the air gap to stator yoke, and that of the N pole is from the stator yoke to the air gap.



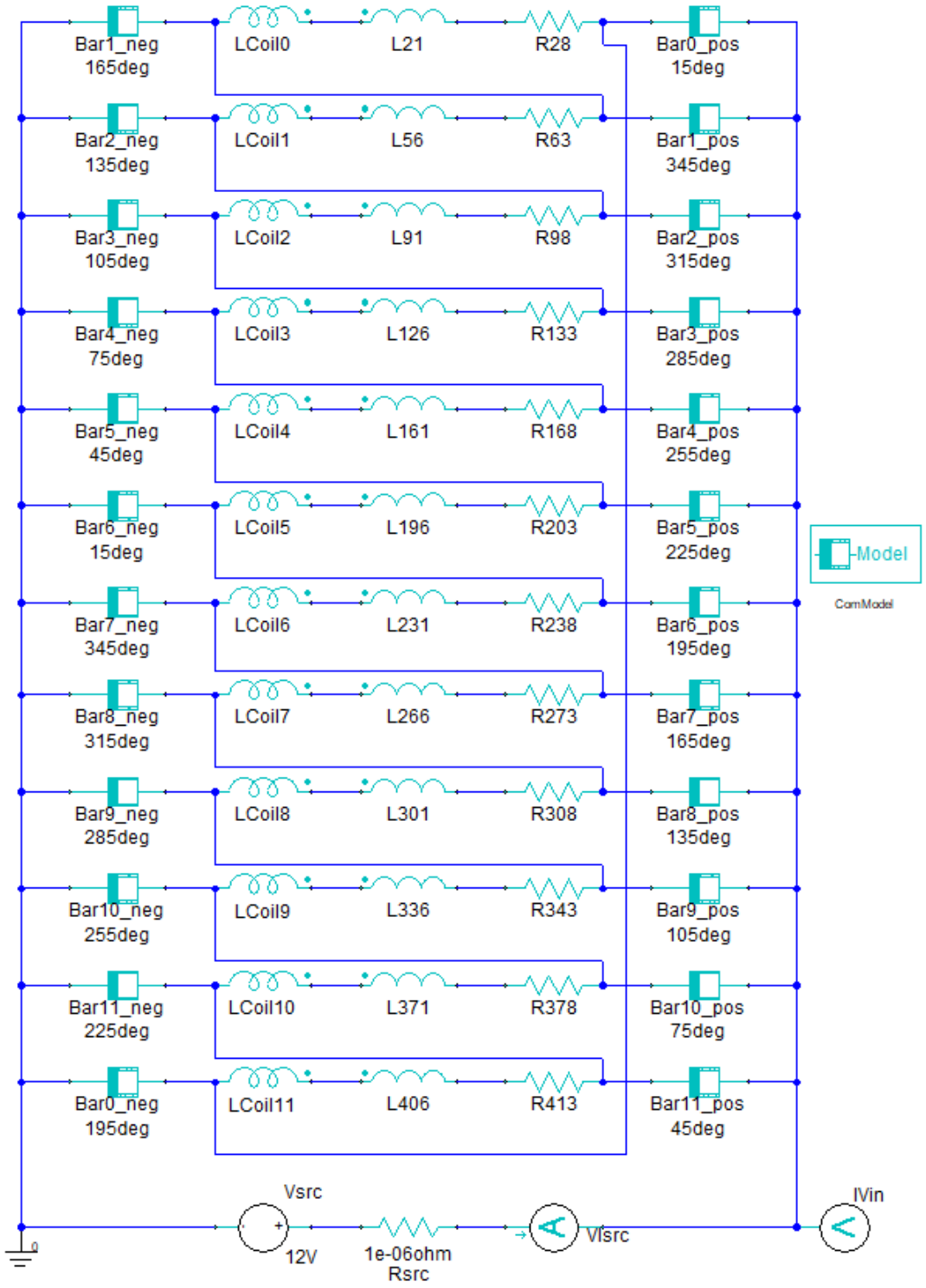
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The DC winding is of lap type with coil pitch of 5 slots. The flat-out extensional drawing of the motor indicating the relationship of permanent magnets, coils, commutating bars, and brushes at the initial position is shown below. With the rotation direction shown, the brush aligned with the S pole is positive, and the brush aligned with the N pole is negative. The "go" terminal of Coil_0 connects to bar0, and its return terminal connects to bar1; the "go" terminal of Coil_1 connects to bar1, and its return terminal connects to bar2; and so on.



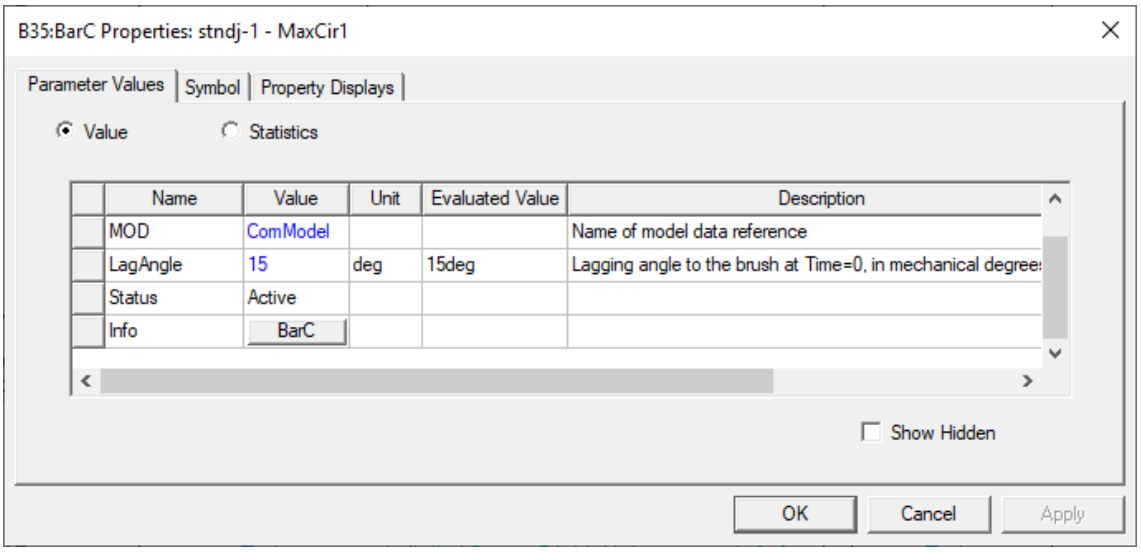
At the initial time, bar0 lags the positive brush by 15 mechanical degrees (a half commutating bar pitch), and it lags the negative brush by 195 mechanical degrees; bar1 lags the positive brush by -15 mechanical degrees, or 345 degrees, and it lags the negative brush by 165 mechanical degrees; and so on. The functional connection of each commutating bar with the positive brush or negative brush is modeled by BarC (commutating bar) elements, as shown below.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



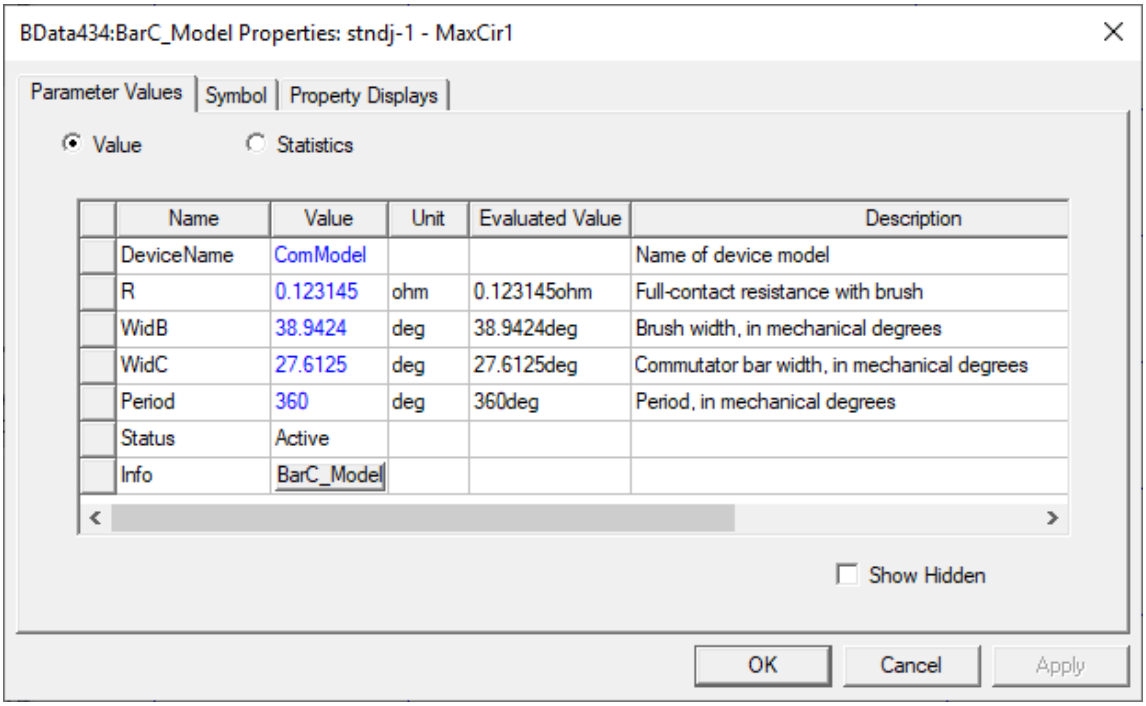
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Elements Bar0_pos through Bar11_pos represent the functional connections of bar0 to bar11 with the positive brush, respectively. Elements Bar0_neg through Bar11_neg give the functional connections of bar1 to bar11, and bar0 with the negative brush, respectively. The additional inductor in series with each coil inductance represents the end turn effect that is desirable to consider in a 2D model and possibly also in a 3D model that does not include the end turn geometry. The additional series resistor in series with each coil represents the global resistance of the coil and needs to be included in both 2D and 3D simulations. The values below the BarC element symbols are the respective lagging angles in mechanical degrees. The element parameters can be edited by double-clicking on the element, as shown below.



The input value "ComModel" for MOD is the model name that defines the parameters of commutating bars and brushes. The parameters of ComModel can be edited by double-clicking the BarC_Model element.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note: As a rule, each BarC element references a unique BarC_Model element.

In this design, the commutator diameter is 24 mm, and the brush width is 8 mm. Therefore, the brush width in mechanical degrees is the following:

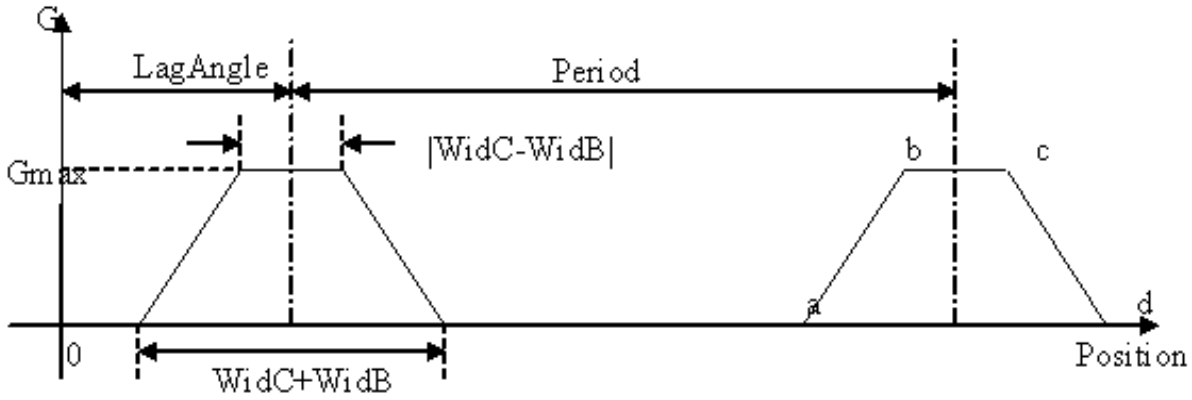
*WidB = 2 * arcsin(8/24) = 38.9 (deg)*

The number of commutator bars is 12 (the same as the number of slots), and the commutator insulation thickness is 0.5 mm. Therefore, the commutator bar width in mechanical degrees is the following:

*WidC = 360/12 - 2 * arcsin(0.5/24) = 27.6 (deg)*

The electric conductance between a commutating bar and a brush varies with the rotor position and can be derived from the model parameters, as shown below, where *Gmax = 1 / R*.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Electric conductance as a function of rotor position

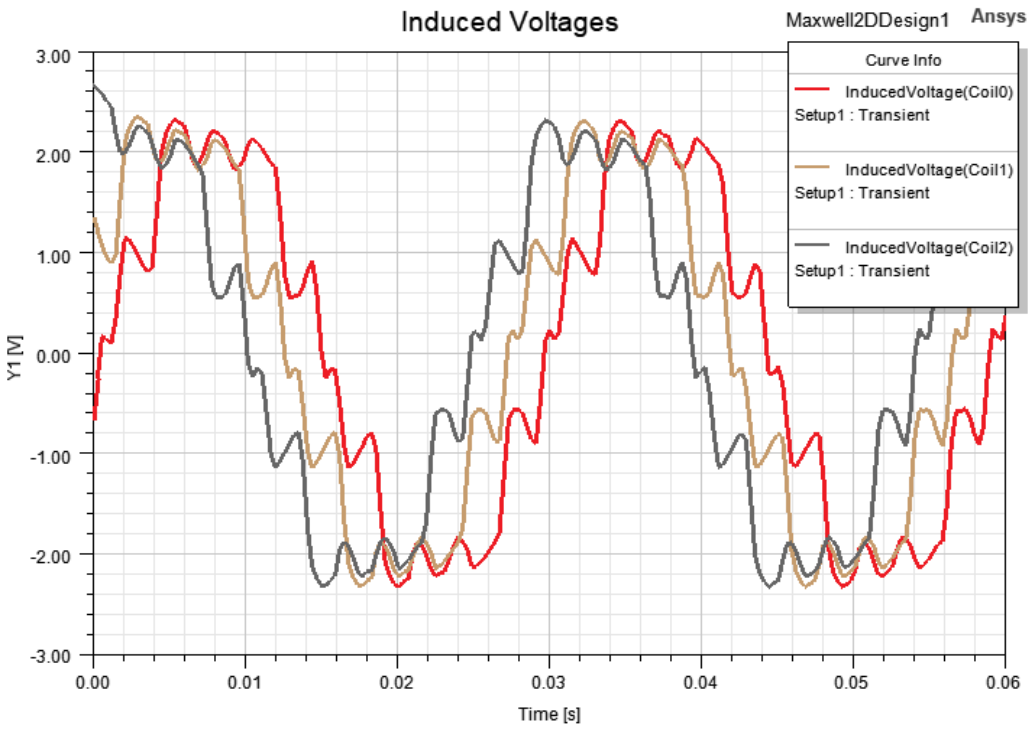
LagAngle is treated as a BarC element parameter because it is different for different commutating bars. All other values (Gmax, WidB, WidC, and Period) can be obtained from BarC_Model parameters. Positions a, b, c, and d correspond to the positions when one side of a commutating bar (solid color) aligns to one side of a brush, as shown below.

Different conducting position when WidB > WidC

Different conducting position when WidB < WidC

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The computed no-load induced voltages for three of the coils, taking into account the commutating process, are shown below.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28 - Maxwell 3D Technical Notes

The Ansys Maxwell® 3D interactive software package uses the finite element method (FEM) analysis to simulate and solve three-dimensional electromagnetic field problems. The Maxwell application integrates with other Ansys software to perform complex tasks while remaining easy to use.

Maxwell 3D Solutions

The Ansys Maxwell 3D® application supports the following five types of solutions:

- **Electric** 3D fields, which can fall in one of three categories:

    - Electrostatic 3D fields in dielectrics caused by a user-specified distribution of voltages and charges. Additional computed quantities you can specify include torque, force, and capacitances.
    - Electric 3D fields in conductors, characterized by a spatial distribution of voltage, electric field, and DC current density. The main additional quantity in this case is power loss.
    - A combination of the first two with conduction solutions being used as boundary conditions for an electrostatic problem.
- **Magnetostatic** linear and nonlinear 3D fields caused by a user-specified distribution of DC current density, voltage, permanent magnets, or externally applied magnetic fields. Additional computed quantities you can specify include torque, force, and inductances (self and mutual).
- **Harmonic (AC Magnetic)** (sinusoidal variation in time) steady-state 3D magnetic fields with pulsation-induced eddy currents in massive (solid) conductors caused by one of the following: by a user-specified distribution of AC currents (all with the same frequency but with possibly different initial phase angles), or by externally applied magnetic fields.

  The electric field solution is correct *inside* a conductor due to Ohm's law and self-consistent current densities (including current-bunching in solid conductors). *Outside* a conductor, Maxwell 3D's electric field permits some wave effects, such as propagation to a radiation boundary, but is limited in application and will ignore capacitance effect along source conductors (no charge-bunching). In general, there is no change in the electrical phase angle of the applied current source along the length of a conductor.
- **Transient Magnetic** (time domain) 3D magnetic fields caused by permanent magnets and windings supplied by voltage and/or current sources with arbitrary variation as functions of time; electrical circuits will be connected with the windings. Rotational or translational motion effects can also be included in the simulation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Transient Magnetic with A-Phi Formulation** (time domain) 3D magnetic fields caused by permanent magnets, voltages, currents, and windings supplied by voltage and/or current sources with arbitrary variation as functions of time; electrical circuits can be coupled with the field solution via the windings. Displacement current (capacitive) effect can also be included in the simulation.
- **Transient Electric** (time domain) 3D electric fields caused by time-varying voltages, charge distributions, or current excitations in inhomogeneous materials. The transient electric field simulator computes time-varying electric fields. Electric potential is the solution quantity.

The complex functionality built into the Maxwell 3D solvers is accessed through the main user interface (called the desktop). With the version 11 interface and later, you can model the problem in a fairly arbitrary order (rather than following the steps in a precise order as was required in previous versions of Maxwell).

This flexibility allows experienced users to develop a modeling style that suits their preferences. Once the model is created, the automated Maxwell solution sequence takes over and fully controls the solution process without any interaction from the user. When the solution becomes available, the user can perform a variety of postprocessing tasks as required by the design application.

Finite Element Analysis

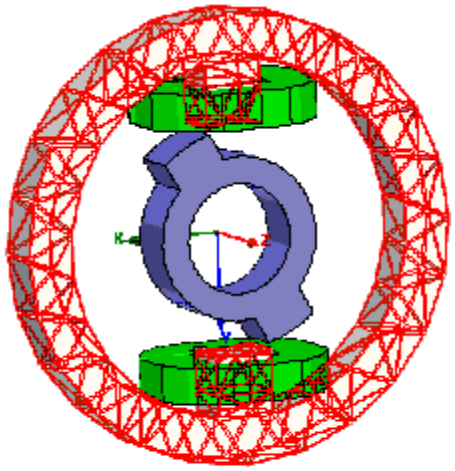
Finite element analysis (FEA) is a very sophisticated tool widely used by engineers, scientists, and researchers to solve engineering problems arising from various physical fields such as electromagnetic, thermal, structural, fluid flow, acoustic, and others. Currently, the finite element method is clearly the dominant numerical analysis method for the simulation of physical field distributions, with success not paralleled by any other numerical technique. In essence, the finite element method finds the solution to any engineering problem that can be described by a finite set of spatial partial derivative equations with appropriate boundary and initial conditions. It is used to solve problems for an wide variety of static, steady-state, and transient engineering applications from diverse markets such as automotive, aerospace, nuclear, biomedical, etc.

The finite element method has a solid theoretical foundation. It is based on mathematical theorems that guarantee an asymptotic increase of the accuracy of the field calculation toward the exact solution as the size of the finite elements used in the solution process decreases. For time domain solutions the spatial discretization of the problem must be refined in a manner coordinated with the timesteps of the calculation according to estimated time constants of the solution (such as magnetic diffusion time constant).

Maxwell solves the electromagnetic field problems by solving Maxwell's equations in a finite region of space with appropriate boundary conditions and – when necessary – with user-specified initial conditions in order to obtain a solution with guaranteed uniqueness. In order to obtain the set of algebraic equations to be solved, the geometry of the problem is discretized automatically into tetrahedral elements. All the model solids are meshed automatically by the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

mesher. The assembly of all tetrahedra is referred to as the finite element mesh of the model or the mesh. Inside each tetrahedron, the unknowns characteristic for the field being calculated are represented as polynomials of second order. Thus, in regions with rapid spatial field variation, the mesh density needs to be increased for good solution accuracy (see also adaptive mesh refinement).



Mesh shown on the outer armature of a rotational actuator

Types of Solutions and Degrees of Freedom (DOFs)

Solving an electromagnetic field problem is always based on solving Maxwell's equations. However, the process of obtaining the solution is typically based on solving a second-order consequence of Maxwell's equations with the consideration of applicable constitutive equations. At the same time – as a rule – a subset of complete Maxwell's equations is considered according to characteristic aspects of the application. Thus for reasons of efficiency of the solution, applications are classified as electrostatic, magnetostatic, frequency domain or time domain and as a consequence a specific type of solver is used in each case. This allows the users to obtain the solution with the desired accuracy but always within the limits of the fundamental assumptions made when the application was classified along the lines of the above mentioned criteria. The guidelines that can be used to correctly identify the type of solution to use are detailed in the topics that follow. The unknowns for each type of solution can be different, depending on the formulation.

Size of Mesh Vs. Accuracy

There is a trade-off among the size of the mesh, the desired level of accuracy, and the amount of available computing resources.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The accuracy of the solution depends on the size of each of the individual elements (tetrahedra). Generally speaking, solutions based on meshes using thousands of elements are more accurate than solutions based on coarse meshes using relatively few elements. To generate a precise description of a field quantity, each element must occupy a region that is small enough for the field to be adequately interpolated from the nodal values.

However, generating a field solution involves inverting a matrix with approximately as many elements as there are tetrahedra nodes. For meshes with a large number of elements, such an inversion requires a significant amount of computing power and memory. Therefore, it is desirable to use a mesh fine enough to obtain an accurate field solution but not so fine that it overwhelms the available computer memory and processing power.

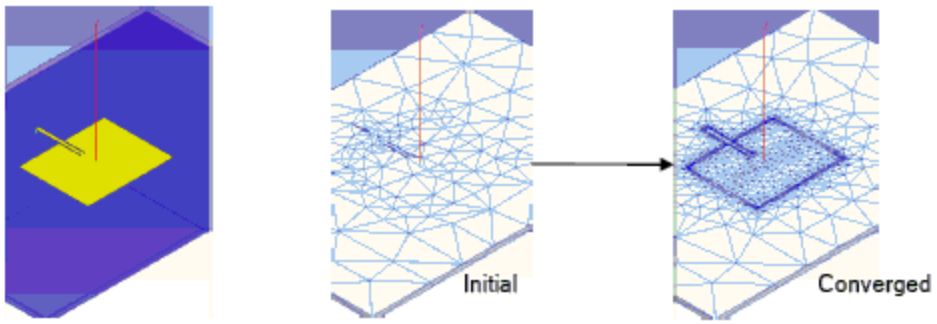
To produce the optimal mesh, Maxwell uses an iterative process, called an adaptive analysis, in which the mesh is automatically refined in critical regions. First, it generates a solution based on a coarse initial mesh. Then, it refines the mesh in areas of high error density and generates a new solution. When selected parameters converge to within a desired limit, Maxwell breaks out of the loop.

The Mesh Generation Process

The general mesh generation process is outlined below:

1. An initial mesh is generated and includes surface approximation settings. If necessary, the mesher will automatically perform any repairs needed to recover an accurate mesh representation of a model. The solution profile will indicate when mesh repairs have been made, and the results of these repairs will be displayed per object in the mesh statistics panel.
2. Any mesh operations that were defined are used to refine the mesh.
3. Using the resulting mesh, the electromagnetic fields are computed that exist inside the structure when it is excited at the solution frequency.
4. If you are performing an adaptive analysis, the current finite element solution is used to estimate the regions of the problem domain where the exact solution has strong error. Tetrahedra in these regions are refined.
5. Another solution is generated using the refined mesh.
6. The error is recomputed, and the iterative process (solve — error analysis — adaptive refinement) repeats until the convergence criteria are satisfied or the maximum number of adaptive passes is completed.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



7. If a frequency sweep is being performed, then the problem is solved at the other frequency points without further refining the mesh. An adaptive solution is performed only at the specified solution frequency.

> **Note:**
>
> An initial mesh is not generated each time it starts the solution process. The initial mesh is generated only if a current mesh is unavailable.

Seeding the Mesh

In Maxwell, mesh operations are optional mesh refinement settings that enable you to provide Maxwell with engineering guidance based on your knowledge of the parts of the model geometry that are critical to the structure's electromagnetic performance. Providing such guidance prior to beginning the adaptive analysis process can reduce (sometimes extensively) the number of passes necessary to converge upon a field solution as well as the final number of tetrahedra in the mesh for that solution. Although adaptive analysis convergence targets areas where field behavior is found, refining the mesh using more than the standard criteria, such as material characteristics, can result in finding areas of critical field behavior as soon as the first few passes are solved.

The technique of guiding the mesh construction is referred to as "seeding" the mesh. Seeding is performed using the **Mesh** commands on the **Maxwell** menu.

You can instruct Maxwell to refine the length of tetrahedral elements on a surface or within a volume until they are below a certain value (length-based mesh refinement) or you can refine the surface triangle length of all tetrahedral elements on a surface or volume to within a specified value (skin depth-based mesh refinement.) These types of mesh operations can be defined at any time. If you apply them before the adaptive solution process, they are used to refine the initial mesh after it has been generated. You can also choose to generate a mesh without solving, in which case mesh operations are applied to the current mesh.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In a few circumstances, you may also want to define a mesh operation that modifies the surface approximation settings for one or more faces. Surface approximation settings are only applied to the initial mesh.

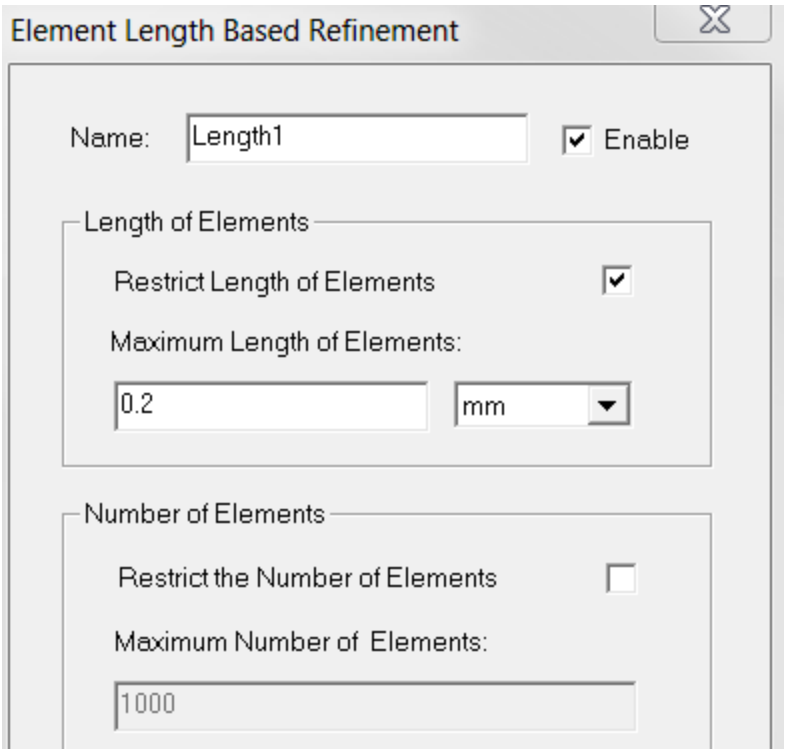
Guidelines for Seeding the Mesh

While seeding the mesh is not required, it is useful in the following conditions:

- Seeding the mesh inside a volume in the model geometry where regions of strong electric or magnetic fields (with strong capacitive or inductive loading) are expected. Examples include a capacitively loaded gap in a resonant structure, sharp waveguide angles or corners, or gaps between multi-coupled lines in filter structures.
- Seeding the mesh on every face of higher aspect ratio boundaries, such as long PCB traces or on the surfaces of long wires. Spacing the mesh points roughly equal to the trace width of the wire diameter enables you to more accurately capture the behavior of the high-aspect structure from the first adaptive pass.

Length-Based Mesh Refinement

Length-based mesh refinement controls the length of the tetrahedral elements and the growth of the whole mesh. You can set a ceiling on the mesh growth with the option **Restrict the Number of Elements** to ensure a mesh of reasonable size after the length refinement.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you define a mesh operation on a selection of objects (more than one) and set a maximum length, the operation occurs on the whole mesh. If you select just one object, when you refine its boundary, elements are added both inside and outside of that object.

When there are no more elements to refine or when no more elements can be added, the refinement process stops because it has either met the growth limit or the element length target. However, there are more quality improvement swaps, smoothing, and matching boundary issues, performed at the finishing stage that might slightly alter the mesh size.

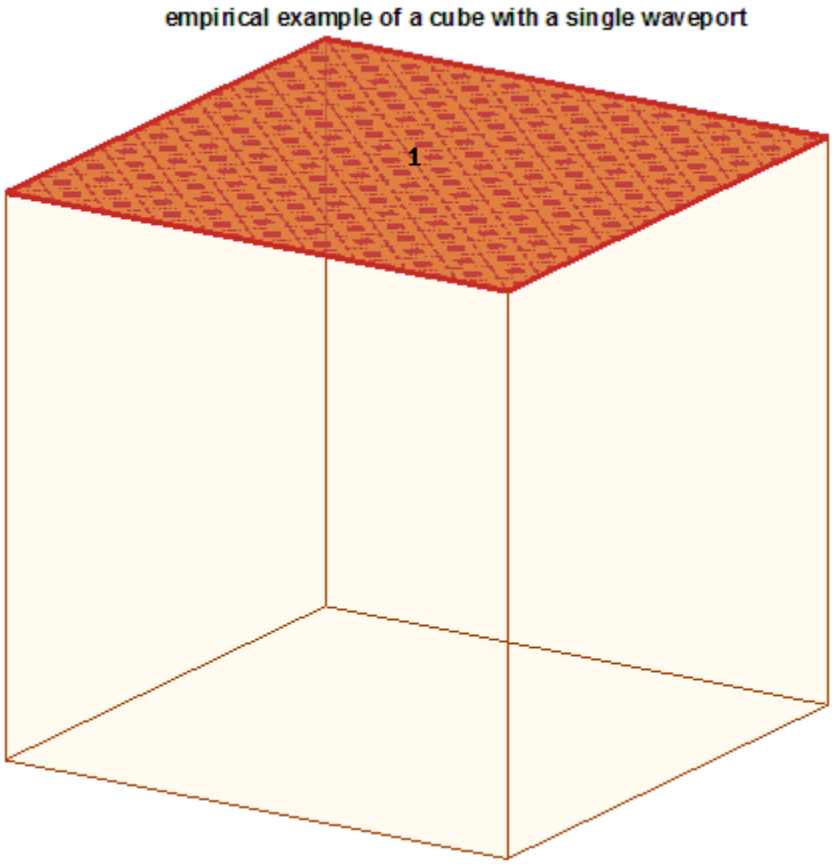
> **Important:**
>
> Be careful about specifying length targets for voluminous structures. This is because element count and target length have a cubic relationship. The relationship between length target and element count is inversely proportional, to the order of n^3 (that is length target = $1/n^3$) where ***n*** is the element count.

Restrict Length of Elements

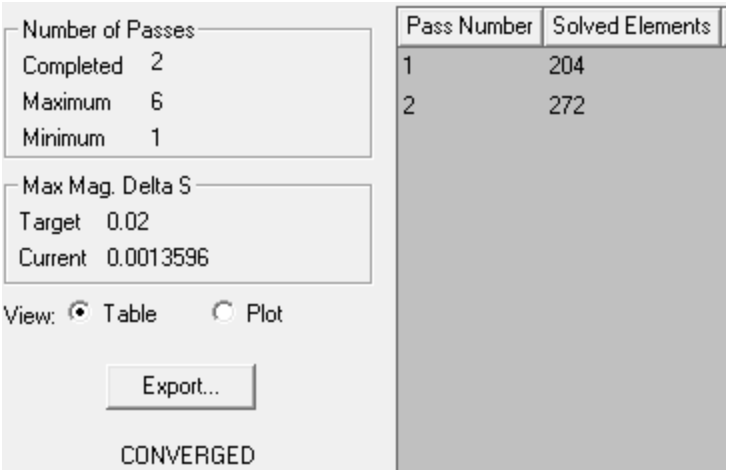
Selecting the **Restrict Length of Elements** check box enables you to set the stopping criteria for seed refinement. When the longest edge of a tetrahedral element under the specified value of **Maximum Length of the Tetrahedral Element** is reached, seed refinement stops.

The following example of a cube 1000 mm in size and material vacuum illustrates seed refinement when it is specified by a value of the **Maximum Length of The Tetrahedral Element** as the stopping criteria. The cube has a wave port assigned on one of its faces and the adaptive frequency is 0.25 GHz.

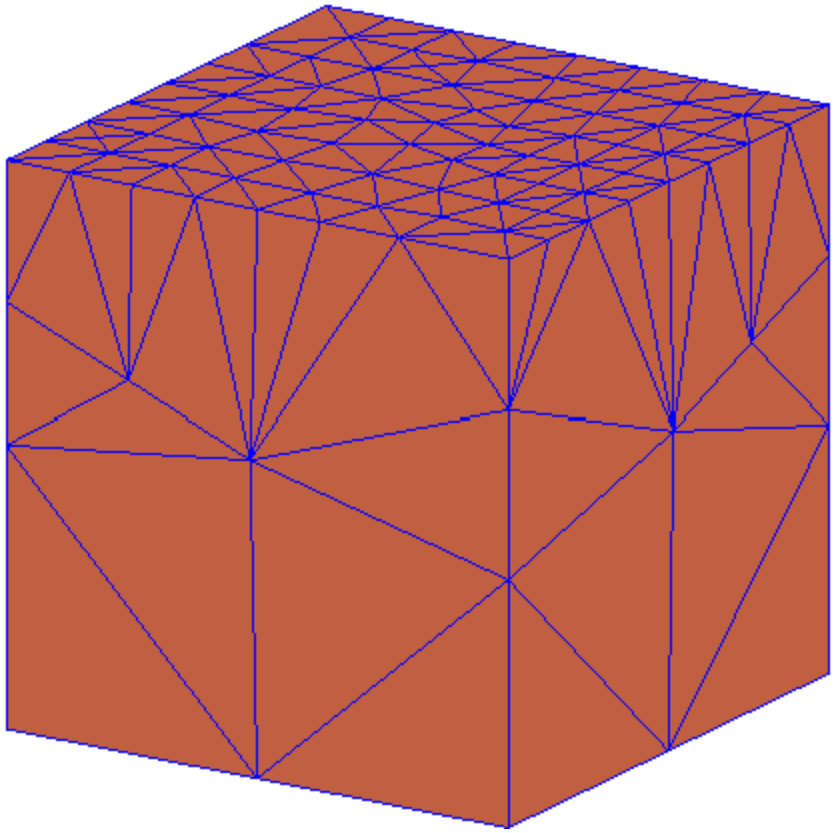
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



First the design is solved with no seed refinement. The mesh plot on the cube after convergence is shown in the following figures.

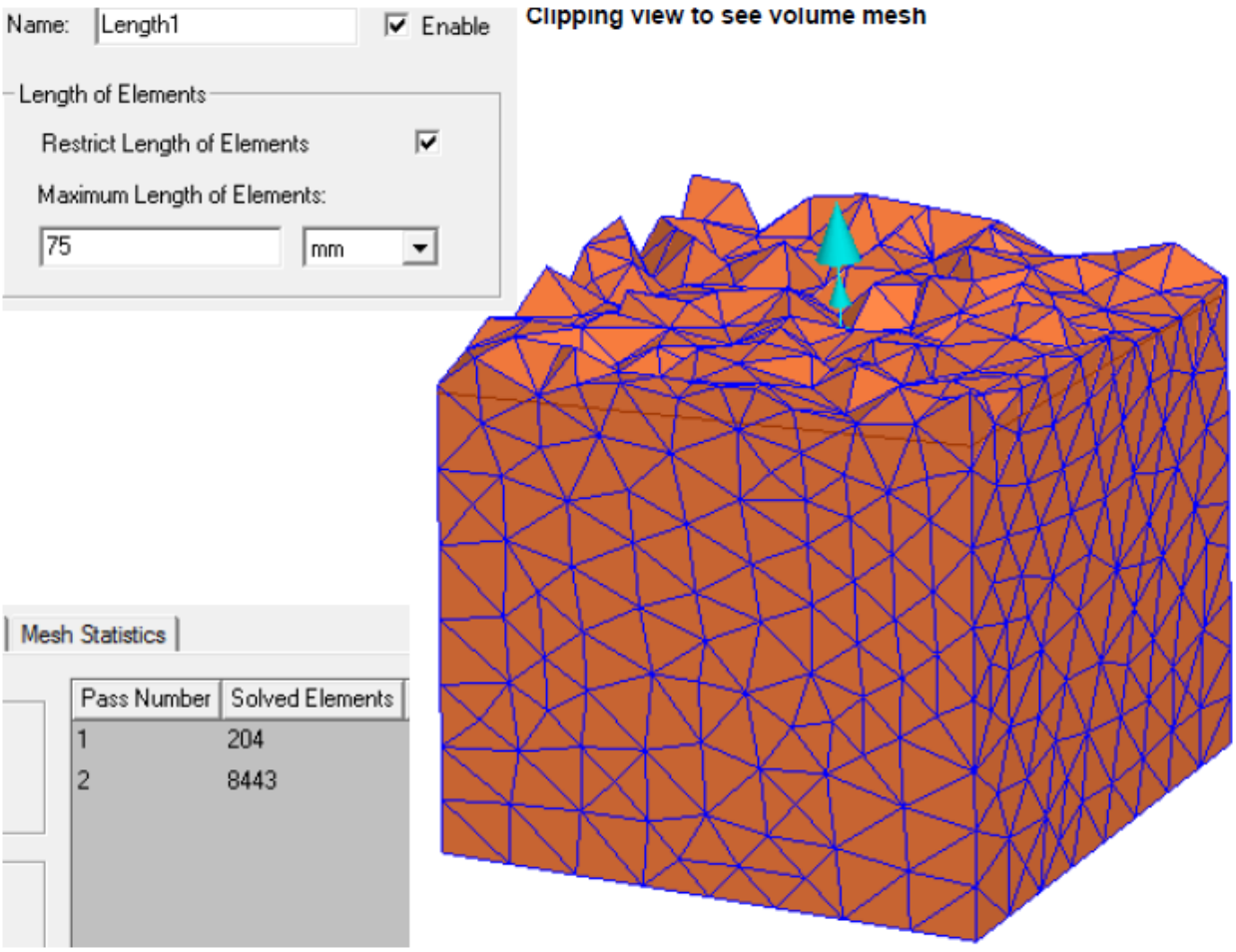


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A plane is drawn at the center of the cube and the normal to the plane is along the Z axis. Using the **Clipping** view feature, the cube is virtually clipped along the normal. After seed refinement, the mesh plot of the cube along the normal is shown in the following figure.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Skin Depth-Based Mesh Refinement

When you request skin depth-based mesh refinement, you instruct Maxwell to refine the surface triangle length of all tetrahedral elements on a face to within a specified value. A layered mesh is created based on the surface mesh. The layers are graded based on the skin depth and number of layers you specify.

During skin depth-based mesh refinement, Maxwell creates a series of layers that are planes parallel to the object face, and that are spaced within the specified skin depth. For each point on the surface of the face, a series of points (P0, P1, P2, ..., Pn) are added to the mesh, where n is the number of layers. P0 is the point on the surface and the distance from P0 to Pn is the skin depth. The points are spaced in a non-uniform manner, with the distance between them decreasing in a geometric progression, as you move from Pn to P0.

For example, if

Skin Depth:	12 mm

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of Layers of Elements: 4

then

Distance [P0,P1]:	0.8 mm
Distance [P1,P2]:	1.6 mm
Distance [P2,P3]:	3.2 mm
Distance [P3,P4]:	6.4 mm
Distance [P0,P4]:	0.8 + 1.6 + 3.2 + 6.4 = 12 mm

The skin depth-based refinement first satisfies the surface triangle edge length criterion, then introduces the series of points to each additional layer. If a limit has been placed on mesh growth, one of the following happens:

- The limit is set high enough to complete the skin depth refinement.
- The limit is set high enough to satisfy the surface triangle edge length criterion, but not high enough to complete the depth seeding.
- The limit is not set high enough to satisfy even the surface triangle edge length criterion.

Because refining by skin depth can add many seeding points, you should first refine the surface of the object using length-based mesh refinement to obtain an accurate count of the number of points HFSS will add when refining by skin depth. This allows you to reach the surface edge length criterion and approximate the number of elements in the mesh and the number of points on the surfaces before proceeding to skin depth seeding.

The refinement criteria you specified are used to refine the current mesh.

Surface Approximation Settings

Object surfaces in Maxwell may be planar, cylindrical or conical, toroidal, spherical, or splines. The original model surfaces are called *true surfaces*. To create a finite element mesh, Maxwell first divides all true surfaces into triangles. These triangulated surfaces are called faceted surfaces because a series of straight line segments represents each curved or planar surface.

For planar surfaces, the triangles lie exactly on the model faces; there is no difference in the location or the normal of the true surface and the meshed surface. When an object's surface is non-planar, the faceted triangle faces lie a small distance from the object's true surface. This

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

distance is called the *surface deviation*, and it is measured in the model's units. The surface deviation is greater near the triangle centers and less near the triangle vertices.

The normal of a curved surface is different depending on its location, but it is constant for each triangle. (In this context, "normal" is defined as a line perpendicular to the surface.) The angular difference between the normal of the curved surface and the corresponding mesh surface is called the *normal deviation* and is measured in degrees.

The *aspect ratio* of triangles used in planar surfaces is based on the ratio of circumscribed radius to the in-radius of the triangle. It is unity for an equilateral triangle and approaches infinity as the triangle becomes thinner.

You can modify the surface deviation, the maximum permitted normal deviation, and the maximum aspect ratio of triangles settings on one or more faces at a time in the **Surface Approximation** dialog box. (Click **Maxwell** > **Mesh Operations** > **Assign** > **Surface Approximation**.)

The surface approximation settings are applied to the initial mesh.

> **Note:**
>
> For the initial mesh, all the vertices of the triangles lie on the true surfaces. During mesh refinement all the points are first added to the faceted surface for all products, not to the true surfaces. Then, for all Maxwell projects and for curvilinear meshing enabled HFSS projects , the points on the faceted surfaces will be moved to the curved surface locations when such movement does not degrade the quality of the mesh.

Guidelines for Modifying Surface Approximation Settings

If you intend to modify the surface approximation settings for an object face or faces, keep the following guidelines in mind:

- When necessary, override the default surface approximation settings to represent curved surfaces more accurately. More accurate representation will increase the mesh size and consume more CPU time and memory. The default settings are adequate for most circumstances.
- If you want to obtain a faster solution by using a cruder representation of curved surfaces, set the coarser setting for the whole object, not just a single face.
- It is difficult to satisfy aspect ratio demands if the aspect ratio value is set close to 1 because an arbitrary shape cannot be filled with only equilateral triangles. Therefore, setting the aspect ratio to 1 can lead to unreasonably large meshes. The aspect ratio is limited to 4 for planar objects and 1.2 for curved objects.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

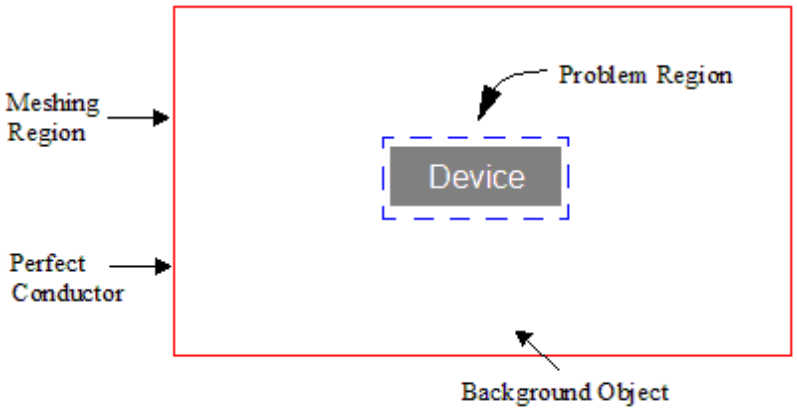
Meshing Region Vs. Problem Region

Maxwell distinguishes between the problem region and the meshing region. The *problem region* is the region in which the solution is generated and the mesh is refined. The *meshing region*, which includes the problem region, is the area in which an initial mesh is generated. After an initial mesh is generated, the mesh is refined only in the problem region.

The problem region encompasses an area that is just large enough to include the entire design, but no larger. Maxwell automatically defines the problem region during the solution process. If you are interested in effects outside of the structure, such as radiated effects, then you can create a virtual object to expand the size of the problem region to include these areas.

The meshing region, like the problem region, is a box that completely encloses the structure. However, the meshing region must be at least 10 times larger than the model. The part of the meshing region not occupied by objects is considered to be the *background* object. The background extends to the boundaries of the meshing region and fills in any voids not occupied by objects. Because the background object is defined as a perfect conductor, no solution is generated inside the background even though an initial mesh is generated for it. Maxwell automatically defines the meshing region during the solution process.

The problem region and the meshing region are illustrated below.



Model Resolution

Model Resolution is a setting that determines the smallest details of a model that the mesher should capture and represent in the mesh.

Many times the analysis starts with the geometry already drawn in a different tool for different purpose. Some tools are designed for manufacturing and the resulting models contain lots of extra details not needed for electromagnetic analysis. If the user removes such details in the original tool the results will be better. But if the user does not have access to the original drawing

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tool or redrawing the model without these details is not possible, **Model Resolution** is another way to remove the details from analysis.

When the user sets the model resolution length to be L, the mesher will start with a surface representation of the model accurate to the modeler's tolerance limit. Then it will progressively remove edges, move points, merge points etc., within the allowable model resolution limit and simplify the surface mesh. During this process, tiny fillets, rounds, and chamfer protrusions are removed.

Other common model translation anomalies are also handled using **Model Resolution**. For example, some geometry engines will blindly export all of the surfaces as splines. When a user imports such a model for analysis, it would result in very large number of triangles. If the surface can be represented by a smaller set of triangles using Model Resolution, this procedure would reduce the number of triangles in the surface mesh.

The user can start with a model resolution length around 0.1*wavelength. If the model resolution length chosen by the user is too large, the mesher will detect it and report it as an error. The model resolution length is specified in the user units of the modeler. It can be set on selected bodies only. The default value is 100* the tolerance limit of the modeler.

Electric Field Calculation

The Electric field solver uses the electric scalar potential values as DOFs. There are two basic types of static electric field solutions, as well as the combination of these two:

- For the electrostatic solution, it is assumed that all objects are stationary (velocity is zero everywhere in the space where the electrostatic field is calculated), there is no time variation of any of the electromagnetic quantities ($\frac{\partial}{\partial t} = 0$) and there is no current flow in conductors (conductors are in electrostatic equilibrium) and thus Joule losses are zero everywhere. All conductors are considered to be perfect and equipotential such that there is no electric field inside conductors. The unknowns (DOFs) are electric scalar potentials at all the nodes of the tetrahedra of the mesh. The distribution of conductivity in the space of the problem is irrelevant, only the distribution of permittivity is relevant for the solution process.

  > **Note:** For the electrostatic solution, if two or more conductors touch each other, they should be united into one object by using the Modeler Boolean operation "Unite" before analysis.

- For the DC current flow solution, only the conductors (with electric conductivity σ > Insulator/Conductor Threshold) are considered in the solution process. The DOFs are also the electric scalar potentials at the nodes of the mesh. Here the distribution of electric conductivity is relevant for the solution process while the permittivity is irrelevant. There is DC current flow in conductors (conductors are not equipotential) and there is non-zero

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Joule loss (ohmic power loss) in the conductors (which are part of an excited current path). The conductors are considered to be immobile (velocity is zero everywhere). The magnetic field associated with the DC current flow is not calculated and is totally decoupled from the calculated electric field distribution in conductors.

- The combination of the DC current flow and electrostatic solution is based on the division of the arrangement into conductors and insulators. Conductors are those objects whose electric conductivity σ is greater then or equal to the Insulator/Conductor threshold, and insulators are those whose conductivity σ is less then the Insulator/Conductor threshold. The solution of such problems is performed in two steps: first the DC conduction problem in the conductors is computed, then the electrostatic solution in insulators is calculated using the electric scalar potential of the conductors as a voltage boundary condition.

For the electric field solution, there are ten nodal DOFs (electric scalar potential unknowns) associated with each tetrahedron at each of the four vertices and at each of the six mid-edge nodes. Therefore, a quadratic approximation of the electric scalar potential inside each tetrahedron can be obtained.

The **electrostatic field solver** solves a Poisson type partial differential equation for the electric potential unknown, Φ, with appropriate boundary conditions:

$$\nabla \bullet (\varepsilon_r \cdot \varepsilon_0 \nabla \Phi) = -\rho_v$$

where

- Φ(x,y,x) is the electric potential scalar function of position.
- ε_r(x,y,x) is the relative permittivity, which is a function of position and can be isotropic or orthotropic. ($\varepsilon_0 = 8.854 \text{x} 10^{-12}$)
- F/m is the permittivity of a vacuum.
- ρ_v(x,y,x) is the volume density of the electric charge.

Once the electric potential is obtained by solving the above second order differential equation in the domain of the problem, Maxwell's equations and the applicable constitutive equation are then used to obtain the electric field strength, $\vec{E}$, and electric flux density, $\vec{D}$, vectors, as follows:

$$\vec{E} = -\nabla \Phi$$

$$\vec{D} = \varepsilon_r \cdot \varepsilon_0 \cdot \vec{E}$$

Typical sources for electrostatic problems are net **charges** (assumed to have a uniform distribution) applied to perfect insulator model objects or on surfaces that cannot touch conductors and **voltages** (electric potential applied to perfect conductor model objects or on surfaces, also called a Dirichlet boundary condition). Additionally, a **floating** boundary condition can be applied to perfect conductors (surrounded by insulators) or to surfaces surrounded by perfect insulators.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Volume charge density is another allowed excitation, which can have a spatial distribution specified by the user and can be applied to insulating objects only that do not touch conductors.

At the interface between two dielectrics, the normal component of the electric flux density vector has no jump if the charge density at the respective interface is zero:

$$\varepsilon_1 \cdot E_{n1} = \varepsilon_2 \cdot E_{n2}$$

where the permittivity and normal electric field values are considered in the two dielectrics and are designated by the subscripts 1 and 2. Alternatively, when crossing a surface with non-zero charge density, the normal component of the electric flux density has a jump equal to the respective local superficial charge density. It follows that in dielectrics (such as air, for example) in the immediate vicinity of conductive objects (perfect conductors), the (normal) component of the electric flux density is equal to the neighboring local charge density on the metallic surface (since in metallic objects the field value is zero).

> **Note:** When voltages are applied to conductors, the charge transferred from the DC source(s) to the conductors almost instantly receives a superficial distribution. For real (good) conductors, the time constant for the diffusion of the charge is of the order of 10^{-19} s. Therefore, for all practical instances, the charge relaxation process is considered to occur instantly. Once the charge on conductors reaches the steady state (almost instantaneously), the electric field in the neighboring dielectrics has a distribution that is governed by the laws of electrostatics, and the field in conductors is null while the field distribution in dielectrics is dictated – as far as material properties are concerned – by the respective permittivities and is independent of the conductivity of the electric conductors. This is why, in electrostatics, real conductors can be considered as perfect conductors.

Electrostatic regime of the electromagnetic field is rarely achieved in practice rigorously speaking. However, very good approximations can be used for practical situations within the limits specified above.

The DC current flow solver solves the equation $\nabla \cdot (\sigma \nabla \Phi) = 0$

where

- Φ(x,y,x) is the electric potential scalar function of position.
- σ(x,y,z) is the electric conductivity (in SI it is measured in S/m), which is a function of position and can be isotropic or orthotropic.

Once the electric potential is obtained by solving the above second order differential equation in the domain of the problem, Maxwell's equations and the applicable constitutive equation (Ohm's law in a local formulation) are

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

then used to obtain the electric field strength and the electric current density $\vec{J}$ vectors:

$$\vec{E} = -\nabla\Phi$$

$$\vec{J} = \sigma \cdot \vec{E}$$

Typical sources for DC current flow problems are **currents** applied on surfaces of conductors and **voltages** (electric potential applied to surfaces of conductors). The direction of the applied current is either "in" or "out", always normal to the respective surfaces.

> **Note:** Multiple conduction paths are allowed. Each conduction path that has a current excitation must also have either a voltage applied or a **sink** to ensure a unique solution.

The sink is a particular excitation (without a numerical value assigned to it) that is used to ensure the divergence-free character of the calculated DC current density vector in each conduction path.

At the interface between two different conductors, the normal component of the current density vector is continuous: $\sigma_1 \cdot E_{n1} = \sigma_2 \cdot E_{n2}$

For DC current flow problems, the solution is dictated everywhere inside the conductors by the distribution of electric conductivity, as far as material properties are concerned.

Charge, voltage, floating, current, and sink type excitations are called "terminal" type excitations since they can be referenced in a matrix extraction setup.

In perfect conductors, no computation occurs, and the electric scalar potential (the unknown in the problem) has a uniform value. However, in the post-processing phase, the (constant) distribution of the electric scalar potential (voltage) is correctly rendered. No solution is available in solids explicitly excluded from the solution.

The solution of any field problem is only possible if appropriate boundary conditions have been set. The boundary conditions have three purposes:

- Boundary conditions are always necessary from a mathematical perspective in order to ensure the uniqueness of the solution calculated by Maxwell.
- Boundary conditions occasionally represent a convenient way of modeling different ideal situations. For example, in order to model the field in a dielectric sandwiched between two very thin conducting objects (the thickness is at least two orders of magnitude less than the other relevant dimensions of the respective conducting objects), only the two respective surfaces need to be modeled on either side of the dielectric object. In the setup, those top and bottom surfaces carry appropriate boundary conditions, and the field in the dielectric is correctly simulated without having to draw the respective conductors.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Boundary conditions also occasionally provide a convenient way of taking advantage of symmetry situations. In such cases, the models must have the same geometric and material properties, as well as electric symmetry.

Typical boundary conditions include voltage, floating, and insulation boundary conditions. In the case of an **insulating boundary condition**, a conductor must exist at least on one of the sides of the respective surface.

The default boundary condition applied to all faces of the boundary of the electric field (static or stationary) problem is a Neumann homogeneous condition. This condition enforces an electric field tangent behavior of the solution in the immediate vicinity of the boundaries. There is no need to apply any particular boundary condition at the interface between objects with different material properties, as the solution automatically ensures – in average – the continuity of the tangential component of the electric field strength $(\vec{E})$ and the continuity of the normal component of the electric flux density vector $(\vec{D})$ across surfaces with zero superficial electric charge density (for electrostatic problems).

Symmetry for an Electric Field Solution

A symmetry boundary models a plane of symmetry in a structure. Use this type of boundary condition to take advantage of both geometric symmetry and electric symmetry. Doing so enables you to reduce the size of your model – allowing you to conserve computing resources. Two types of symmetry boundaries are available:

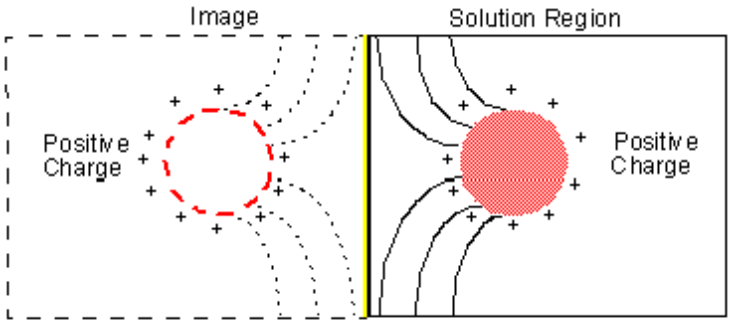
- Even Symmetry (Flux Tangential)
- Odd Symmetry (Flux Normal)

These boundaries can only be assigned to the outside edges of the solution region.

Even Symmetry (Flux Tangential) for an Electric Field Solution

Use an even symmetry boundary to define a plane of symmetry where the signs (positive or negative) of the voltages and charges on one side of the plane are the same as those on the other side. Electric flux is tangential to the boundary and thus does not cross it. To define an even symmetry boundary, the simulator sets the selected edge to a Neumann boundary – acting as an electrical mirror to the model.

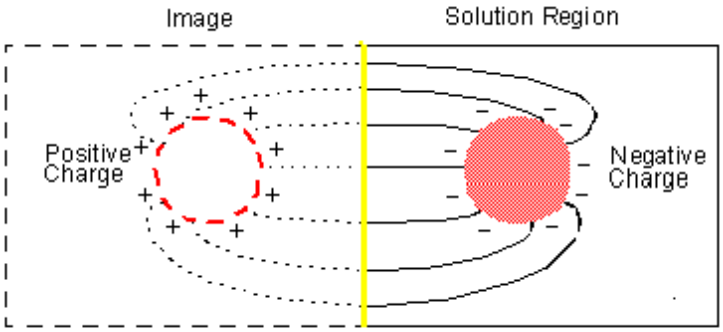
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The E-field is tangential to an even symmetry boundary.

Odd Symmetry (Flux Normal) for an Electric Field Solution

Use an odd symmetry boundary to define a plane of symmetry where the signs (positive or negative) of all charges and voltages on one side of the plane are the opposite of those on the other side. Electric flux is normal to the boundary. To define an odd symmetry boundary, the simulator sets the selected edge to a voltage boundary with a potential of zero volts.



The E-field is perpendicular to an odd symmetry boundary.

Matching Boundary for an Electric Field Solution

Matching boundaries allow you to model planes of periodicity where the E-field on one surface matches the E-field on another. They are very useful for modeling devices such as motors, in which the electric field repeats every 180°, 120°, 90°, or less. They enable you to model the smallest possible periodic segment of the device – reducing the amount of computing resources needed during the solution.

To set up matching boundaries, you must create an independent boundary and a dependent boundary. Unlike symmetry boundaries, **E** does not have to be tangential or normal to these boundaries. The only condition is that the fields on the two boundaries must have the same magnitude and direction (or the same magnitude and opposite directions).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Independent Boundary for an Electric Field Solution

The simulator computes the electric field on an independent boundary using the charges and voltages that you specified for the model as input. No other special conditions are imposed.

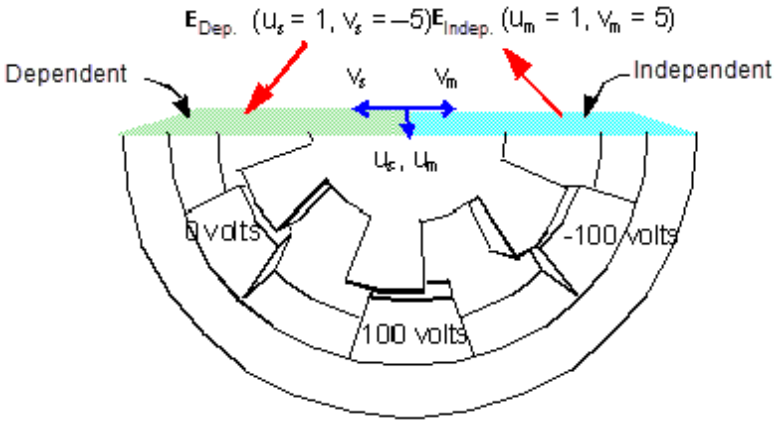
Dependent Boundary for an Electric Field Solution

The electric field on the dependent boundary is forced to match the field on the independent boundary. The magnitude of the electric field on both boundaries is the same. The fields on the two boundaries can either point in the same direction, or in opposite directions.

When to Use Matching Boundaries for an Electric Field Solution

Consider a simple electrostatic micromotor in which the rotor is held at zero volts and the six stator poles are switched between zero volts, 100 volts, and -100 volts. The E-field pattern at any point in time repeats itself every 180° — causing the field in one half of the motor to match the field in the other half.

If you use matching boundaries, you only need to model half of the motor, as shown below. The E-field on the dependent boundary (the left side of the motor) is forced to match the magnitude and point in the opposite direction from the E-field on the independent boundary (the right side of the motor) - simulating the field pattern that would occur if the entire motor was modeled.



A symmetry boundary cannot be used in place of matching boundaries in this example. The electric field is not necessarily either perpendicular or tangential to the motor's periodic surfaces. In the example above, the electric field would be exactly tangential to the periodic surface only when the poles of the rotor are aligned with the poles of the stator. In the other positions of the rotor, the field is not tangential and matching boundaries are required.

A **combined DC current flow + Electrostatic solution type** is also possible. In this case, the voltage distribution on conductors from the DC current flow solution is used as a Dirichlet

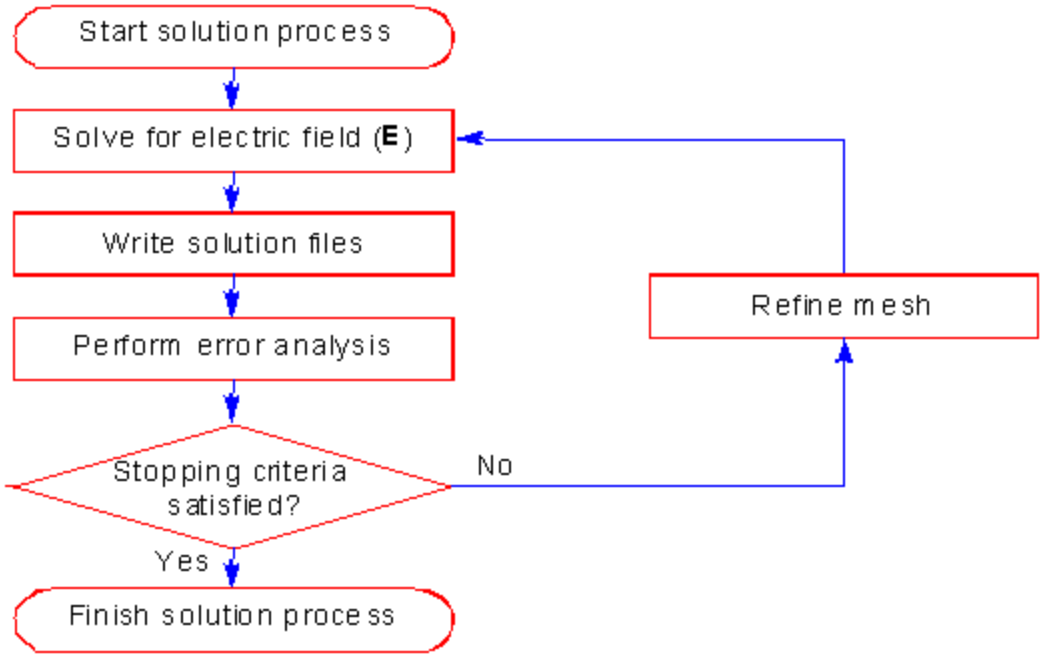
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(applied voltage) boundary condition in the electrostatic phase of the solution process. This type of sequence allows a certain class of problem involving electrostatic fields surrounding conductors with DC current flow to achieve a comprehensive solution. In this case, the solution provides the electric field in conductors and dielectrics.

Solution Process for an Electric Field Solution

Maxwell's electrostatic solver computes and stores the value of the electric potential at the vertices and midpoints of the edges of each tetrahedron in the finite element mesh. It solves for the electric field using the following relationship: $\vec{E} = -\nabla\Phi$.

After *E* is calculated, Maxwell writes out solution files and performs an error analysis. In an adaptive analysis, it refines the tetrahedra with the highest error, and continues solving until the stopping criterion is met. In a non-adaptive solution, the process stops.



Electric Field Energy for an Electric Field Solution

The energy, U, that is stored in a static electric field is given by the following equation:

$$U = \frac{1}{2}\int_{Vol} \vec{\mathbf{E}} \bullet \vec{\mathbf{D}} dVol$$

where

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **E** is the electric field.
- **D** is the electric flux density.

Capacitance Matrix for an Electric Field Solution

Capacitance matrices can be expressed in terms of charge and voltage, or current and time-varying voltage. The unit of capacitance is F.

> **Note:** For a capacitance matrix, the elements are defined based on excitations, not solids.

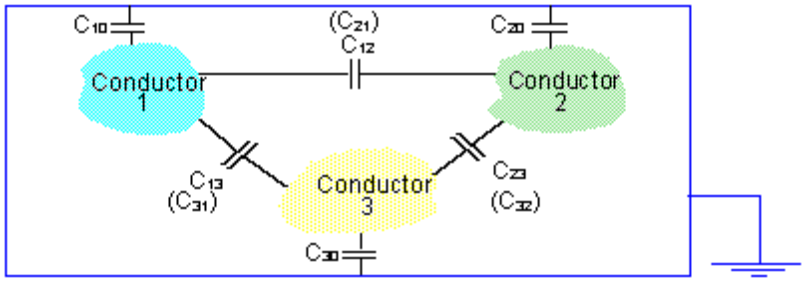
Capacitance in Terms of Charge and Voltage

A capacitance matrix represents the charge coupling within a group of conductors - that is, the relationship between their charges and voltages. Given a ground reference for the three conductors below, the net charge on each object is

$$Q_1 = C_{10}V_1 + C_{12}(V_1 - V_2) + C_{13}(V_1 - V_2)$$

$$Q_2 = C_{20}V_2 + C_{21}(V_2 - V_1) + C_{23}(V_2 - V_3)$$

$$Q_3 = C_{30}V_3 + C_{31}(V_3 - V_1) + C_{32}(V_3 - V_2)$$



This can be expressed in matrix form as the following:

$$\begin{bmatrix} Q_1 \\ Q_2 \\ Q_3 \end{bmatrix} = \begin{bmatrix} C_{10} + C_{12} + C_{13} & -C_{12} & -C_{13} \\ -C_{12} & C_{20} + C_{12} + C_{23} & -C_{23} \\ -C_{13} & -C_{23} & C_{30} + C_{13} + C_{23} \end{bmatrix} = \begin{bmatrix} V_1 \\ V_2 \\ V_3 \end{bmatrix}$$

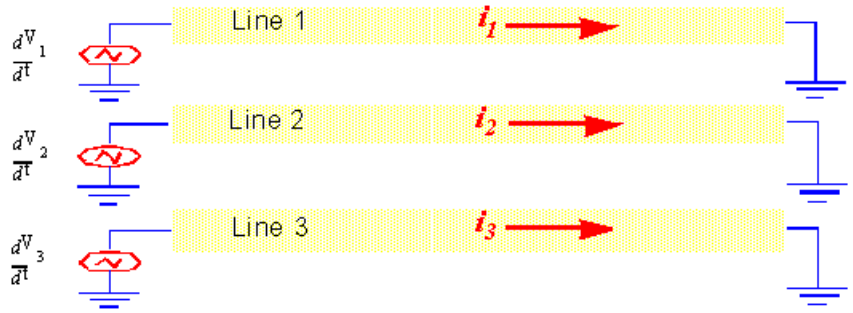
The capacitance matrix above gives the relationship between Q and V for the three conductors and ground. An n-conductor device would have an n x n capacitance matrix.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Capacitance in Terms of Current and Time Varying Voltage

A capacitance matrix can also represent the relationship between currents and time varying voltages in a system of conductors. Given the three transmission lines shown below, the currents caused by the time varying voltage source on each line are given by the following equation:

$$\begin{bmatrix} i_1 \\ i_2 \\ i_3 \end{bmatrix} = \begin{bmatrix} C_{10}+C_{12}+C_{13} & -C_{12} & -C_{13} \\ -C_{12} & C_{20}+C_{12}+C_{23} & -C_{23} \\ -C_{13} & -C_{23} & C_{30}+C_{13}+C_{23} \end{bmatrix} \begin{bmatrix} \frac{dV_1}{dt} \\ \frac{dV_2}{dt} \\ \frac{dV_3}{dt} \end{bmatrix}$$



If dV_2/dt and dV_3/dt are set to zero, this relationship becomes the following:

$$\begin{bmatrix} i_1 \\ i_2 \\ i_3 \end{bmatrix} = [C] \begin{bmatrix} \frac{dV_1}{dt} \\ 0 \\ 0 \end{bmatrix} = \begin{bmatrix} C_{10}+C_{12}+C_{13} \\ -C_{12} \\ -C_{13} \end{bmatrix} \frac{dV_1}{dt}$$

This gives the currents that are induced on Line 2 and Line 3 when a time varying voltage source is applied to Line 1 – that is, the capacitive coupling between the three lines, or the short circuit capacitance.

The capacitance matrix can be exported to a circuit/system simulator, such as the Twin Builder application, where it can be used in a transient simulation.

Matrix Elements for an Electric Field Solution

If one volt is applied to Conductor 1 and zero volts are applied to the other two conductors, the capacitance matrix becomes the following:

$$\begin{bmatrix} Q_1 \\ Q_2 \\ Q_3 \end{bmatrix} = [C] \begin{bmatrix} 1 \\ 0 \\ 0 \end{bmatrix} = \begin{bmatrix} C_{10}+C_{12}+C_{13} \\ -C_{12} \\ -C_{13} \end{bmatrix}$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Diagonal Elements for an Electric Field Solution

These terms (such as C_{11}) are the sum of all capacitances from one conductor to all other conductors. These terms represent the self-capacitance of the conductors. Each is numerically equal to the charge on a conductor when one volt is applied to that conductor and the other conductors (including ground) are set to zero volts. For instance, $C_{11} = C_{10} + C_{12} + C_{13}$. This term is equal to the charge on Conductor 1 when it is set to one volt and the other conductors are set to zero volts.

Off-Diagonal Elements for a Capacitance Matrix

The terms in each column (such as C_{12} , C_{13}) are numerically equal to the charges induced on other conductors in the system when one volt is applied to a single conductor. For instance, in column one of the capacitance matrix shown above, C_{12} is equal to - C_{12}. This is equal to the charge induced on Conductor 2 when one volt is applied to Conductor 1 and zero volts are applied to Conductor 2.

The off-diagonal terms are the negative values of the capacitances between the corresponding conductors (the mutual capacitances). In column one of the example capacitance matrix, the off-diagonal terms represent the capacitances between Conductor 1 and the other two conductors; in column two, the terms represent the capacitance between Conductor 2 and the other conductors; and so forth.

Symmetry for a Capacitance Matrix

The capacitance matrix is symmetric about the diagonal. This indicates that the mutual effects between any two objects are identical. For instance, C_{13}, the capacitance between Conductor 1 and Conductor 3 (C_{13}), is equal to C_{31}, the capacitance between Conductor 3 and Conductor 1.

Solution Process for a Capacitance Matrix

To solve for the capacitance matrix, the electrostatic solver generates a single field solution. The system computes each entry of the capacitance matrix using the following relationship:

$$C = \int E \bullet D d\Omega$$

where the contributions of each conductor to the capacitance matrix are modeled using numerical matrix manipulation techniques.

$$D = \varepsilon E$$

Only conductors with voltage excitations can be included in a capacitance matrix calculation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lumped Capacitance

Lumped capacitance is essentially the result of a 1x1 capacitance matrix. It is used to calculate the electric field energy, U, from the E-field and D-field. Because the energy can also be expressed as

$$U = \frac{1}{2}CV^2$$

the capacitance is

$$C = \frac{2V}{V^2}$$

Maxwell assumes that the problem is set up so that the following is true:

- One conductor (or group of conductors) is set to +1 volt.
- Another conductor (or group of conductors) is set to zero volts.
- All other conductors are floating.

Therefore, the lumped capacitance between all conductors set to one volt and all those set to zero volts is 2*U*.

Warning: The lumped capacitance computation, as described above, is invalid if you assigned voltages other than +1 V or 0 V to the conductors in your model.

Note: For cases where a matrix extraction is desired in Maxwell, the respective extraction needs to be compatible with the hypothesis made regarding the equipotentiality of the respective surfaces, lack of radiation, etc., characteristic to a low frequency circuit approximation. In the circuit representation, the respective lumped elements are interconnected with other circuit elements at the nodes that physically represent the "terminals" in the electromagnetic field problem. Thus, it is possible (within the limits of the lumped circuit elements and applicable theory) to have circuit elements with field effects, such as capacitors and inductors, capable of storing energy. You can access these elements through the provided terminals/nodes, which can then be viewed as energy ports since most of the extracted elements have the capability to store energy. The volume charge density excitation is not a "terminal" type of excitation since the objects to which it is applied cannot, as a rule, be considered equipotential.

Conductance Matrix for a 3D DC Conduction Solution

The Maxwell 3D conductance matrix relates the currents and the voltages as

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\begin{bmatrix} I_1 \\ I_2 \\ \cdots \\ I_n \end{bmatrix} = \begin{bmatrix} G_{11} & G_{12} & \cdots & G_{1n} \\ G_{21} & G_{22} & \ldots & G_{2n} \\ \ldots & \ldots & \ldots & \ldots \\ G_{n1} & G_{n2} & \ldots & G_{nn} \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ \cdots \\ V_n \end{bmatrix}$$

Or in more concise form:

$$I = GV$$

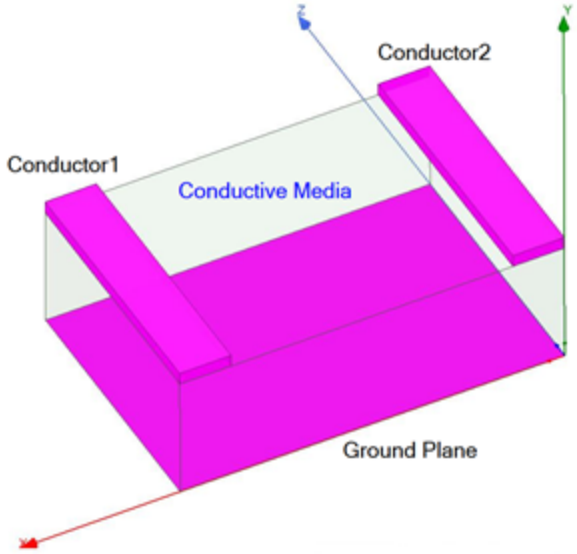
where

- *I* is conduction current,
- *G* is conductance, and
- *V* is voltage

For a simple two-conductor system with a ground plane (V=0), where there are conductive media in the middle: Referring to the figure below, the current flowing through **Conductor1** and **Conductor2**, I1 and I2, are related to their voltages V1 and V2 as

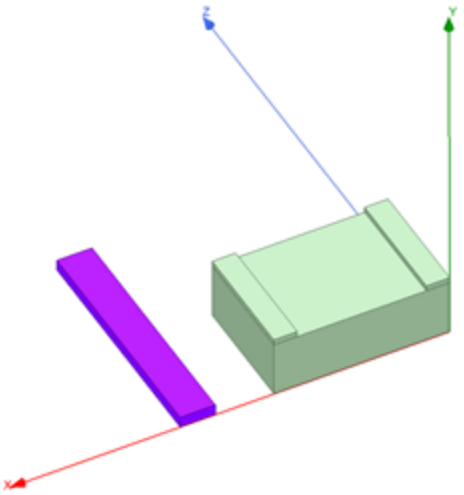
$$I_1 = G_{11} * V_1 + G_{12} * V_2$$

$$I_2 = G_{21} * V_1 + G_{22} * V_2$$



Maxwell 3D can also handle designs that have disconnected conducting regions (multiple, singly connected regions).

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The relationship between the Maxwell conductance matrix G^M and the Spice conductance matrix G^S is

a. $G^M_{i,i} = G^S_{i,1} + G^S_{i,2} + G^S_{i,3} + \ldots + G^S_{i,n}$ ($i = 1,2,\ldots,n$)

b. $G^M_{i,j} = - G^S_{i,j}$ (for i not equal to j)

A ground reference is needed for extracting this matrix. For the Maxwell 3D DC conduction solver, the “sink” excitation, which can be assigned only to surfaces, will be treated as ground. If there is no sink excitation, you must explicitly choose at least one "ground" from the list of voltage excitations in the parameter matrix dialog box.

In the conductance matrix extraction, if there are n “signals” (which can be associated with 3D volume body or 2D sheet body), each one of the n signal ports will be excited with 1 volt while leaving the other signal ports at 0 volt (the ground will be kept at 0 voltage). After calculating the solutions for the excitation settings, losses are computed using the entries of the Maxwell conductance matrix G^M.

Lorentz Force for an Electrostatic Calculation

The component of the Lorentz force due to current in an electric field is

$$\boldsymbol{F} = \int_{Vol} q\boldsymbol{E}\, dVol$$

where

- q is the charge.
- $\boldsymbol{E}$ is the electric field.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Warning: The Lorentz force does not compute the correct force on objects that are assigned materials with a relative permittivity greater than one. To find the force on these objects, use virtual force.

Lorentz Torque for an Electrostatic Calculation

The system uses Lorentz forces to compute the torque around the x, y, and z axes. The Lorentz torque is given by the following equation:

$$\overline{\boldsymbol{T}} = \int_{Vol} q(\overline{r} \times \overline{\boldsymbol{E}})\, dVol$$

where

- ***r*** is the displacement vector from the axis of rotation,
- *q* is the charge, and
- ***E*** is the electric field.

This integral is computed for the x-, y-, and z-axes, giving the net torque on the object(s) about each axis of rotation.

Warning: The Lorentz torque does not compute the correct torque on objects that are assigned materials with a relative permeability greater than one. To find the torque on these objects, use virtual torque.

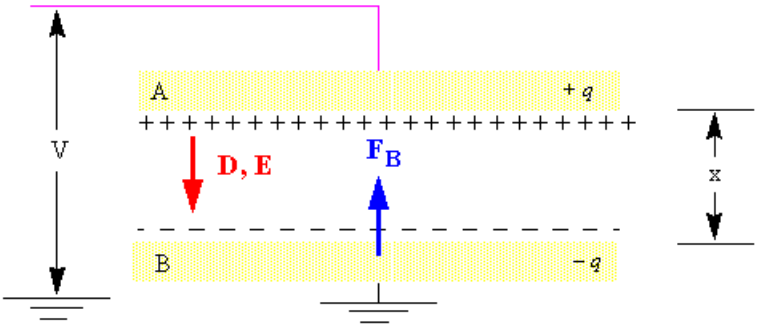
Virtual Force for an Electrostatic Calculation

To compute the virtual force on an object, the electrostatic field simulator uses the principle of virtual work. In the structure shown below, the force on the bottom plate (plate B) in the direction of the displacement, x, is given by the following relationship:

$$F_B = \left.\frac{dU(v,x)}{dx}\right|_{V = ConstantPotential}$$

where U is the electric field energy.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



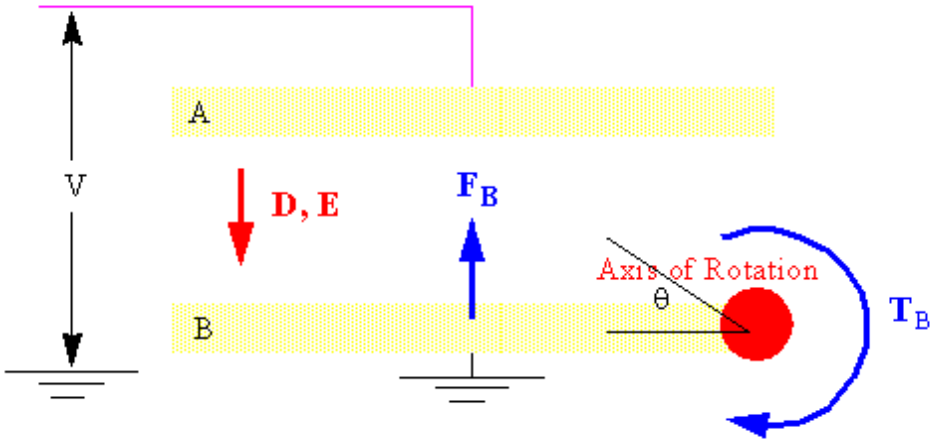
Unlike the classical virtual work method, the plate is not actually moved during the numerical process of the force computation. Instead, only the tetrahedra that lie along the outside surface of the object are virtually distorted. U and its derivative, *dU/dx*, are calculated from a single field solution using finite element interpolation functions.

Virtual Torque for an Electrostatic Calculation

Similar to the virtual force calculation, the system uses virtual work principles to compute the torque on an object. In the structure shown below, the virtual torque on the bottom plate (plate B) about the axis of rotation is given by the following relationship:

$$T_B = \frac{dU(v,\theta)}{d\theta}\Big|_{V = ConstantPotential}$$

where U is the electric field energy.



Unlike the classical virtual work method, the plate is not actually rotated during the numerical process of the torque computation. Instead, only the tetrahedra that lie along the outside surface

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of the object are virtually distorted. U and its derivative, $dU/d\theta$, are calculated from a single field solution using finite element interpolation functions.

Ohmic Loss for DC Conduction

The Ohmic loss, *P*, that is generated in conductors when solving DC current flow problems, is given by the following equation:

$$P = \int_{Vol} \left(\vec{E} \cdot \vec{J} \right) dVol$$

where

- $\vec{E}$ is the electric field.
- $\vec{J}$ is the electric current density.

For the sake of brevity, the ohmic loss density in the post processor should be labeled as **Ohmic Loss**. The total ohmic loss of an object or the whole arrangement can be calculated by integrating over a proper volume in the Field Calculator.

Surface Charge Density

The Surface Charge Density, ρ_s, that is generated on a surface between two materials when solving electric field problems, is given by the following equation:

$$n \cdot (D_2 - D_1) = \rho_s$$

$$n \cdot (\varepsilon_2 E_2 - \varepsilon_1 E_1) = \rho_s$$

where

- E_1 is the electric field and ε_1 the permittivity in the material on one side of the boundary.
- E_2 is the electric field and ε_2 the permittivity in the material on the opposite side of the boundary.
- *n* in the normal vector of the surface.

To calculate the surface charge, you must first select a surface in the modeling window.

Surface Current Density

For impedance boundary, the Surface Current Density, J_{surf}, is computed as the cross product of the surface normal of the impedance boundary and the difference between H fields on the top and bottom of the boundary., and is given by the following equation:

$$J_{surf} = n \times (H_{top} - H_{bottom})$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where

- n is the surface normal vector.
- $(H_{top} - H_{bottom})$ is the difference between H fields on the top and bottom of the impedance boundary.

In order to calculate the surface current density, you must first select a surface in the modeling window.

Magnetostatic Field Calculation

In a **magnetostatic solution**, the magnetic field is produced by DC currents flowing in conductors/coils and by permanent magnets. The electric field is restricted to the objects modeled as real (non-ideal) conductors. The electric field existing inside the conductors as a consequence of the DC current flow is totally decoupled from the magnetic field. Thus, as far as magnetic material properties are concerned, the distribution of the magnetic field is influenced by the spatial distribution of the permeability. There are no time variation effects included in a magnetostatic solution, and objects are considered to be stationary. The energy transformation occurring in connection with a magnetostatic solution is only due to the ohmic losses associated with the currents flowing in real conductors.

The magnetostatic field solution verifies the following two Maxwell's equations:

$$\nabla \times \vec{H} = \vec{J}$$

$$\nabla \cdot \vec{B} = 0$$

with the following constitutive (material) relationship being also applicable:

$$\vec{B} = \mu_0 \left(\vec{H} + \vec{M}\right) = \mu_0 \cdot \mu_r \cdot \vec{H} + \mu_0 \cdot \vec{M}_p$$

where

- $\vec{H}\left(x, y, z\right)$ is the magnetic field strength.
- $\vec{B}\left(x, y, z\right)$ is the magnetic flux density.
- $\vec{J}\left(x, y, z\right)$ is the conduction current density.
- $\vec{M}_p\left(x, y, z\right)$ is the permanent magnetization.
- $\mu_0 = 4 \cdot \pi \cdot 10^{-17} H/m$ is the permeability of vacuum.
- μ_r is the relative permeability.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For nonlinear materials, the dependency between the H and B fields is nonlinear and can be isotropic or orthotropic (in the case of anisotropic material behavior, μ_r is a tensor). Similarly, for permanent magnets, nonlinearity can occur in practical cases and is allowed. Additionally, if a demagnetized condition is to be taken into account for (nonlinear) permanent magnets operating below the knee, Maxwell provides an advanced setup option allowing a solution based on a previously computed demagnetization operating point. If nonlinearity occurs in soft materials (with negligible hysteresis) simultaneously with orthotropic behavior, Maxwell requires that the BH curves for the principal directions in the respective material(s) be provided. From these curves, the energy dependency on H is extracted for each of the respective principal directions and is used in the process of obtaining the nonlinear permeability tensor used in the Newton-Raphson iterative solution process:

$$\vec{B} = \vec{B}_0 + [\tilde{\mu}] \cdot \left(\vec{H} - \vec{H}_0\right)$$

$$[\tilde{\mu}] = \frac{\partial}{\partial \vec{H}} \vec{B} = [\Delta\tilde{\mu}] + [\mu]$$

where $\vec{B}_0$ and $\vec{H}_0$ are the previous field solution, $[\Delta\tilde{\mu}]$ is a general full tensor, and $[\mu]$ is given by the following:

$$[\mu] = \begin{bmatrix} \mu_x & & \\ & \mu_y & \\ & & \mu_z \end{bmatrix}$$

with μ_x, μ_y, μ_z taking into account the anisotropic effects of any laminations present in the model.

The 3D magnetostatic solver considers the magnetic field H with the following components:

$$\vec{H} = \vec{H}_p + \nabla\varphi + \vec{H}_c$$

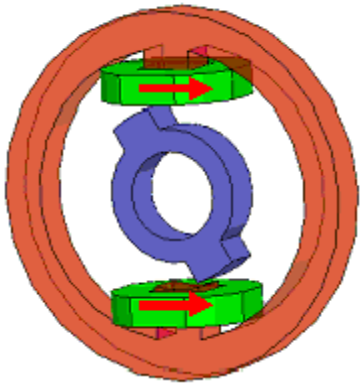
where φ is the magnetic scalar potential, $\vec{H}_p$ is a particular solution constructed by assigning values to all the edges in the mesh in such a way that Ampere's law holds on all contours of all tetrahedra faces in the mesh, and $\vec{H}_c$ accounts for the permanent magnets if any. Thus, the DOFs are the nodal values of the magnetic scalar potential with ten values per tetrahedron at each of the four vertices and all six mid-edge nodes, ensuring a quadratic approximation inside each finite element; *i.e.* the order of the magnetic scalar potential is 2.

There are major advantages of this formulation over other existing ones, including using considerably fewer computational resources (due to the scalar nature of the DOFs), not requiring a gauge due to excellent numerical stability, significantly reducing cancellation errors, and capably of automatically multiplying connected iron regions. The magnetostatic solver handles both 3D linear and nonlinear problems. In the case of nonlinear applications, a classic Newton-Raphson iterative algorithm with user-controlled accuracy is used.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

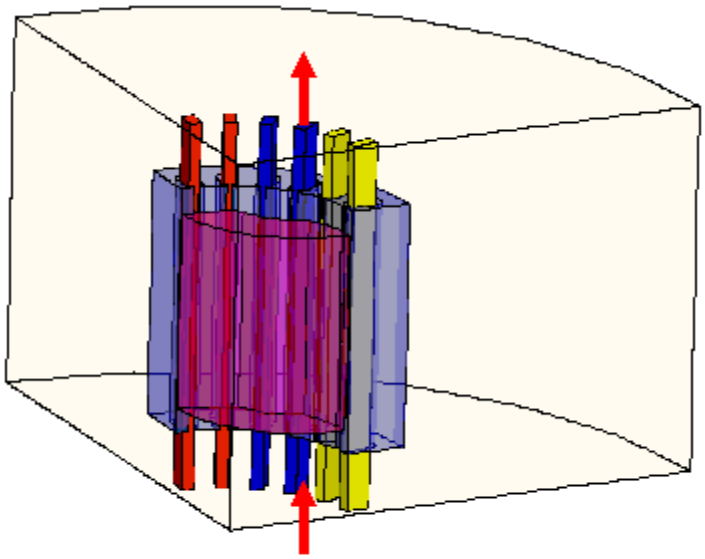
The magnetostatic solver calculates the magnetic field distribution produced by a combination of known DC current density vector distribution and a spatial distribution of objects with permanents magnetization. It is also possible to apply boundary conditions to a model such that the simulation of the immersion of a device into an external magnetic field is also possible. In this latter case, the boundary conditions should be applied in such a way that Maxwell's equations are not violated inside the domain of the solution or at the boundaries.

Typical sources for magnetostatic field problems include voltage, current, permanent magnetization, and current density. When applying the sources for the magnetic field problems, the applied current distribution must be divergence free in the entire space of the solution as it is physical for quasi-stationary conduction current density distributions. Thus, the conduction paths(s) for the applied current distributions must be closed when totally contained within the solution space for the problem, or must begin and end at the boundaries.



Conduction paths in the two coils are closed (one current excitation applied on terminals in each coil)

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Current excitations are applied at the boundaries at both ends of conductors (no "terminals" need to be defined)

The total current applied to conductors that touch the boundaries does not require the existence of terminals at the ends where the current is applied because the respective planar surfaces of the conductors in the plane of the region (background) can be used to apply the excitations. In the case of a closed conduction path, a terminal (sheet object) must be created, matching the cross-section of the conductor at the desired location such that the excitation (current/voltage) can be applied.

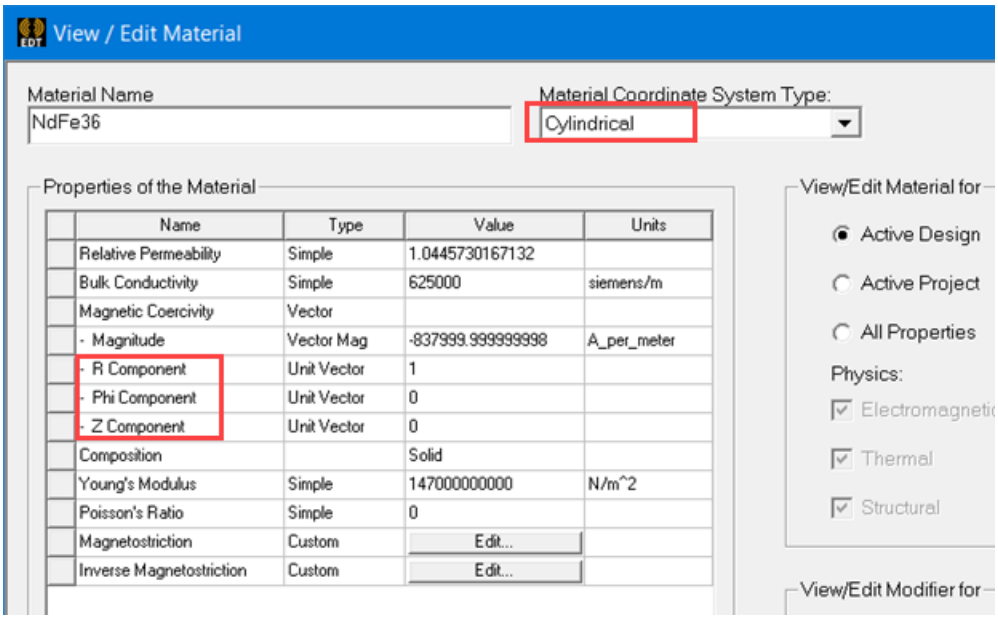
> **Note:** Current and voltage excitations cannot be mixed on any conduction path.

Voltage excitations (at least two) must be specified such that the current can flow in the respective conductors to form an uninterrupted conduction path. **Voltage drop** excitation should by used in the case of closed conduction paths (possibly containing objects with different electric conductivities), primarily in situations where the total current is unknown. One voltage drop excitation per conduction path should be applied. The total current for any conduction path should add up to zero.

The magnetostatic solver does not compute the electric field distribution associated with a voltage distribution outside conductors. However, the electric field inside conductors is indirectly available for postprocessing via the conduction current density distribution, which is available, so it is possible to calculate the ohmic loss in conductors with applied current or voltage excitations.

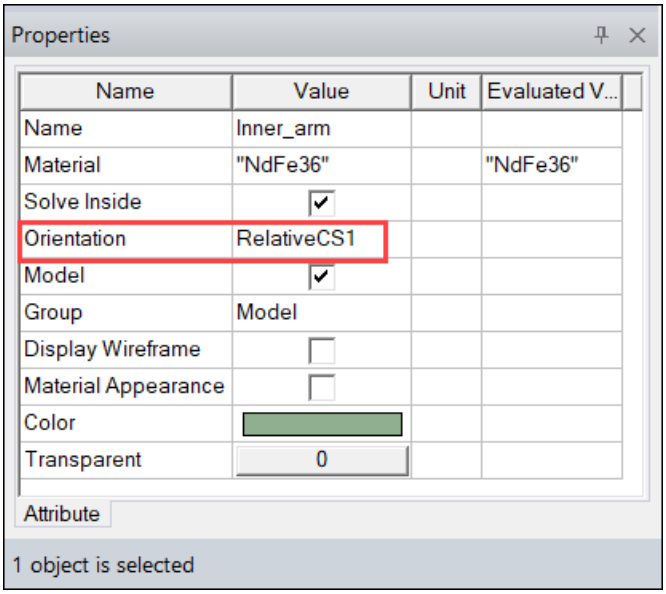
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Permanent magnetization in Maxwell is treated as a material property, but we mention it here because of its source characteristics:



Permanent magnetization can be specified in Cartesian, cylindrical, or spherical material coordinate systems, as shown in the above window where the R, Phi, and Z components (in a cylindrical system, considered here as example) can be specified in any meaningful combination. The actual direction of the vector property is decided when the property is assigned to the object and when the desired coordinate system is chosen (the global coordinate system always exists, while additional ones can be defined). If necessary, additional relative coordinate systems can be defined and used to specify the orientation of the vector property.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



While the type of material coordinate system is embedded into the material definition, the actual orientation is not, such that a material property defined, for instance, in a cylindrical system can be applied to multiple objects with different orientations if multiple relative local coordinate systems were defined and used in the process of assigning the respective material property.

A **typical boundary conditions** used in magnetostatic problems is the **magnetic field tangent** (the default, natural boundary condition that is automatically applied on all surfaces of the problem space -the surfaces of the geometry entity containing the model inside it). This default boundary condition can be overwritten if other boundary conditions are applied on exterior surfaces of the solution space. The default boundary condition confines the magnetic field to the solution space; therefore, this boundary must be placed at some distance from the sources to avoid over-constraining the fields by placing the boundaries to close to the model objects. While it is difficult to provide "recipes" with general validity on the placement of the boundaries of problems, a good rule of thumb asserts that if a model can be imagined as being contained in a sphere of radius R, then the boundaries can be placed at a 4-5 radii R from the imaginary center of the model.

The **Zero Tangential H Field** allows the user to prescribe a normal (in average) field orientation on an arbitrary surface. This boundary condition does not require any further user input, meaning that no value and coordinate system specification are necessary.

In the case of **Tangential H Field** boundary condition, the values of two tangential components and a surface coordinate system must be defined and subsequently used in the assignment process of this boundary condition. This boundary condition is restricted to two kinds of surfaces: planar or cylindrical. The CS in the planar case would be a rectangular coordinate system with X and Y components of H in the plane. The coordinate system in the cylindrical case would be a

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

coordinate system with a Z axis coinciding with the axis of the cylinder, and the two components to input would be the PHI and Z axis components of H.

In the case of the tangential H field, caution is advised: This boundary condition should be specified such that Ampere's theorem is not violated inside the field domain or at the boundaries.

Symmetry boundary conditions are used to solve problems with symmetry and, thus, allow the users to take advantage of the significantly reduced problem size for a given accuracy.

Symmetry boundary conditions can be one of two kinds:

- Odd (flux tangent)
- Even (Flux normal)

Although the use of this symmetry boundary condition overlaps with previous conditions to some extent, the symmetry boundary conditions should only be used in clear symmetry cases.

Symmetry for a Magnetostatic Field Solution

A Symmetry boundary models a plane of symmetry in a structure. Use this type of boundary condition to take advantage of geometric and magnetic symmetry in a structure. Doing so enables you to reduce the size of your model, which helps to conserve computing resources. Two types of symmetry are available:

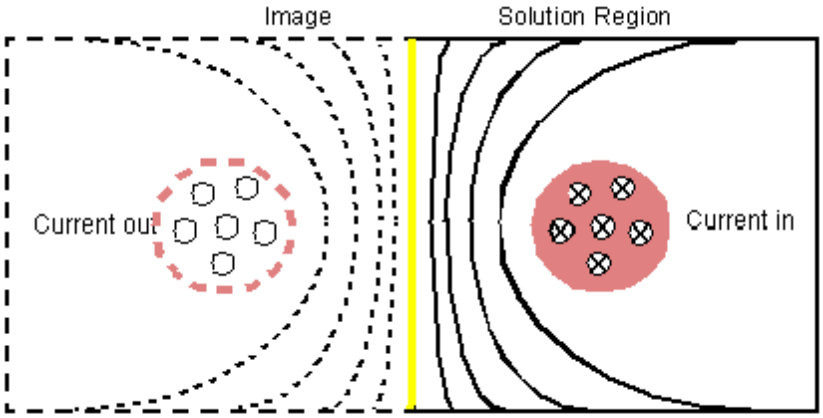
- Odd Symmetry (Flux Tangential)
- Even Symmetry (Flux Normal)

These boundaries can only be assigned to the outside edges of the solution region.

Odd Symmetry (Flux Tangential) for a Magnetostatic Field Solution

Use an odd symmetry boundary to model a plane of symmetry in which current on one side of a plane flows in the opposite direction to current on the other side of the plane. Magnetic flux is tangential to this type of boundary. To define an odd symmetry boundary, the simulator sets the selected edge to a Neumann boundary.

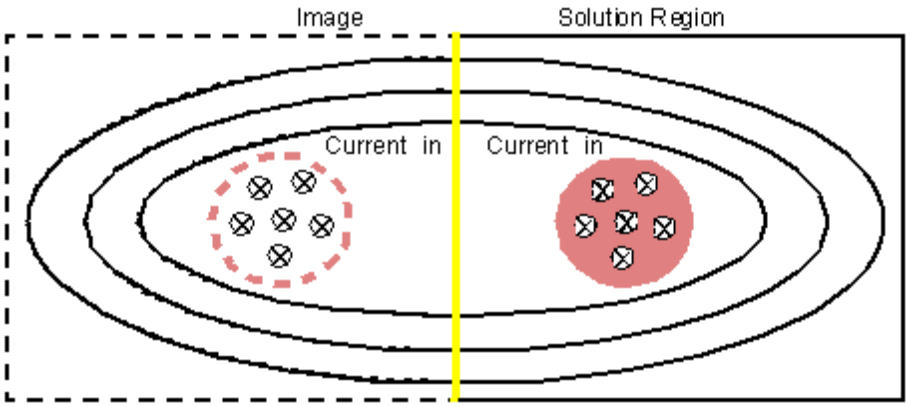
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The B-field is tangential to an odd symmetry boundary.

Even Symmetry (Flux Normal) for a Magnetostatic Field Solution

Use an even symmetry boundary to define a plane of symmetry where the direction of current flow is the same on both sides of the plane. Magnetic flux is normal to this type of boundary. To define an even symmetry boundary, the simulator sets the selected edge to a Magnetic Field boundary with a field value of zero – acting as a magnetic mirror to the model.



The B-field is perpendicular to an even symmetry boundary.

> **Warning:** When using even symmetry boundaries, be careful that you do not violate Ampere's law.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Matching Boundary for a Magnetostatic Field Solution

Matching boundaries allow you to model planes of periodicity where the H-field on one surface exactly matches the H-field on another. They force the magnetic field at each point on one surface (the "independent" boundary) to match the magnetic field at each corresponding point on the other surface (the "independent" boundary). They are very useful for modeling devices such as motors, in which the magnetic field repeats every 180°, 120°, 90°, or less. Basically, they enable you to model the smallest possible periodic segment of the device – reducing the amount of computing resources needed during the solution.

To set up matching boundaries, you must create both an independent boundary and a dependent boundary. Unlike Symmetry boundaries, **H** does not have to be tangential or normal to these boundaries. The only condition is that the fields on the two boundaries must have the same magnitude and direction (or the same magnitude and opposite directions).

Independent Boundary for a Magnetostatic Field Solution

The simulator computes the magnetic field on an independent boundary using currents, permanent magnets, and magnetic fields as input. The field is then mapped to the dependent boundary.

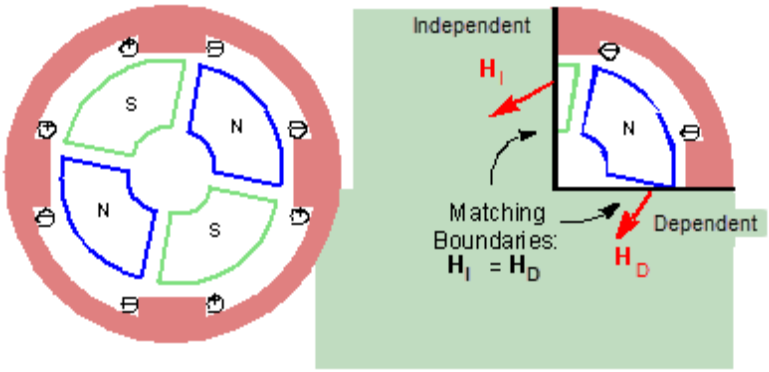
Dependent Boundary for a Magnetostatic Field Solution

The magnetic field on the dependent boundary is forced to match the field on the independent boundary. The magnitude of the magnetic field on both boundaries is the same. However, the fields on the two boundaries can either point in the same direction, or in opposite directions, as specified by the user during setup.

When to Use Matching Boundaries for a Magnetostatic Field Solution

Matching boundaries enable you to take advantage of periodicity in a structure. For example, below is a diagram of the cross-section of a simple brushless DC motor. The field in such a motor repeats itself every 90 degrees; that is, the field pattern in one quarter of the motor matches the magnitude and direction (or the opposite of the direction) of the field pattern in the other three quarters. With matching boundaries, all you have to model is one quarter of the structure.

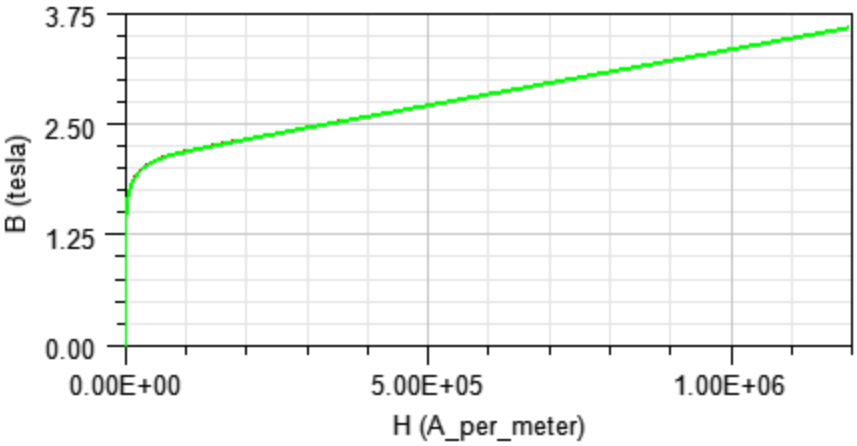
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A Symmetry boundary cannot be used to simulate periodicity because the magnetic field is not necessarily either perpendicular or tangential to periodic surfaces. For example, in the quarter model shown on the right, the magnetic field is exactly perpendicular to the bounding surfaces only when the gap separating the permanent magnets is perfectly horizontal or vertical. For all other positions of the rotor, matching boundaries are required.

Nonlinear Materials

If a material has a permeability that varies with the flux density, a **B** vs. **H** curve (B-H curve) such as the one below is needed to describe the material's nonlinear behavior.



In nonlinear materials, the B-field (magnetic flux density) is a function of itself:

$$\boldsymbol{B} = \mu r(B)\mu_O H$$

where μ_r(B), the relative permeability, depends on the magnitude of the B-field at each point in the material. Therefore, to model the magnetic behavior of the material, a curve relating the B-field directly to the H-field is used to describe the nonlinear relationship.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See also Advanced Nonlinear Options for Magnetostatic Solutions or Magnetization for Transient Solutions for more information about permeability and magnetization settings.

Solution Process for a Magnetostatic Field Solution

In general, the solution process involves a phase in which the distribution of current density (if not already known) is obtained, followed by the calculation of the distribution of magnetic field that corresponds to the distribution of the sources. In the case of solid conductors with an applied current source, an automatic process is initiated such that following an iterative process involving adaptive mesh refinement, the current density vector distribution is obtained and subsequently the corresponding magnetic field distribution is computed. In the case of stranded conductors, it is assumed that the current density is uniform on the cross-section of the respective conductors. Thus, the stranded option for current excitations should mostly be used only in cases where the cross-section of conductors is constant, consistently with the assumption that the respective object is a coil built with strands of wire. For this reason, conductors (coils) could be created by either sweeping around an axis using zero segments, or sweeping along a path having only true surface segments in order to create a smooth surface coil having a completely uniform cross-sectional area over its entire length.

Magnetic Field Energy for a Magnetostatic Field Solution

The magnetic energy of a system is given by the following expressions. The expressions represent total values of energy for the volumes taken into account. Note that the integrals have simpler expressions if the material property of the object considered is a linear one. In case of linear material properties magnetic energy and co-energy values are identical ($W = W_C$).

- In linear materials, the energy is

$$W = \frac{1}{2} \int_{Vol} B \bullet H \, dVol$$

- In nonlinear materials, the energy is

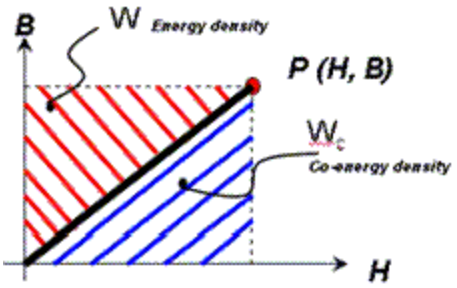
$$W = \int_{Vol} w \bullet dVol = \int_{Vol} \left(\int_0^B H \bullet dB \right) dVol$$

  where

  - **H** is the magnetic field.
  - **B** is the magnetic flux density.

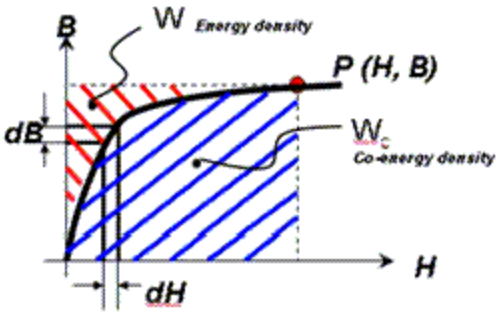
The following figure represents graphically the definitions of both energy and co-energy densities. Note that both are local quantities (*i.e.* are function of X, Y, Z coordinates of the location in the model) and also depend on the operating point on the applicable B-H curve. Energy is equal to the area above the B-H curve, and coenergy is equal to the shaded blue area below the B-H curve.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In **Linear** Materials

- μ is constant
- Energy is equal to Co-Energy



In **nonlinear** Materials

- μ varies according to B-H Curve
- Energy is less than Co-Energy

For the sake of brevity, the magnetic flux density in the post processor should be labeled as **Magnetic Flux**. The total magnetic flux of an object or the whole arrangement can be calculated by integrating over a proper volume in the Field Calculator.

Magnetic Coenergy for a Magnetostatic Field Solution

The magnetic coenergy of a system is given by the following expressions. The expressions represent total values of co-energy for the volumes taken into account. Note that the integrals have simpler expressions if the material property of the object considered is a linear one. In case of linear material properties magnetic energy and co-energy values are identical ($W = W_c$).

- In linear materials, the co-energy is

$$W_c = \frac{1}{2} \int_{Vol} B \bullet H \, dVol$$

- In nonlinear materials, the coenergy is

$$W_c = \int_{Vol} w_c \bullet dVol = \int_{Vol} \left(\int_0^H B \bullet dH \right) dVol$$

$$W_c = \frac{1}{2} \int_{Vol} B \bullet H \, dVol$$

- In nonlinear materials, the coenergy is

$$W_c = \int_{Vol} w_c \bullet dVol = \int_{Vol} \left(\int_0^H B \bullet dH \right) dVol$$

where:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **H** is the magnetic field.
- **B** is the magnetic flux density.

The coenergy is related to the magnetic field energy.

> **Note:** If permanent magnets are part of the model, as a rule, the magnetic energy reported by the solver and the postprocessor may be different. This occurs because the adaptive meshing part of the solution sequence runs a special computation so that it can avoid situations with a total zero energy of the field (problems with permanent magnets and no other excitation). To avoid such situations, the solver uses (and reports) the total energy, which includes the absolute value of the energy inside the permanent magnets. Because of this, the energy reported by the solver in these cases is greater than the energy reported by the postprocessor.

Magnetic Apparent Energy for a Magnetostatic Field Solution

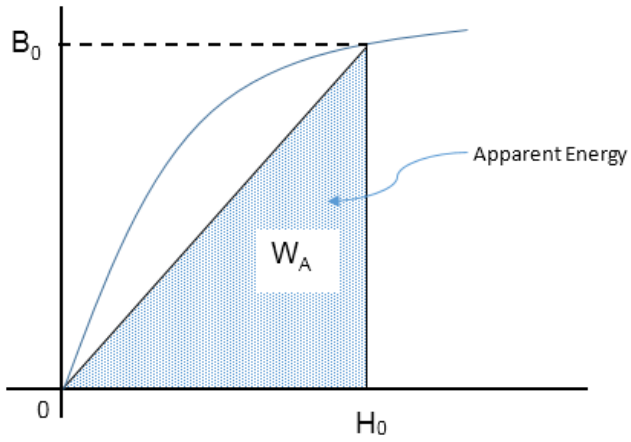
The magnetic apparent energy of a system is given by the following expression:

$$W_A = \frac{1}{2} H_0 B_0$$

where

- $\mathbf{H_0}$ is the magnetic field final operating point.
- $\mathbf{B_0}$ is the magnetic flux density final operating point .

Apparent energy is equal to the shaded area below.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Inductance Matrix for a Magnetostatic Field Solution

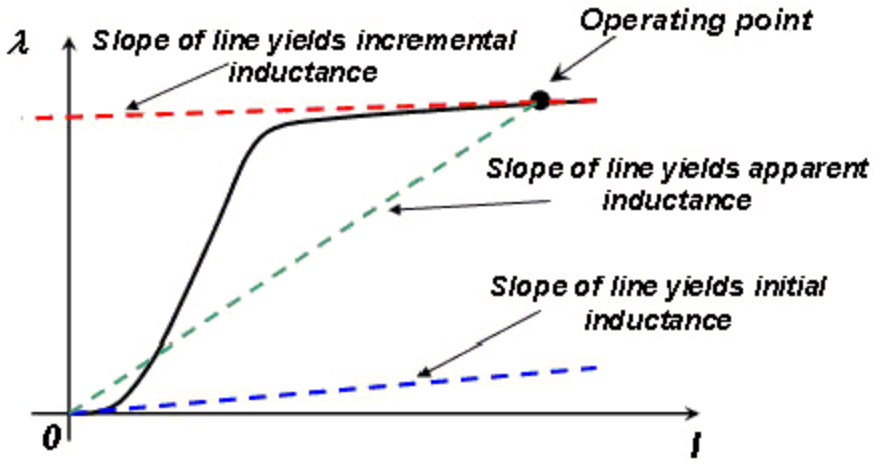
Inductance matrices can be expressed in terms of flux linkage and current or in terms of voltage and time-varying current. Inductance values are given in henries.

Types of Inductance

For nonlinear materials, the definition of inductance requires additional detail compared to the linear case. Three commonly used inductance values are illustrated in the figure above - each inductance has useful applications. Maxwell uses apparent inductance to calculate flux linkage as a function of the independent variables, since it changes with current as the material properties change. To obtain apparent inductances for nonlinear materials in Maxwell, a two-step procedure is followed:

1. A nonlinear magnetostatic solution is generated with all sources at user specified values. This establishes a value of permeability that varies with each mesh element, since the degree of saturation varies throughout the device.
2. These self-consistent values are used in a linear solution for the inductance matrix, with each coil current set to one ampere. The resulting values are apparent inductances, which vary with each specified coil current operating point because the material properties change.

The inductance calculation is now linearized on the self-consistent values. If the coil currents had been increased to the specified values in the first step, in the linearized model, the resulting magnetic field would be identical to the nonlinear solution. Therefore, the inductances from the second step are the ratios of respective flux linkages to coil current, at the nonlinear operating point originally determined.



Nonlinear Inductance: Current vs. Flux Linkage

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The slope of any line drawn on the figure above has units of inductance (henries) and any area enclosed on this plot represents units of energy in joules if magnetic flux is expressed in Webers and current is expressed in Amperes. The three inductances are defined as:

Incremental (differential) inductance $L_{inc} = \frac{d\lambda}{dI}$	The slope of a line tangent to the curve at the operating point. Incremental inductance is the usual textbook definition. For small signal AC analysis in Maxwell SPICE or Saber, incremental inductance is evaluated using an operating point determined from a DC solution. Incremental inductance can also be used in time domain system simulations by using the compensation theorem, but it has no useful relationship to stored energy.
Apparent inductance $L_{app} = \frac{\lambda}{I}$	The slope of a line from the origin to the operating point. Apparent inductance gives the total flux linkage as a function of current, so it is well-suited for the state equation or basis function approach to time domain simulations. For small-signal AC analysis, apparent inductance cannot be used directly, but can be used to supply the proper derivatives to Maxwell SPICE. Energy can be calculated with apparent inductance, but the result is not really the stored energy because it ignores the path taken along the nonlinear curve. Apparent inductance does not provide the actual stored energy at the specified operating point. However, during time domain simulations, apparent inductance varies with current, and tracks the nonlinear curve shown in the figure.
Initial inductance L_{init}	The slope of a line tangent to the curve at the origin. Initial inductance applies to many magnetic materials that have a "toe" in the magnetization curve. Laboratory measurements at 0 A DC measure initial inductance, not the "linear" value that may be somewhat higher. Note: Initial inductance is a particular case of the incremental inductance where the slope of the respective line passes through the origin of the (flux - current) plane.

For linear materials, all three inductance values are equal. For nonlinear materials, the three values are generally different. Except for initial inductance, each value varies with the operating point. Ignoring the "toe" of the curve, the relation $L_{app} > L_{inc}$ holds. Energy taken from or supplied to the external circuit, as determined from the terminal voltage and current, will be correct.

The apparent inductance calculated by Maxwell at the actual operating point due to all sources in the model (currents in the coils but also permanent magnets) is the base of all inductance calculations. Incremental (differential) inductance information can be easily derived from the flux vs current characteristic.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

It should be noted that the apparent value relates to the local frozen, current dependent absolute permeability as described above in the two step process. Incremental inductance relates to another physical quantity which is the differential (incremental) permeability defined as dB/dH.

Solver calculated magnetostatic energy (reported as part of the solution convergence data) is the apparent energy, always equal to the global, model-wise average between energy and co-energy. For the linear material case in the entire model there is only one value of the inductance $L = L_{init} = L_{incr} = L_{app}$.

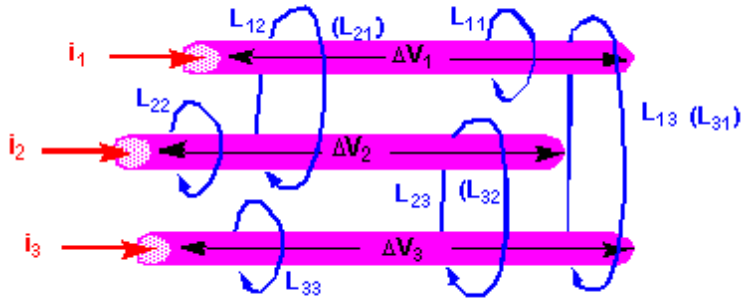
Inductance in Terms of Flux Linkage and Current

An inductance matrix represents the magnetic flux linkage between current loops. Given the three current loops below, the relationship between induced flux and currents is

$$\lambda_1 = L_{11}i_1 + L_{12}i_2 + L_{13}i_3$$

$$\lambda_2 = L_{21}i_1 + L_{22}i_2 + L_{23}i_3$$

$$\lambda_3 = L_{31}i_1 + L_{32}i_2 + L_{33}i_3$$



This can be expressed in matrix form as

$$\begin{bmatrix} \lambda_1 \\ \lambda_2 \\ \lambda_3 \end{bmatrix} = \begin{bmatrix} L_{11} & L_{12} & L_{13} \\ L_{21} & L_{22} & L_{23} \\ L_{31} & L_{32} & L_{33} \end{bmatrix} = \begin{bmatrix} i_1 \\ i_2 \\ i_3 \end{bmatrix}$$

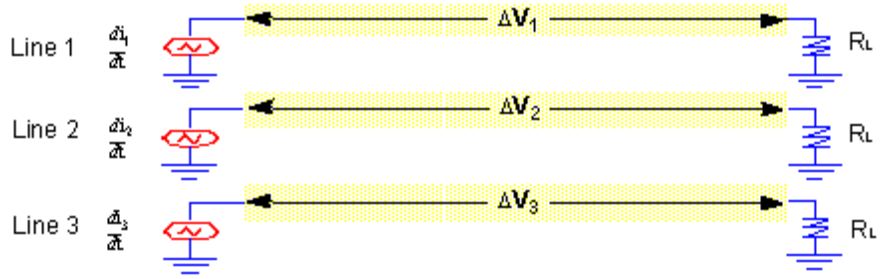
The inductance matrix above gives the relationship between λ and *i* for the three independent current loops. A device with *n* current loops would have an *n* inductance matrix.

Inductance in Terms of Voltage and Time-Varying Current

An inductance matrix can also represent the relationship between voltage and current fluctuations in a system. Given the three independent transmission lines shown below, the voltage changes caused by the time varying current source on each line are given by

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\begin{bmatrix} \Delta V_1 \\ \Delta V_2 \\ \Delta V_3 \end{bmatrix} = \begin{bmatrix} L_{11} & L_{12} & L_{13} \\ L_{21} & L_{22} & L_{23} \\ L_{31} & L_{32} & L_{33} \end{bmatrix} = \begin{bmatrix} \frac{di_1}{dt} \\ \frac{di_2}{dt} \\ \frac{di_3}{dt} \end{bmatrix}$$



The inductance matrix above gives the relationship between ΔV and di/dt for the three transmission lines. If di_2/dt and di_3/dt are set to zero, this relationship becomes

$$\begin{bmatrix} \Delta V_1 \\ \Delta V_2 \\ \Delta V_3 \end{bmatrix} = [L] \begin{bmatrix} \frac{di_1}{dt} \\ 0 \\ 0 \end{bmatrix} \begin{bmatrix} L_{11} \\ L_{12} \\ L_{13} \end{bmatrix} \frac{di_1}{dt}$$

This gives the voltage changes that are induced on Lines 2 and 3 when a time-varying current source is applied to Line 1 – that is, the inductive coupling between the three loops.

The inductance matrix can be exported to a circuit/system simulator, such as Twin Builder, where it can be used in a transient simulation.

Matrix Elements for an Inductance Matrix

If one amp is applied to Current Loop 1 and zero amps is applied to the other two loops, the inductance matrix becomes the following:

$$\begin{bmatrix} \lambda_1 \\ \lambda_2 \\ \lambda_3 \end{bmatrix} = [L] \begin{bmatrix} 1 \\ 0 \\ 0 \end{bmatrix} = \begin{bmatrix} L_{11} \\ L_{12} \\ L_{13} \end{bmatrix}$$

Diagonal Elements for an Inductance Matrix

Terms such as L_{11} represent the self-inductance of each current loop. Self-inductance is numerically equal to the flux linkage in a current loop when one amp is flowing in it, and no current is flowing in the other loops. For example, L_{11} is equal to the flux in Current Loop 1 when one amp is flowing in that current loop, and no current is flowing in the other loops.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Off-Diagonal Elements for an Inductance Matrix

Terms such as L_{12} and L_{13} represent the mutual inductances between the current loops. Mutual inductance is numerically equal to the flux linkage in a current loop when one amp is flowing through another loop, and no current is flowing anywhere else. For example, L_{12} is equal to the flux linkage in Loop 1 when one amp is applied to Loop 2 and no current is flowing in the other loops.

Symmetry for an Inductance Matrix

The inductance matrix is symmetric about the diagonal. This indicates that the mutual effects between any two loops are identical. For instance, L_{13}, the inductance between Current Loop 1 and Current Loop 3, is equal to the inductance between Current Loop 3 and Current Loop 1.

Solution Process for an Inductance Matrix

To solve for the inductance matrix, the magnetostatic solver generates a single field solution. The system computes each entry of the inductance matrix using the following relationships:

$$U = \frac{1}{2}LI^2 = \frac{1}{2}\int H \bullet B d\Omega$$

But, assuming that the current, *I*, is one ampere:

$$L = \int H \bullet B d\Omega$$

where $B = \mu H$.

Each conductor in the matrix is assumed to be part of an independent current loop. The contributions of each loop to the inductance matrix are modeled using numerical matrix manipulation techniques.

Warning: Inductance is valid for different current levels in linear and nonlinear models:

- In models containing only linear materials, the system assumes that one amp of current flows through each loop. The inductance matrix that is computed for these models is valid for all current levels.
- In models containing nonlinear materials, the system uses the currents you specified when setting up boundaries and sources. The inductance matrix that is computed for these models is valid only for these current levels.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Grouping Inductance Matrix Elements

The results of the inductance matrix calculation may be post processed into series/parallel windings using the grouping function. The operations of the grouping function can be one of two cases:

- Grouping all coils in series connection by grouping into 1 branch
- Grouping in a mixed series/parallel combination by setting the number of branches to a value of 2 or more.

Series Grouping

The inductances of the coils can be calculated as follows:

$$Lcoil_i = L_i \cdot n_i^2$$

Where $\mathbf{L_i}$ is the per-turn inductance of the source calculated by the field solver and $\mathbf{n_i}$ is the number of turns specified in the grouping dialog box. Combining inductors in series, which is the case when the number of branches is set to 1, is done according to:

$$L_{seriesgroup} = \sum_{i=1}^{m} Lcoil_i + \sum_{\substack{i,j=1 \\ i \neq j}}^{m} M_{ij} \cdot n_i \cdot n_j + \sum_{\substack{i,j=1 \\ i \neq j}}^{m} M_{ji} \cdot n_i \cdot n_j$$

where

- m is the number of source entries being grouped.
- $Lcoil_i$ represents the self inductance term for each coil entry to be grouped according to the equation above.
- M_{ij} and M_{ji} are the mutual inductance terms between the sources being grouped. M_{ij} and M_{ji} must generally be considered separately as the matrix may be antisymmetric.
- n_i are the number of turns for each coil in the group.

Parallel Branches

If the number branches in the group is set to a value other than 1, the coils are first grouped in series as described above, and then divided into by the number if branches specified.

The grouped inductance is then calculated as follows:

$$L_{branch} = \frac{L_{seriesgroup}}{k^2}$$

where k is the number of branches set for the group.

The calculation above only relies on the inductance matrix calculated during the field solution and can therefore be manipulated as a post processing calculation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lorentz Force

The component of the Lorentz force due to current in a magnetic field is

$$\boldsymbol{F} = \int_{Vol} \boldsymbol{J} \times \boldsymbol{B}\, dVol$$

where

- $\boldsymbol{J}$ is the current density.
- $\boldsymbol{B}$ is the magnetic flux density.

Note: The Lorentz force does not compute the correct force on objects that are assigned materials with a relative permeability greater than one (that is, materials where $\mu \neq \mu_0$). To find the force on these objects, use virtual force.

Note: For PCB applications, the parameter force calculated by using virtual force is not accurate if the mesh in air gaps between two layers is not fine enough. It is strongly recommended to compute parameter forces on conducting objects by using the Lorentz force option.

Lorentz Torque

The system uses Lorentz forces to compute the torque around the x-, y-, and z-axes. The Lorentz torque is given by the following equation:

$$\boldsymbol{T} = \int_{Vol} \boldsymbol{r} \times (\boldsymbol{J} \times \boldsymbol{B}) dVol$$

where:

- $\boldsymbol{r}$ is the displacement vector from the axis of rotation.
- $\boldsymbol{J}$ is the current density.
- $\boldsymbol{B}$ is the magnetic flux density.

This integral is computed for the x-, y-, and z-axes, giving the net torque on the object(s) about each axis of rotation.

Note: The Lorentz torque does not compute the correct torque on objects that are assigned materials with a relative permeability greater than one (that is, materials where $\mu \neq \mu_0$). To find the torque on these objects, use virtual torque.

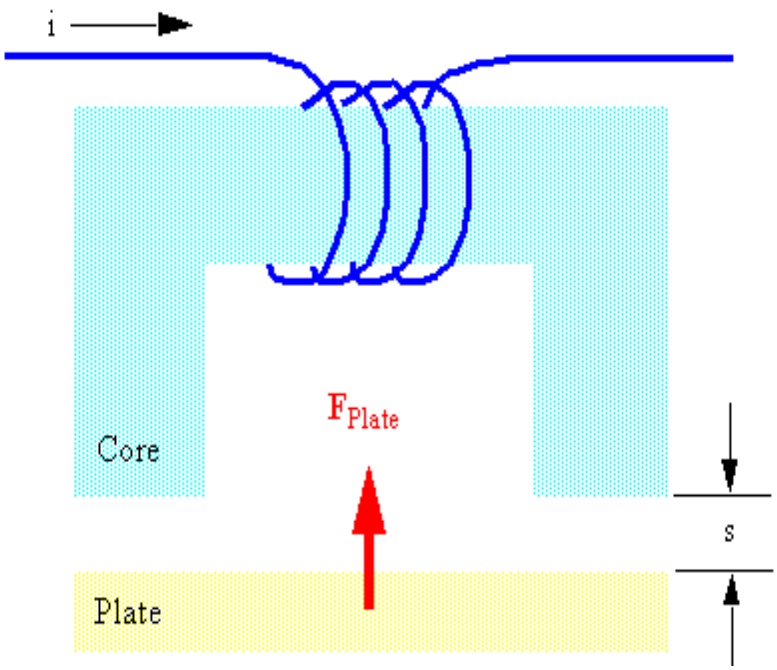
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Virtual Force

To compute the virtual force on an object, the system uses the principle of virtual work. In the structure shown below, the force on the plate in the direction of the displacement, s, is given by the following relationship:

$$F_{plate} = \frac{dW(s,i)}{ds}\Big|_{i=Constant}$$

where *W(s,i)* is the magnetic coenergy of the system. The current, *i*, is held constant.



Unlike the classical virtual work method, the plate is not actually moved during the force computation. Instead, only the tetrahedra that lie along the outside surface of the object are virtually distorted. Thus, the force computation only requires one field solution.

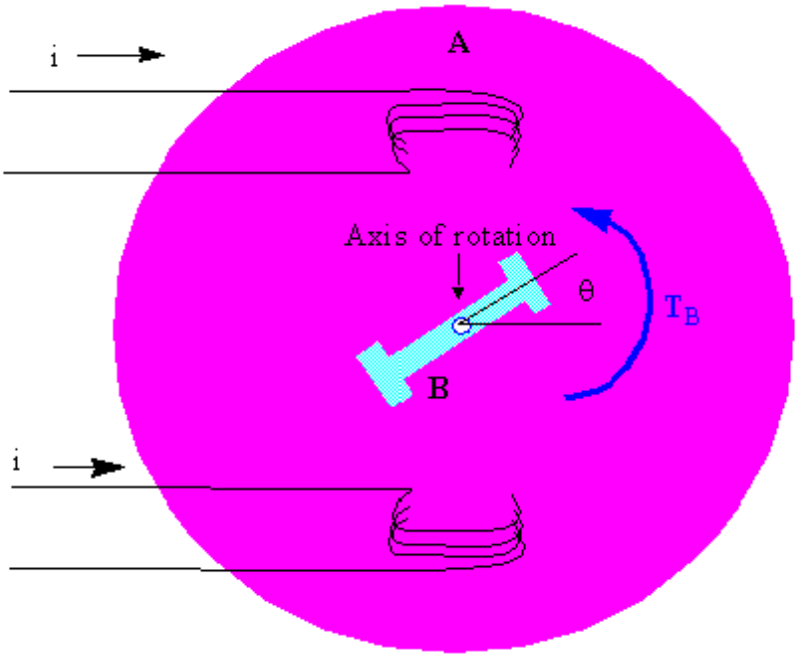
Virtual Torque

Similar to the virtual force calculation, the system uses virtual work principles to compute the torque on an object. In the structure shown below, the torque on Object B about the axis of rotation is given by the following relationship:

$$T_B = \frac{dW(\theta,i)}{d\theta}\Big|_{i=Constant}$$

where *W(θ, i)* is the magnetic coenergy of the system. The current, *i*, is held constant.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Unlike the classical virtual work method, Object B is not actually rotated during the force computation. Instead, only the tetrahedra that lie along the outside surface of the object are virtually distorted. Thus, the change in the system's coenergy (and therefore the virtual torque) is given by the change in the coenergy of these tetrahedra.

AC Magnetic (Eddy Current) T-Omega Solver

The **3D AC Magnetic solver (frequency domain or harmonic solver)** uses the $\vec{T} - \Omega$ (Omega) formulation. It is based on the assumption that all electromagnetic fields pulsate with the same frequency (specified by the user) and have magnitudes and initial phase angles calculated by the Maxwell solver. There are no moving objects (velocity is zero everywhere). Permanent magnets cannot be part of the model, and all materials are assumed to be linear theoretically.

For designs with nonlinear materials, analysis is based on the fundamental components of B and H at a specified frequency as an approximation. See Nonlinear Eddy Field Simulation for more information.

For nonlinear systems, the AC Magnetic solver can be used as an alternative to the transient solver to solve for harmonic losses.

Electromagnetic radiation can also be simulated.

In the non-conducting regions the magnetic field strength is given by the following equation:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\vec{H} = \vec{H}_p + \nabla\Omega$$

using node element with the same quadratic approximation as for the case of the magnetostatic solver.

In regions with non-zero conductivity where eddy current calculation has been set up, the following equation is true:

$$\vec{H} = \vec{H}_p + \nabla\Omega + \vec{T}$$

where $\vec{T}$ is the electric vector potential calculated using edge elements. At the interface between conductors and non-conductors the tangential component of the electric vector potential is constrained to zero.

The default order of the magnetic scalar potential is 2, and the default order of the electric vector potential is 1.

The quantity calculated by the Maxwell solver is in this case the magnetic field H(x,y,z).

For AC Magnetic problems, typical sources associated with non-winding excitation are current and current density; typical sources associated with winding excitation are current and voltage. In applying the sources for the magnetic field problems, keep in mind that the applied current distribution must be divergence free in the entire space of the solution as it is physical for (quasi) stationary conduction current density distributions. Thus, the conduction paths(s) for the applied current distributions must be closed when totally contained within the solution space for the problem or must begin and end at the boundaries. The total current applied to conductors that touch the boundaries doesn't require the existence of terminals at the ends where the current is applied, the respective planar surfaces of the conductors in the plane of the region (background) can be used to apply the excitations.

Typical boundary conditions used in AC Magnetic problems include **magnetic field tangent** (the default, natural boundary condition which is automatically applied on all surfaces of the problem space -the surfaces of the geometry entity containing the model inside it). This default boundary condition can be overwritten if other boundary conditions are applied on exterior surfaces of the solution space. The default boundary condition confines the magnetic field to the solution space and thus this boundary must be placed at some distance from the sources of the problem to avoid over-constraining the fields by placing the boundaries to close to the model objects. While it is difficult to provide "recipes" with general validity on the placement of the boundaries of problems, a good rule of thumb says that if a model can be imagined as being contained in a sphere of radius R, the boundaries can be placed at a 4-5 radii R from the imaginary center of the model.

The **Zero Tangential H Field** allows you to prescribe a normal (in average) magnetic field orientation on an arbitrary surface. This boundary condition does not require any further user input, that is, no value and coordinate system specification are necessary.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the case of **Tangential H Field** boundary condition the values of two tangential complex components (real and imaginary) and a surface coordinate system have to be defined and subsequently used in the assignment process of this boundary condition. This boundary condition is restricted to two kinds of surfaces: planar or cylindrical. The CS in the planar case would be a rectangular coordinate system with X and Y components of H in the plane. The coordinate system in the cylindrical case would be a coordinate system with a Z axis coinciding with the axis of the cylinder and the two components to input would be the PHI and Z axis components of H.

In the case of the tangential H field caution is advised: this boundary condition should be specified such that Ampere's theorem is not violated inside the field domain or at the boundaries. This boundary condition is very useful to simulate the behavior of devices "immersed" in an electromagnetic filed of desired magnitude and orientation.

Symmetry boundary conditions are used to solve problems with symmetry and thus allows the users to take advantage of the significant reduction of the problem size for a given accuracy. Symmetry boundary conditions are of two kinds, **Odd (flux tangent)** or **Even (Flux normal)**. Although the use of this symmetry boundary condition overlaps with previous conditions to some extent, the symmetry boundary conditions should only be used in clear symmetry cases.

The **Insulating** boundary condition is of particular use for applications where very thin insulating layers are impractical to model due for example to high aspect ratio geometries that would be generated and associated meshing difficulties. Thus insulating boundary condition can be assigned to surfaces of separation between conductors. Another remarkable situation where such a boundary condition proves to be extremely useful is in modeling faults in conductors (cracks for example). Thus, modeling 2D (sheet) objects at the location of the respective cracks and applying the insulation boundary condition proves to be a very effective way of modeling the flaws without having to generate 3D cracks that would be difficult and impractical to mesh.

For more information on AC Magnetic simulation setup, see AC Magnetic (Eddy Current) Solver Settings.

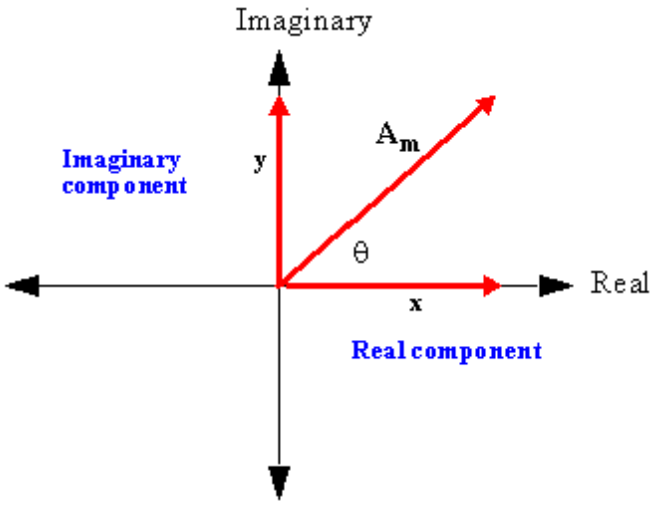
Real and Imaginary Components

In general, for an AC Magnetic (Eddy Current) analysis, you must enter a magnitude and phase for all AC voltages, currents, and other boundary or source quantities. But if the magnitude and phase angle of a quantity are functions of position, specify the functions in terms of real and imaginary components (x+jy). The "x+jy" description of a phasor indicates that the phasor is the sum of two components - a sinusoid that peaks at $\omega t = 0°$and a sinusoid that peaks at $\omega t = 90°$.

- The "x" component of the phasor is the real component and is represented by a phasor that lies on the real axis of the complex plane.
- The "y" component is the imaginary component and is represented by a phasor that lies on the j-axis of the complex plane.

The real and imaginary components of a three-phase system are shown below:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



They are related to the magnitude and phase of a sinusoid in this way:

$$A_m e^{j\theta} = x + jy = A_m\left(\cos\theta + jsin\,\theta\right)$$

$$A_m = \sqrt{x^2 + y^2}$$

$$\theta = atan\left[\frac{y}{x}\right]$$

where the real and imaginary components are

$$Re = A_m \cos\theta$$

$$Im = A_m \sin\theta$$

For a symmetric three-phase system, θ = 0, 120, 240 degrees, respectively.

For an AC Magnetic simulation, all sources (currents, fields) must have the same frequency. Therefore, the simulation frequency is specified as a global input parameter.

In simulations where the electromagnetic radiation is included the displacement, currents are included in the calculation. In such a situation, the dependency between H and J is given by the following equation:

$$\nabla \times \vec{H} = J + \frac{\partial}{\partial t}\left(\varepsilon \vec{E}\right)$$

Solution Process for an AC Magnetic T-Omega Solution

To solve for the magnetic field, **H**, the solver computes the values as follows:

$$\nabla \times \left(\frac{1}{\sigma + j\cdot\omega\cdot\epsilon}\nabla \times H\right) = -j\cdot\omega\cdot\mu\cdot H$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In conductors where eddy currents occur, **H** is computed directly from source currents and applied magnetic fields.

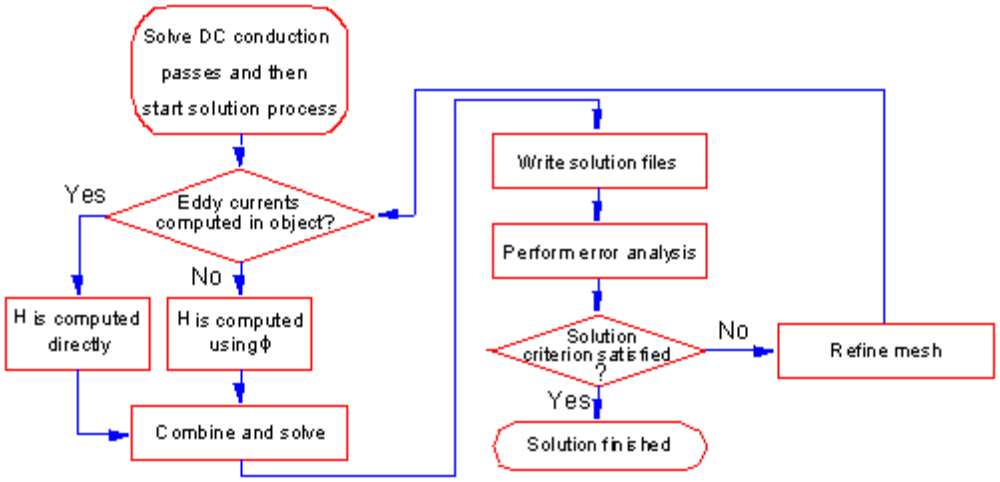
In non-conducting regions, H is computed from the magnetic scalar potential:

$$\nabla \bullet (\mu \nabla \phi) = 0$$

Directly solving for **H** requires more computing resources than using the magnetic potential, so this is done only in regions where the magnetic potential cannot be used.

The solver combines the solutions and solves for the magnetic field. **H** is forced to be continuous on the boundaries, producing a continuous field solution throughout the model. It then saves the completed solution to a file and performs an error analysis. In an adaptive analysis, it refines the tetrahedra with the highest error and continues solving until the stopping criterion is met.

> **Warning:** Always use the peak value in the applied excitations in an eddy current solution!



Nonlinear Field Simulation (3D)

Refer to the following topics for detailed technical information on nonlinear 3D AC Magnetic (Eddy Current) field simulation:

Nonlinear AC Magnetic (Eddy Current) Field Simulation

Nonlinear AC Magnetic (Eddy Current) Theory

Sinusoidal B

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Skin Depth for an AC Magnetic (Eddy Current) T-Omega Solution

Induced currents allow magnetic fields to penetrate conductors only to a certain depth, of about 4-5 skin depths. The skin depth is approximated by the following formula:

$$\delta = \sqrt{\frac{2}{\omega \sigma \mu_0 \mu_r}}$$

where

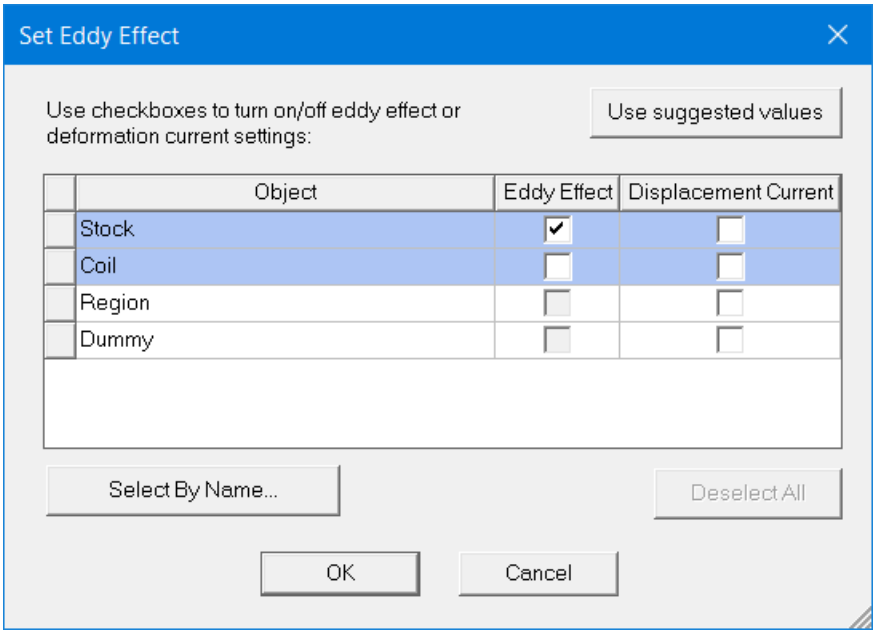
- ω is the angular frequency, which is equal to $2\pi f$ (f is the frequency at which source currents and voltages oscillate during the solution).
- σ is the conductor's conductivity in siemens/meter.
- μ_r is the conductor's relative permeability.
- μ_0 is the permeability of free space, which is equal to 4×10^{-7} H/m.

Currents are concentrated near the surface of the conductor, decaying rapidly past the skin depth. As the formula above indicates, the skin depth gets smaller as the frequency increases.

> **Note:**
>
> Due to the skin concentration of current, AC inductances and resistances are not equal to their DC equivalents. This affects the inductance and resistance values computed during impedance computations.

Both the skin depth / proximity effects and the displacement effects must be specifically requested. Under the excitation setup, the respective effects can be specified for objects in the model as shown below:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Eddy (skin/proximity) effect can only be specified for objects with non-zero conductivity.

By default, eddy/displacement current effects are turned off. Including these in the calculation has a potentially significant impact on the solution time since the electric vector potential calculation is activated, causing the size of the solved matrix to increase significantly if the regions with the electric vector potential to be calculated have a large number of finite elements.

Displacement current calculation is normally activated in dielectrics (including vacuum) and allows electromagnetic waves to propagate. In such cases, a radiation boundary condition should also be used in the problem setup.

Magnetic Field Energy

The energy density in the general case includes the magnetic and electric energy densities. The AC magnetic field energy is given by the following:

$$U = \frac{1}{4} Re\left(B \bullet H^* + \left(E \bullet D^*\right)\right)$$

where:

- ***B*** is the magnetic flux density.
- ***H*** is the magnetic field.
- ***E*** is the electric field.
- ***D*** is the electric displacement.
- *Re* is the real part operation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The superscript * denotes complex conjugate.
- The symbol • is the dot product.

For the sake of brevity, the energy density in the post processor is labeled as **Energy**. The total energy (U) of an object or the whole arrangement can be calculated by integrating the energy density over a proper volume in the Field Calculator, as follows:

$$U = \frac{1}{4}\iiint_{Vol} Re\left(B \bullet H^* + \left(E \bullet D^*\right)\right) dVol$$

This represents the average energy over time, not the instantaneous energy at a specific point in the cycle. (The 1/4 factor is explained by the use of peak values for excitations.)

Hysteresis Loss

For an AC Magnetic (Eddy Current) solution, the hysteresis loss is given by the following

$$U = -\int_{Vol} \omega Im\left[\frac{1}{2}\left(B \bullet H^*\right)\right] dVol$$

where

- ***B*** is the magnetic flux density,
- ***H**** is the complex conjugate of the magnetic field, ***H***, and
- ω is the angular frequency, equal to $2\pi f$ (where f is the solution frequency).

The magnetic hysteresis loss density (p_h) is given by the following:

$$p_h = \frac{1}{2}\omega Im\left(B \bullet H^*\right)$$

where

- *Im* is the imaginary part, and
- ω is the angular frequency.

For the sake of brevity, the hysteresis loss density in the post processor is labeled as **Hysteresis Loss**. The total magnetic hysteresis loss (p_h) of an object or the whole arrangement can be calculated by integrating the hysteresis loss density over a proper volume in the Field Calculator, as follows:

$$p_h = -\frac{1}{2}\omega \iiint_{Vol} Im\left(B \bullet H^*\right) dVol$$

Hysteresis loss applies if the structure includes materials with a non-zero value for the tangent of the magnetic loss angle (equivalent to a non-zero imaginary permeability). Otherwise, it is equal to zero.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ohmic Loss

For an AC Magnetic solution, ohmic loss is given by

$$p = \int_{Vol} \frac{\boldsymbol{J} \bullet \boldsymbol{J}^*}{2\sigma} dVol \quad \left(\text{Watts}\right)$$

where

- ***J*** is the current density.
- ***J**** is the complex conjugate of the current density.
- σ is the conductivity in siemens/meter.

Ohmic loss is used to compute the power loss in a structure ($P=I^2R$).

For impedance boundaries, ohmic loss is given by

$$p = \sqrt{\frac{\omega \mu_0 \mu_r}{8\sigma}} \int_{Sur} \boldsymbol{H}_t \bullet \boldsymbol{H}_t{}^* ds \quad \left(Watts\right)$$

where

- *w* is the angular frequency, which is equal to 2πf. (f is the frequency at which source currents and voltages oscillate during the solution.
- *s* is the conductor's conductivity in siemens/meter.
- μ_r is the conductor's relative permeability.
- μ_0 is the permeability of free space, which is equal to 4π x 10–7 H/m.
- H_t is the tangential component of H on the impedance boundary.
- H_t^* is the complex conjugate tangential component of H on the impedance boundary.

The ohmic loss density (p_o) is given by the following:

$$p_o = \frac{1}{2} Re\left(\boldsymbol{E} \cdot \boldsymbol{J}_c^*\right)$$

This loss is calculated from the conduction component of the current density:

$$\boldsymbol{J}_c = [\sigma]\boldsymbol{E}$$

For the sake of brevity, the ohmic loss density in the postprocessor is labeled as **Ohmic Loss**. The total ohmic loss (p_o) of an object or the whole arrangement can be calculated by integrating the ohmic loss density over a proper volume in the Field Calculator, as follows:

$$p_o = \frac{1}{2} \iiint_{Vol} Re\left(\boldsymbol{E} \cdot \boldsymbol{J}_c^*\right) dVol$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dielectric Loss

For an AC Magnetic (Eddy Current) solution, the dielectric loss density (p_d) is as follows:

$$p_d = \frac{1}{2} Re\left[\boldsymbol{E} \cdot (\boldsymbol{J} - \boldsymbol{J_c})^*\right]$$

The current density and the conduction current density are defined by the following two equations:

$$\boldsymbol{J} = \nabla \times \boldsymbol{H}$$

$$\boldsymbol{J_c} = [\sigma]\boldsymbol{E}$$

For the sake of brevity, the dielectric loss density in the dielectric loss density in the post processor should be labeled as **Dielectric Loss**. The total dielectric loss (p_d) of an object or the whole arrangement can be calculated by integrating over a proper volume in the Field Calculator, as follows:

$$p_d = \frac{1}{2} \iiint_{Vol} Re\left[\boldsymbol{E} \cdot (\boldsymbol{J} - \boldsymbol{J_c})^*\right] dVol$$

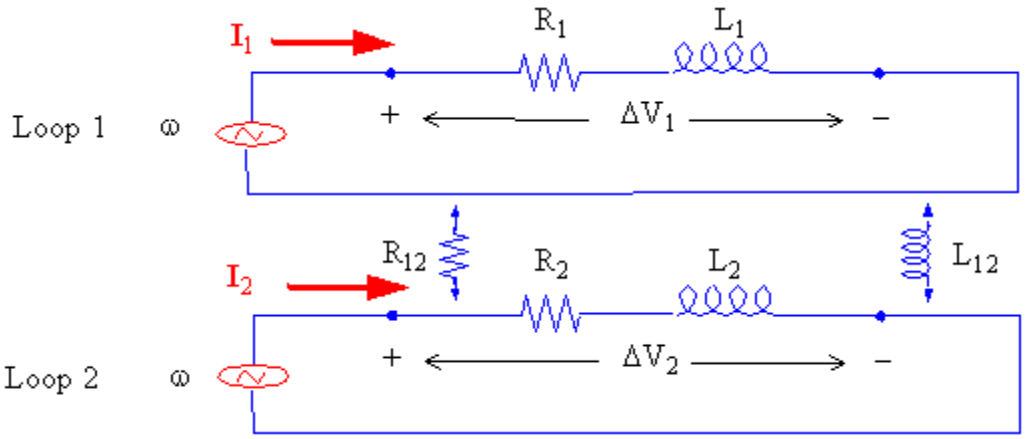
The dielectric loss can also be called *electric hysteresis loss*.

Impedance Matrix

For an AC Magnetic (Eddy Current) solution, an impedance matrix gives the relationship between AC voltages and AC currents for multiple conductors. In the current loops below, the voltage and current in each loop is

$$\Delta V_1 = I_1 R_1 + I_1 j\omega L_{11} + I_2 j\omega L_{12}$$

$$\Delta V_2 = I_2 R_2 + I_2 j\omega L_{22} + I_1 j\omega L_{12}$$



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This can be expressed in matrix form as

$$\begin{bmatrix} \Delta V_1 \\ \Delta V_2 \end{bmatrix} = \begin{bmatrix} Z_{11} & Z_{12} \\ Z_{12} & Z_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix}$$

where

- ΔV_i and I_i are phasors.
- $Z_{11} = R_1 + j\omega L_{11}$ (the self-impedance of Loop 1).
- $Z_{12} = R_{12} + j\omega L_{12}$ (the mutual impedance between Loops 1 and 2).
- $Z_{22} = R_2 + j\omega L_{22}$ (the self-impedance of Loop 2).

A device with n current loops would have an $n \times n$ impedance matrix.

Matrix Elements for an Impedance Matrix

All impedances are complex numbers in the form

$$Z = R + j\omega L$$

where

- ω is the angular frequency of the AC voltages and currents, which is equal to $2\pi f$ (where f is the solution frequency).
- R is the resistance, given in ohms.
- L is the inductance, given in henries.

If one amp flows in Loop 1 and zero amps in Loop 2, the impedance matrix becomes

$$\begin{bmatrix} \Delta V_1 \\ \Delta V_2 \end{bmatrix} = [Z] \begin{bmatrix} 1 \\ 0 \end{bmatrix} = \begin{bmatrix} Z_{11} \\ Z_{12} \end{bmatrix}$$

Diagonal Elements for an Impedance Matrix

The self-impedance of Loop 1 is given by $Z_{11} = R_1 + j\omega L_{11}$.

- R_1 represents the internal resistance of the current loop.
- L_{11} represents the self-inductance of the loop. The $j\omega L_{11}$ term represents the inductive reactance of the loop — the apparent AC inductance due to the loop's self inductance.

Off-Diagonal Elements for an Impedance Matrix

The mutual impedance between Loops 1 and 2 is given by $Z_{12} = R_{12} + j\omega L_{12}$.

- R_{12} represents the mutual resistance of a neighboring conductor as seen from the source conductor.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

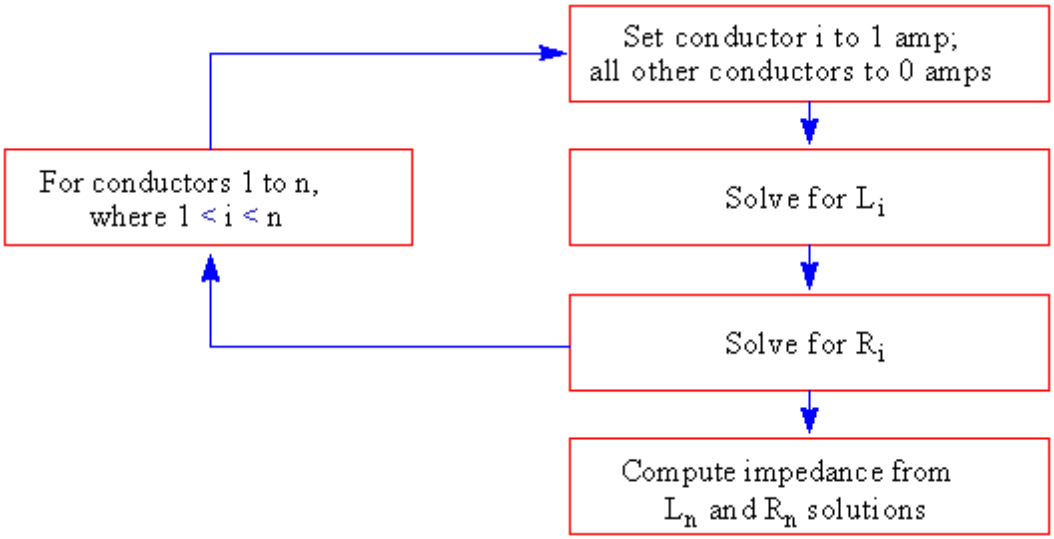
- L_{12} represents the mutual inductance between the loops. The $j\omega L_{12}$ term represents the inductive reactance between the loops — the apparent AC inductance due to the mutual inductance.

Symmetry for an Impedance Matrix

The impedance matrix is symmetric about the diagonal. This indicates that the mutual effects between any two loops are identical.

Solution Process for an Impedance Matrix

The simulator divides the impedance matrix computation into two parts, as shown below.



The solver generates a field solution for each conductor in the matrix. Each conductor is assumed to be part of an independent current loop. In the first solution, the current in the first conductor is set to one amp; currents in the other conductors are set to zero. This is done by imposing current sources on the conductors. In the second solution, the current in the second conductor is set to one amp and all other conductors are set to zero amps, and so forth. Objects that are not included in the impedance matrix are not affected.

Using the field solution as input, the simulator performs inductance and resistance computations. When it finishes solving for the inductance and resistance, the simulator combines them to form the impedance matrix, using the relationship

$$Z_i = R_i + j\omega L_i$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Inductance for an Impedance Matrix

The inductance solution for an impedance computation is similar to a magnetostatic inductance solution. The system computes the following for each matrix entry:

$$L_{ij} = \int B_i \bullet H_j^* d\Omega$$

where

- ***B*** is the magnetic flux density.
- ***H*** is the complex conjugate of the magnetic field.

Resistance for an Impedance Matrix

To find the resistance, the system computes the ohmic loss, P, for each loop

$$P = \frac{1}{2\sigma} \int J \bullet J^* d\Omega$$

where **J** is the current density. Ohmic loss can also be expressed in terms of resistance and total current, $P=RI_{RMS}^2$, where

$$I_{RMS} = \frac{I_{Peak}}{\sqrt{2}}$$

The resistance, therefore, is

$$R = \frac{P}{I_{RMS}^2} = \frac{\frac{1}{2\sigma} \int J \bullet J^* d\Omega}{I_{RMS}^2} = \frac{\int J \bullet J^* d\Omega}{\sigma I_{Peak}^2}$$

To solve this, the system computes the conduction current, **J**, for each conductor. Each subsolution, $\mathbf{J}_i$, represents the contribution of a current loop toward the matrix. After it solves for **J** for all conductors, the system computes the resistance matrix.

> **Note:** Because all current loops are assumed to be independent, an impedance matrix only contains terms for their internal resistance (or self-resistance). It does not include terms for the mutual resistances between loops.

Line Impedance

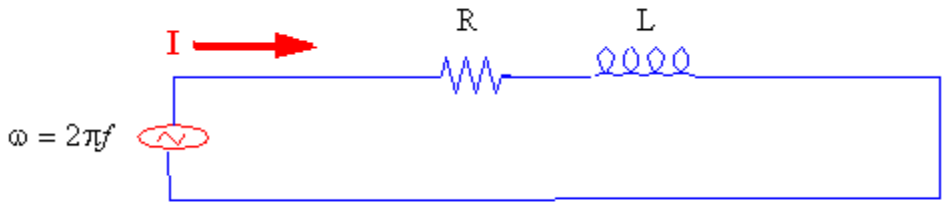
Line impedance is represented by a 1x1 impedance matrix. That is to say, the line impedance is the impedance of a single current loop given by

$$Z = R + j\omega L$$

where

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

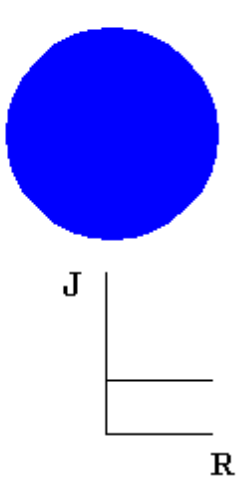
- ω is the angular frequency of the AC voltages and currents, which is equal to $2\pi f$ (where f is the solution frequency).
- R is the self-resistance of the loop.
- L is the self-inductance of the loop.



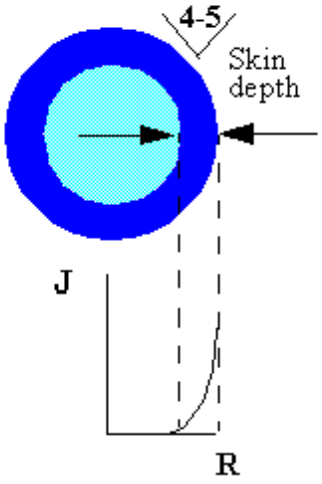
The inductance and resistance components of line impedance are computed separately.

AC Inductance and Resistance for an Impedance Matrix

The inductances and resistances computed during an impedance solution are different from those computed for the equivalent DC case, as shown below. The darker blue area represents the cross-section of the current flowing in the conductors.



DC: **J** is uniformly distributed throughout the conductor.



AC: **J** is concentrated near the surface and decays exponentially toward the conductor's center.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The current density, **J**, in the DC case (the conductor on the left) is evenly distributed throughout the cross-section of the conductor. The current density in the AC case (the conductor on the right) is distributed close to the surface due to skin concentration of currents. Because the area through which current can flow is smaller, it follows that the resistance to the current flow is higher in the impedance matrix than in the equivalent DC case.

In the DC example, no eddy currents occur. The magnetic field created by the current flowing through the conductor is static. In the AC example, the oscillating magnetic field induces currents in other conductors in the model. These induced currents affect the computation of inductance for the impedance matrix, causing it to be different from the equivalent DC inductance.

Lorentz Force

For an AC Magnetic (Eddy Current) solution, the Lorentz force represents the average force on an object (or group of objects) due to currents in a time-varying magnetic field. Lorentz force is given by

$$F(t) = \int_{Vol} J(t) \times B(t) dVol$$

where

- *J(t)* is the current density.
- *B(t)* is the magnetic flux density.

The average force is found by integrating the instantaneous force:

$$F_{AV} = \frac{1}{2\pi} \int_0^{2\pi} F_{Inst} d\omega t = \frac{1}{2\pi} \int_0^{2\pi} \left(\int_{Vol} J(t) \times B(t) dVol \right) d\omega t$$

> **Warning:** The Lorentz force does not compute the correct average force on objects that are assigned materials with a relative permeability greater than one. To find the force on these objects, use virtual force.

Lorentz Torque

For an AC Magnetic (Eddy Current) solution, the Lorentz torque represents the average torque on an object or group of objects due to currents in a time-varying magnetic field. The Lorentz torque is given by

$$T(t) = \int_{Vol} r \times [J(t) \times B(t)] dVol$$

where

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- *r* is the displacement vector from the axis of rotation.
- *J(t)* is the current density.
- ***B****(t)* is the magnetic flux density.

The average torque is found by integrating the instantaneous torque:

$$T_{AV} = \frac{1}{2\pi}\int_0^{2\pi} T_{Inst}\, d\omega t = \frac{1}{2\pi}\int_0^{2\pi}\left(\int_{Vol} r \times [J(t) \times B(t)] dVol\right) d\omega t$$

This integral is computed for the x-, y- and z-axes, giving the average torque on the object(s) about each axis of rotation.

> **Warning:** The Lorentz torque does not compute the correct average torque on objects that are assigned materials with a relative permeability greater than one. To find the torque on these objects, use virtual torque.

Virtual Force

For an AC Magnetic (Eddy Current) solution, the virtual force is computed the same way as the magnetostatic virtual force. The only difference is that the average force over time is computed — not the net (instantaneous) force at a given time. The average virtual force is found by integrating the instantaneous force:

$$F_{AV} = \frac{1}{2\pi}\int_0^{2\pi} F_{Inst}\, d\omega t = \frac{1}{2\pi}\int_0^{2\pi}\left(\frac{dW(s,i)}{ds}\right) d\omega t$$

Virtual Torque

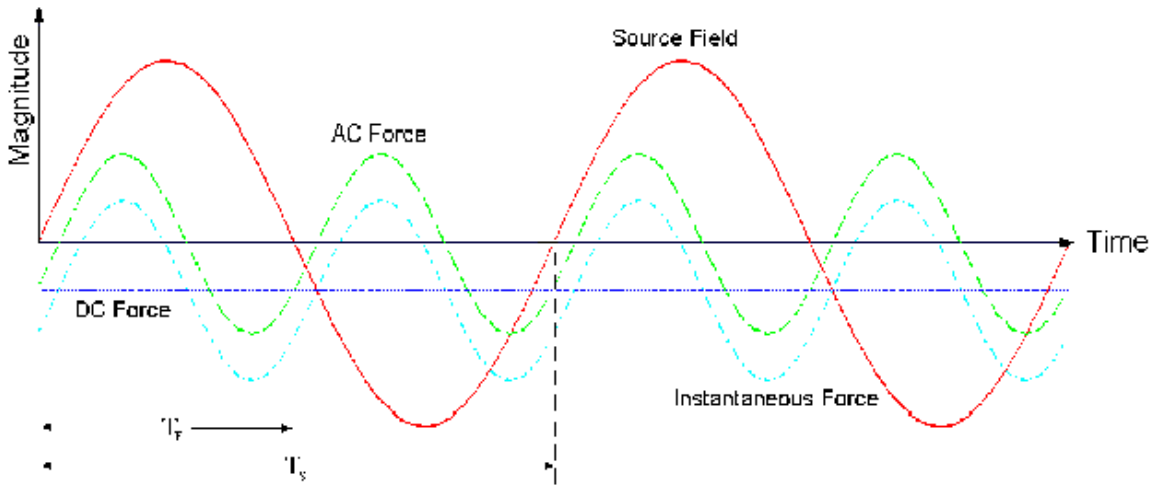
The virtual torque for an AC Magnetic (Eddy Current) solution is computed the same way as the magnetostatic virtual torque. The only difference is that the average torque over time is computed — not the net torque at a given time. The average torque is found by integrating the instantaneous torque:

$$T_{AV} = \frac{1}{2\pi}\int_0^{2\pi} T_{Inst}\, d\omega t = \frac{1}{2\pi}\int_0^{2\pi}\left(\frac{dW(\theta,i)}{d\theta}\right) d\omega t$$

Average Force

The Lorentz force and virtual force computed for an AC Magnetic (Eddy Current) model represent the average force, not the instantaneous force at a specific point in the AC magnetic field's cycle. Similarly, the Lorentz torque and virtual torque represent the average torque over time, since these torque computations make use of the time-averaged force. The difference between the time-averaged (or DC) force, AC force, and instantaneous force is shown below:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Force oscillates at twice the frequency of the source current and magnetic field:

$$f_F = \frac{1}{T_F} = 2f_S$$

where

- f_F is the frequency of the force.
- f_S is the frequency of the source current and magnetic field.
- T_F is the period of the force.

The time-averaged (or DC) force, AC force, and instantaneous force can be determined by

$$F_{DC} = \frac{1}{2}\int Re\left|\overline{J} \times \overline{B}^*\right| dV$$

$$F_{AC} = \frac{1}{2}\int Re\left|\overline{J} \times \overline{B}\right| dV$$

$$F_{INST} = F_{DC} + F_{AC}$$

The AC force, F_{AC} must be evaluated at a particular phase (=ωt) in order to determine its magnitude at an instant in time.

AC Magnetic (Eddy Current) A-Phi Solver (Beta)

> **Note:** The AC Magnetic 3D A-Phi solver is only accessible if it has been activated from **Tools** > **Options** > **General Options** > **Desktop Configuration** > **Beta Options** window. Select the **Maxwell AC Magnetic A-Phi Solution** feature.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The 3D AC Magnetic (Eddy Current) A-Phi formulation, which is selected in the Solution Type window, is based on the electric scalar potential, Φ, and the magnetic vector potential, **A**. The relations between those potentials and the fields **B** and **E** are given by

$$\boldsymbol{B} = \nabla \times \boldsymbol{A}$$

$$\boldsymbol{E} = -j\omega\boldsymbol{A} - \nabla\Phi$$

With these equations and the material constitutive relations,

$$\boldsymbol{B} = \mu H, \text{ or } \boldsymbol{H} = \nu\boldsymbol{B}$$

$$\boldsymbol{J} = \sigma\boldsymbol{E}$$

$$\boldsymbol{D} = \varepsilon\boldsymbol{E}$$

Ampere's law and the current continuity equation, $\nabla \cdot J = 0$, can be rewritten as

$$\nabla \times \nu\nabla \times \boldsymbol{A} + \left(j\omega\sigma - \omega^2\varepsilon\right)\boldsymbol{A} + (\sigma + j\omega\varepsilon)\nabla\Phi = \boldsymbol{J}_{imp}$$

$$-\nabla \cdot \left[\left(j\omega\sigma - \omega^2\varepsilon\right)\boldsymbol{A} + (\sigma + j\omega\varepsilon)\nabla\Phi\right] = 0$$

where

- **H** is the magnetic field.
- $\mathbf{J}_{imp}$ is an imposed current density.
- **D** is the electric induction field.
- **E** is the electric field.
- **B** is the magnetic induction field.
- ν is the magnetic reluctivity.
- μ is the magnetic permeability.
- σ is the electric conductivity.
- ε is the electric permittivity.

The weak form of the above-presented equations is

$$\int_{\Omega} \nu\nabla \times \boldsymbol{A}\left(\nabla \times \boldsymbol{A}'\right)d\Omega + \int_{\Omega}\left(j\omega\sigma - \omega^2\varepsilon\right)\boldsymbol{A}\cdot\boldsymbol{A}'d\Omega + \int_{\Omega}(\sigma + j\omega\varepsilon)\nabla\Phi\cdot\boldsymbol{A}'d\Omega = \int_{\Omega} J_{imp}\cdot\boldsymbol{A}'d\Omega$$

$$\int_{\Omega}\left(j\omega\sigma - \omega^2\varepsilon\right)\boldsymbol{A}\cdot\nabla\Phi'd\Omega + \int_{\Omega}(\sigma + j\omega\varepsilon)\nabla\Phi\cdot\nabla\Phi'd\Omega = \int_{\Gamma_J}\left(\boldsymbol{n}\cdot\boldsymbol{J}\right)\Phi'd\Gamma_J$$

Understanding the terms of this equation can clarify its capabilities:

- $j\omega\sigma\mathbf{A}$ is the induced current.
- $\omega^2\varepsilon\mathbf{A}$ is the wave propagation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- $\sigma\nabla\Phi$ is the conduction current.
- $j\omega\varepsilon\nabla\Phi$ is the displacement current.

In the case of low-frequency applications, the wave propagation and displacement effect terms can be neglected, leading to the low-frequency version of this formulation:

$$\int_{\Omega} \nu\nabla \times \boldsymbol{A}\left(\nabla \times \boldsymbol{A}'\right)d\Omega + \int_{\Omega_c} j\omega\sigma\boldsymbol{A}\cdot\boldsymbol{A}'d\Omega + \int_{\Omega_c} \sigma\nabla\Phi\cdot\boldsymbol{A}'d\Omega = \int_{\Omega_S} \boldsymbol{J}_{imp}\cdot\boldsymbol{A}'d\Omega$$

$$\int_{\Omega_c} j\omega\sigma\boldsymbol{A}\cdot\nabla\Phi'd\Omega + \int_{\Omega_c} \sigma\nabla\Phi\cdot\nabla\Phi'd\Omega = \int_{\Gamma_J} (\boldsymbol{n}\cdot\boldsymbol{J})\Phi'd\Gamma_J$$

in which only the eddy current and the conduction terms are considered.

One important aspect of the low-frequency version is that Φ is defined only in the conductive domains, Ω_C, which reduces the number of unknowns and, consequently, the memory and simulation time. The user can choose between these versions of the A-Phi solver by turning on or off the displacement current effect.

Nonlinear Solution

In a steady-state condition, unlike linear problems in which the field solution is assumed to be based on the sinusoidal peak field values, nonlinear problems are usually based on all harmonic components of the fields.

In the 3D AC Magnetic (Eddy Current) A-Phi solver, the vector field **B** is sinusoidal once it is obtained from $\mathbf{B} = \nabla \times \mathbf{A}$. The **H** field is then computed from the B-H nonlinear property as stated in the constitutive relation:

$$\boldsymbol{H} = \nu(|\boldsymbol{B}|)\boldsymbol{B}$$

Consequently, the magnetic field **H** can be "harmonically rich" due to the material's nonlinearity. This solver uses an effective property based on the sinusoidal **B** to approximate nonlinear field behavior associated with the assigned nonlinear B-H characteristic. Using the B-H curve, the effective reluctivity is expressed by

$$\nu_{eq} = \left(\frac{2}{T}\int_0^{\frac{T}{2}} \nu^2(t)dt\right)^{\frac{1}{2}}$$

The above equation is used to obtain the effective material property for each B field value in the original curve, allowing the generation of a new curve based on the effective properties.

Skin Effect

Induced currents allow magnetic fields to penetrate conductors only to a depth of about 4-5 skin depths. The skin depth is approximated by the following formula:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\delta = \sqrt{\frac{2}{\omega\sigma\mu_0\mu_r}}$$

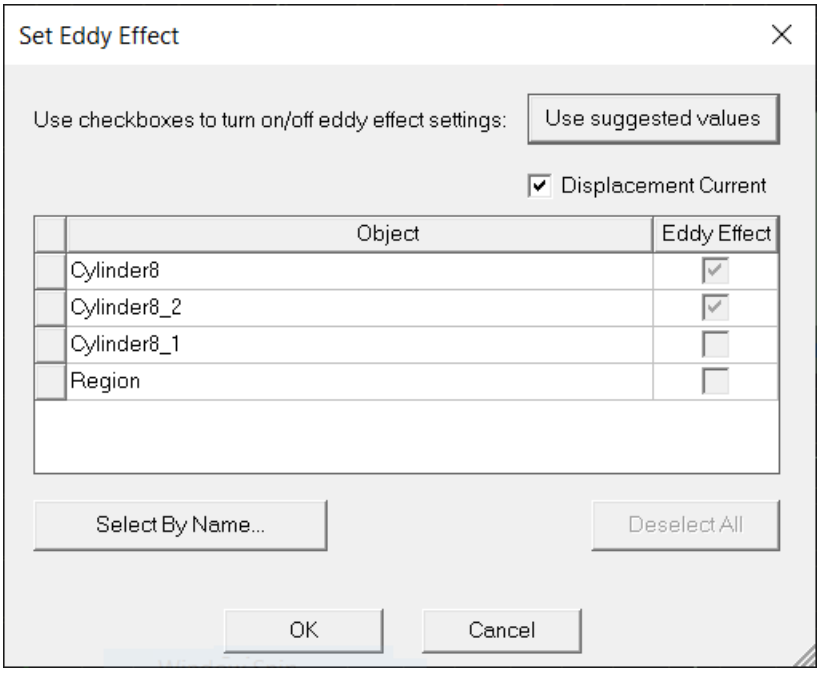
where

- ω is the angular frequency, which is equal to $2\pi f$ (f is the frequency at which source currents and voltages oscillate during the solution).
- σ is the conductor's conductivity in Siemens/meter.
- μ_r is the conductor's relative permeability.
- μ_0 is the permeability of free space, which is equal to $4\pi 10^{-7}$ H/m.

Currents are concentrated near the surface of the conductor, decaying rapidly past the skin depth. As the formula above indicates, the skin depth gets smaller as the frequency increases.

> **Note:**
>
> Due to the skin concentration of current, AC inductances, resistances, and in some cases even the capacitances are not equal to their DC equivalents.

For the 3D AC Magnetic A-Phi solver, the skin effect is turn on automatically; it does not have to be requested in the **Excitations** > **Set Eddy Effects** window. Note that the **Eddy Effect** setting is grayed out because it cannot be turned off:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

It is important to note that, as shown in the above equation, the skin effect only develops in domains with $\sigma > 0$; consequently, non-conductive domains are automatically unchecked.

Activating eddy effects also allows the capture of the proximity effect.

Field Definitions for the AC Magnetic A-Phi Solver

The fields can have different definitions in harmonic problems, mainly when dealing with lossy materials. This section details the fields available in the AC Magnetic (Eddy Current) A-Phi solver.

Field Name	Type Definition
Voltage: electric scalar potential Φ	scalar and complex
Magnetic Induction	vector and complex $\boldsymbol{B} = \nabla \times \boldsymbol{A}$
Magnetic Field	vector and complex $\boldsymbol{H} = \nu(\lvert\boldsymbol{B}\rvert)\boldsymbol{B}$
Electric field (E)	vector and complex $\boldsymbol{E} = -j\omega\boldsymbol{A} - \nabla\Phi$
Total current density (J)	vector and complex $\boldsymbol{J} = (\sigma + j\omega\varepsilon)\boldsymbol{E}$ $j\omega\varepsilon$ only if the displacement effect is turned on
Electric flux density (D)	vector and complex $\boldsymbol{D} = \dot{\varepsilon}\,(\boldsymbol{E})\boldsymbol{E}$ where $\dot{\varepsilon} = \varepsilon' - j\varepsilon''$ and $\varepsilon'' = \varepsilon' tan\delta$.
Conduction current density (J_c)	vector and complex $\boldsymbol{J_c} = \sigma_s\,(\boldsymbol{E})\boldsymbol{E}$ where σ_s is the bulk conductivity
Energy	$\boldsymbol{W} = \frac{1}{4} Re\left\lvert\boldsymbol{B}\cdot\boldsymbol{H}^* - \boldsymbol{D}\cdot\boldsymbol{E}^*\right\rvert$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

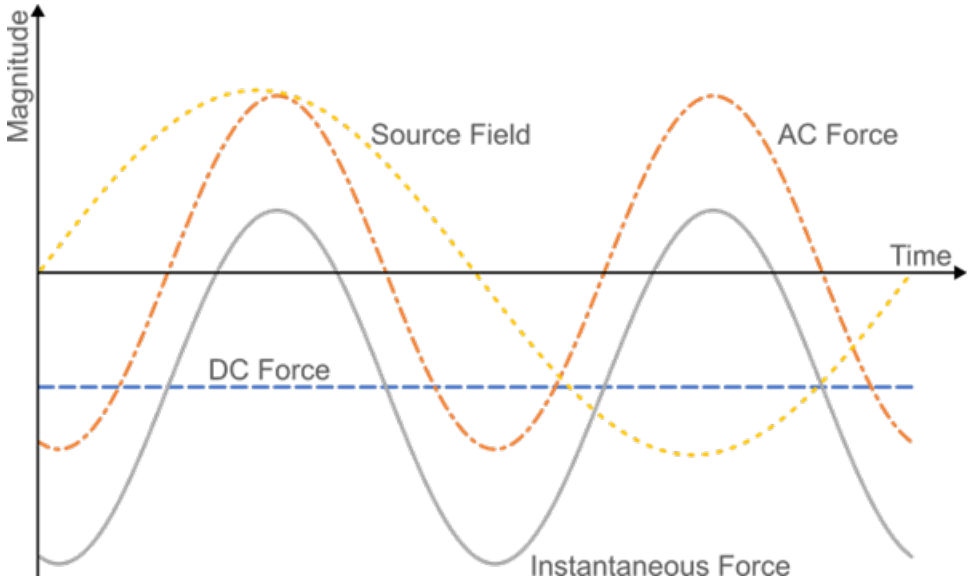
Field Name	Type Definition
Ohmic loss	scalar and real $P_{ohmic} = \frac{1}{2} Re\left\{J_c \cdot E^*\right\}$ or $P_{ohmic} = \frac{1}{2} Re\left\{\sigma_s \left(\boldsymbol{E} \cdot \boldsymbol{E}^*\right)\right\}$
Dielectric loss	scalar and real $P_{delectric} = \frac{1}{2} Re\left\{(\boldsymbol{J} \cdot \boldsymbol{J_c}) \cdot \boldsymbol{E}\right\}$ or $P_{delectric} = \frac{1}{2} Re\left\{\omega \varepsilon'' \left(\boldsymbol{E} \cdot \boldsymbol{E}^*\right)\right\}$
Hysteresis loss	scalar and real $P_{hyst} = -\frac{1}{2} \omega Im\left\{\boldsymbol{B} \cdot \boldsymbol{H}\right\}$

Parameters Extraction

Force and Torque

The force computed for an AC Magnetic model in the harmonic magnetic field's cycle at a given position represents the average force, not the instantaneous force. Similarly, the torque represents the average torque over time because the torque computations make use of the time-averaged force. The difference between the time-averaged (or DC) force, AC force, and instantaneous force is shown below:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The instantaneous Lorentz force is given by

$$F_{INST} = F_{DC} + F_{AC}$$

where

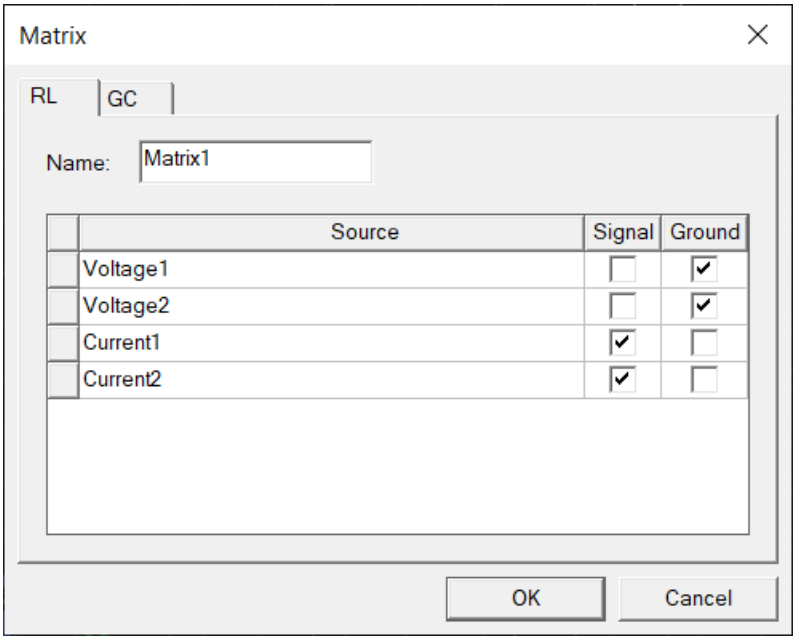
$$F_{DC} = \frac{1}{2}\int_{\Omega} Re\left|\boldsymbol{J} \times \boldsymbol{B}^{*}\right| d\Omega$$

$$F_{AC} = \frac{1}{2}\int_{\Omega} \left|\boldsymbol{J} \times \boldsymbol{B}^{*}\right| d\Omega$$

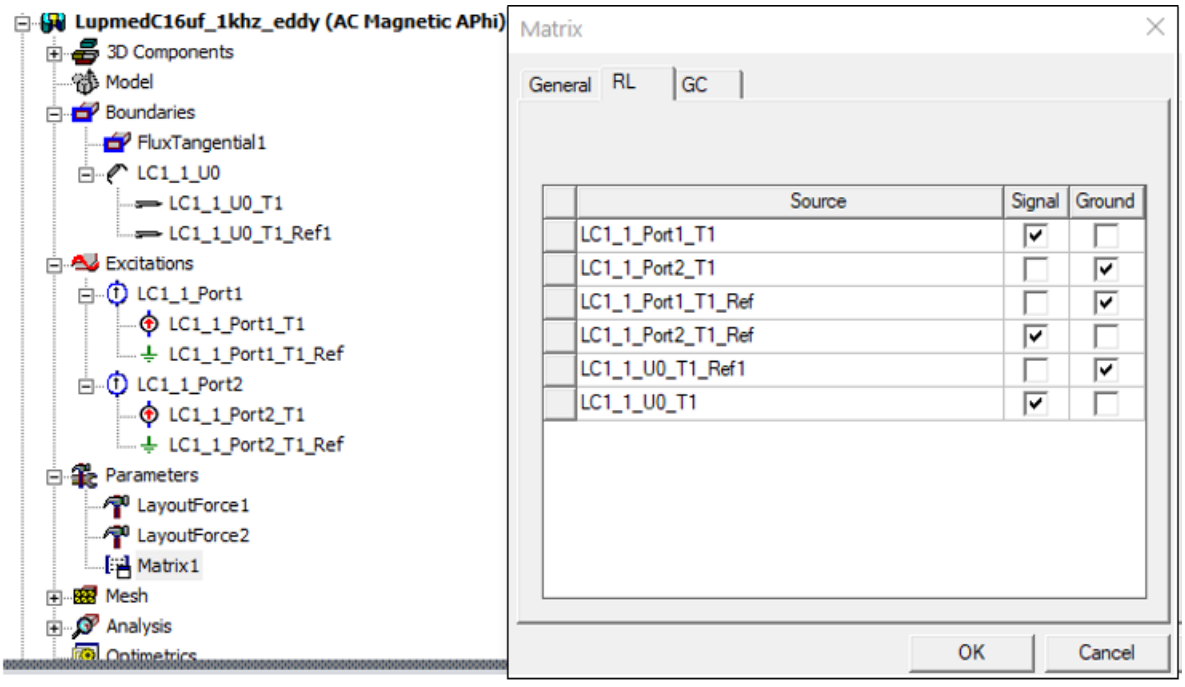
RL and GC Matrices

Unlike the 3D AC Magnetic T-Omega solver, the 3D AC Magnetic A-Phi solver supports the extraction of both RL and GC matrices. The matrices are set up from the **Parameters** > **Assign** > **Matrix** window; each matrix has its own tab and must be configured before simulation.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



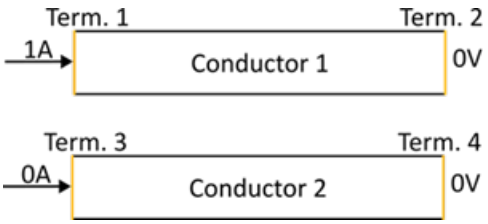
If there is a layout component with lumped circuit elements, the port terminals and circuit element terminals are also listed in the RL and GC matrix setups:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RL Matrix

The RL parameters extraction is performed by consecutively applying a source of 1 A to each terminal set as Signal. The other terminals set as Signal are considered to be a source of 0 A, and the terminals set as Grounded are set to 0 V. Each conduction path with a Signal terminal requires another terminal set to Ground.



If the excitation has a direction vector defined, such as a current excitation with a single potential model or voltage or current excitations with double potential models, the applied current will follow that direction. If the excitation does not have a direction defined, such as a single potential voltage excitation model, the applied current will be pointing inwards the domain.

This process is repeated for each terminal set as Signal, and the parameters are obtained as follows:

$$R = \int_{\Omega} Re\left\{\boldsymbol{E}\cdot\boldsymbol{J}^{*}\right\}d\Omega - \omega\int_{\Omega} Im\left\{\boldsymbol{A}\cdot\boldsymbol{J}^{*}\right\}d\Omega$$

$$L = \int_{\Omega} Re\left\{\boldsymbol{A}\cdot\boldsymbol{J}^{*}\right\}d\Omega$$

GC Matrix

The GC matrix extraction follows the same sequence as in the RL matrix, but applies 1 V to each terminal set as Signal and 0 V to the other Signal and Grounded terminals. For the GC matrix, if there is no ground selected, at least two signals must be selected.

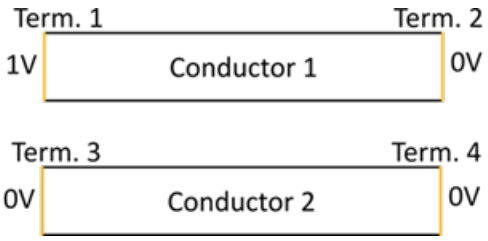
The process is repeated for each Signal terminal, and the parameters are extracted by

$$G = \omega\varepsilon''\int_{\Omega} Re\left\{\boldsymbol{E}\cdot\boldsymbol{E}^{*}\right\}d\Omega$$

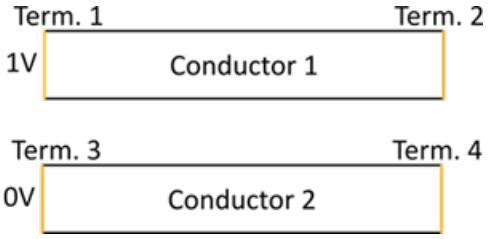
$$C = \int_{\Omega} Re\left\{\boldsymbol{E}\cdot\boldsymbol{D}^{*}\right\}d\Omega$$

For the matrix setup, it might seem intuitive to set the other terminals of the conduction paths to 0 V as in the setup for the RL matrix.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



However, with this excitation setup, there will be a voltage drop of 1 V within the Conductor 1. In the case of good conductors, an equipotential body is expected, but with this setup, there is no equipotential body. A better approach for the GC matrix extraction is to select only one terminal per conduction path as Signal and leave the others unchecked so that specific domain will behave as an equipotential body.

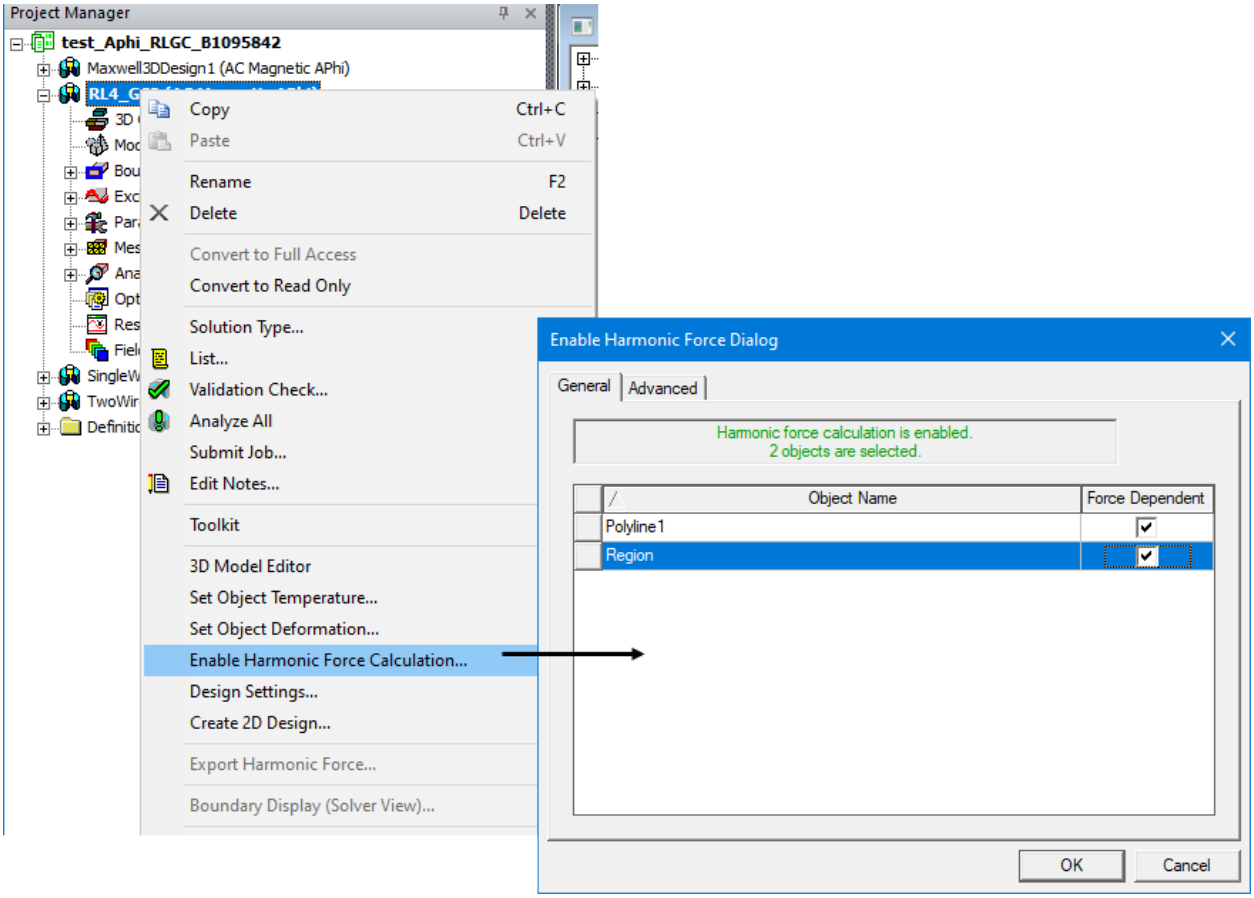


Multiphysics Coupling

The AC Magnetic A-Phi solver supports thermal two-way coupling and stress one-way coupling. Temperature-dependent properties are supported, except for the temperature-dependent B-H curves.

In terms of harmonic response coupling, the AC magnetic A-Phi solver supports object-based harmonic force calculation:

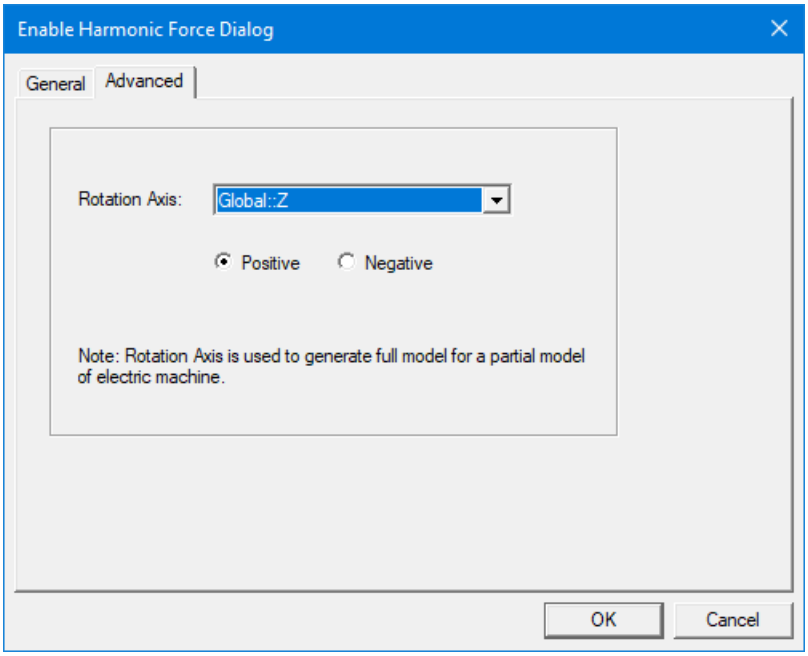
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



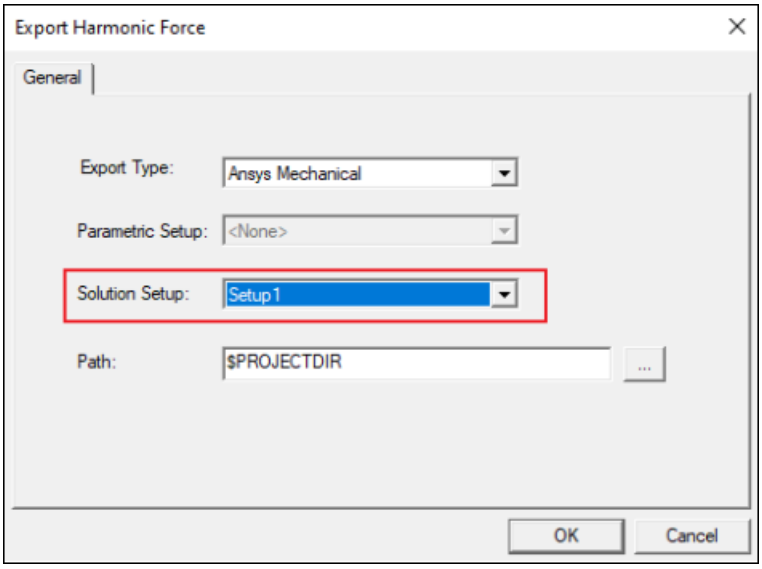
From the **General** tab of the Enable Harmonic Force dialog box, select the objects you want to perform the harmonic force calculation on. If there is a layout component in the design, there will also be a **Layout** tab from which you can select layers and nets for the calculation.

From the **Advanced** tab, you can select an axis in the desired reference frame on which to perform the calculation:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To export volumetric-based harmonic force in selected conductors in .csv format (for PCB applications), right click on the model in the Project Manager tree, and select **Export Harmonic Force**. Specify the solution setup to export and the target output directory. Note that if **Solution Setup** is left blank, the harmonic force will be exported for all solved setups.



3D AC Conduction Solver

AC conduction analysis is the study of

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- electric field,
- losses arising in conductors and imperfect (lossy) dielectrics from the application of an alternating (AC) voltage or external current to the electrodes, and
- C and G parameters extraction.

In this harmonic formulation, the magnetic fields are considered negligible, and the electromagnetic field can be treated as electroquasistatic. The electric field is derived from the electric scalar potential Φ.

One can use this analysis to determine the voltage, electric field, electric flux density, and electric current density distributions, capacitive effects, extract the C and G parameters in electric devices as a function of frequency in response to time-harmonic loading.

Complex equations are used to represent harmonic time-dependency as outlined below.

Governing Equations

Given the Ampere's Law in frequency domain,

$$\nabla \times H = J_c + j\omega D$$

it is possible to obtain the continuity equation solved by the 3D AC conduction solver,

$$\nabla \cdot (\mathbf{J}_c + j\omega \mathbf{D}) = 0$$

where

- **H** is the magnetic field
- $\mathbf{J_c}$ is the conduction current density
- ω is the angular frequency at which the potential is oscillating
- **D** is the displacement flux density.

Constitutive Relations

The constitutive relations are given by

$$\mathbf{J}_c = \sigma_s \left(E\right) \mathbf{E}$$

and

$$\mathbf{D} = \dot{\varepsilon} \ \left(E\right) \mathbf{E}$$

where

- σ_s is the static field electrical conductivity
- $\dot{\varepsilon}$ is the complex dielectric permittivity: $\dot{\varepsilon} = \varepsilon' - j\varepsilon''$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Continuity Equation Including Lossy Domains

Applying the above constitutive relations in the continuity equation, one obtains

$$\nabla \cdot (\sigma_s \mathbf{E} + j\omega\dot{\varepsilon} \mathbf{E}) = 0$$

$$\nabla \cdot [(\sigma_s + \omega\varepsilon'') \mathbf{E} + j\omega\varepsilon' \mathbf{E}] = 0$$

$\sigma_s + \omega\varepsilon''$ represents the equivalent conductivity (σ_e) composed by a static (σ_s) and an alternating (σ_a) part, which is caused by dipole rotation when the material is submitted to an alternating electrical field.

$$\sigma_e = \sigma_s + \omega\varepsilon'' = \sigma_s + \sigma_a$$

Finally, the continuity equation can be written as:

$$\nabla \cdot [\nabla\Phi (\sigma_s + \omega\varepsilon'' + j\omega\varepsilon')] = 0$$

It is of importance to mention that normally the dielectric lossy materials are defined in terms of the dielectric loss tangent (tan δ). In this case, the alternating tan δ is considered:

$$\tan\delta_a = \frac{\varepsilon''}{\varepsilon'}$$

Consequently,

$$\varepsilon'' = \tan\delta_a \varepsilon'$$

Time-Harmonic Field Assumptions

The AC conduction field solver assumes the following conditions about field quantities:

- All time-varying electromagnetic quantities have the periodic waveform

  $$F(t) = F_m \cos(\omega t + \theta)$$

  where all quantities must have the same value of ω, but can have different phase angles (θ).
- If a current is not a pure sinusoid, it is decomposed into sinusoidal harmonics, and solved separately at each frequency.
- The component of **E** due to time-varying magnetic fields caused by conduction currents can be neglected.

Nonlinear Solution - 3D AC Conduction Solver

Unlike linear problems in which the field solution is assumed to be based on the sinusoidal complex peak field values in a steady state, nonlinear problems are usually based on all

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

harmonic components of the fields.

The 3D AC Conduction solver supports the J-E and D-E nonlinear properties, as stated in the constitutive relations:

$$\mathbf{J}_c = \sigma_s\,(E)\ \mathbf{E}$$

$$\mathbf{D} = \dot{\varepsilon}\ \ (E)\ \mathbf{E}$$

Thus for nonlinear problems, though the electric field is sinusoidal due to sinusoidal excitations, the resulting D and/or J fields can be “harmonically rich” due to the material nonlinearity.

Although the D and J fields are not sinusoidal at output, the Maxwell 3D AC Conduction solver uses an effective property based on the sinusoidal E to approximate nonlinear field behavior associated with the assigned nonlinear JE and DE characteristic. Using the JE curve as an example, the effective conductivity is expressed by

$$\sigma_{eff} = \frac{\frac{1}{T}\int_0^T J(|E|,t)dt}{\frac{2}{\pi}\widehat{E}}$$

The above equation is used to obtain the effective material property for each electric field value in the original curve, allowing to generate a new curve based on the effective properties. During the simulation process, the effective curve is used to calculate the conductivity and permittivity.

> **Note:** Though the complex permittivity *($\dot{\varepsilon} = \varepsilon' - j\varepsilon''$)* is considered in the constitutive relation, the D-E curve describes only its real part ε'.

Field Definitions for the 3D AC Conduction Solver

The fields can have different definitions in harmonic problems, mainly when dealing with lossy materials. In this section the fields available in the 3D AC conduction solver are presented.

Field Name	Type Definition
Voltage: electric potential Φ	scalar and complex
Electric field (E)	vector and complex $E = -\nabla\Phi$
Electric flux density (D)	vector and complex $\mathbf{D} = \dot{\varepsilon}\ \ (E)\ \mathbf{E}$ where: $\dot{\varepsilon} = \varepsilon' - j\varepsilon''$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Field Name	Type Definition
Total current density (J)	vector and complex $J = (\sigma_s(E) + \omega\varepsilon'' + j\omega\varepsilon'(E))E$
DC current density (Jc)	vector and complex $\mathbf{J}_c = \sigma_s(E)\ \mathbf{E}$
Ohmic loss	scalar and real $P_{ohmic} = \frac{1}{2}Re\{J_c \cdot E^*\}$ or $P_{ohmic} = \frac{1}{2}Re\{\sigma_s(E \cdot E^*)\}$
Dielectric loss	scalar and real $P_{dielectric} = \frac{1}{2}Re\{(J - J_c) \cdot E\}$ or $P_{dielectric} = \frac{1}{2}Re\{\omega\varepsilon''(E \cdot E^*)\}$
Energy	scalar and real $Energy = \frac{1}{4}Re\{\mathbf{D} \cdot \mathbf{E}^*\}$
Surface charge density (Qsurf)	scalar and complex $n \cdot (D_2 - D_1) = \rho_s$ or $n \cdot (\varepsilon_2 E_2 - \varepsilon_1 E_1) = \rho_s$ where • E_1 is the electric field and ε_1 the permittivity in the material on one side of the boundary. • E_2 is the electric field and ε_2 the permittivity in the material on opposite side of the boundary. • n is the normal vector of the surface.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Losses Calculation - 3D AC Conduction Solver

The average apparent power along the domain Ω is given by

$$S = \frac{1}{2}\int_{\Omega} J^* \cdot E d\Omega$$

Consequently, the active (including Ohmic and dielectric) and reactive powers can be expressed by its real and imaginary parts, as follows

$$P_{active} = \frac{1}{2} Re\left\{\int_{\Omega} J^* \cdot E d\Omega\right\}$$

$$P_{reactive} = \frac{1}{2} Im\left\{\int_{\Omega} J^* \cdot E d\Omega\right\}$$

Virtual Force

For an AC Conduction solution, virtual force in an AC problem is computed the same way as the magnetostatic virtual force. The only difference is that the average force over time is computed – not the net (instantaneous) force at a given time. The average virtual force is found by integrating the instantaneous force:

$$F_{AV} = \frac{1}{2\pi}\int_0^{2\pi} F_{Inst}\, d\omega t = \frac{1}{2\pi}\int_0^{2\pi} \left(\frac{dW(s,i)}{ds}\right) d\omega t$$

Virtual Torque

For a 3D AC Conduction simulation, virtual torque is computed the same way as the magnetostatic virtual torque. The only difference is that the average torque over time is computed – not the net torque at a given time. The average torque is found by integrating the instantaneous torque:

$$T_{AV} = \frac{1}{2\pi}\int_0^{2\pi} T_{Inst}\, d\omega t = \frac{1}{2\pi}\int_0^{2\pi} \left(\frac{dW(\theta,i)}{d\theta}\right) d\omega t$$

Admittance - 3D AC Conduction Solver

Admittance can best be explained as the inverse of impedance, and is expressed by:

$$Y = G + j\omega C$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where

- ω is equal to $2\pi f$, where f is the frequency of the AC voltage source.
- Y is the admittance in Siemens.
- G is the conductance in Siemens.
- ωC is the susceptance in Siemens.

> **Note:** The matrix values displayed by the software are conductance in ohms and capacitance in farads (not conductance and susceptance) and therefore do not include jω.

Admittance Matrix - 3D AC Conduction Solver

The electric parameters are obtained by the equivalence between the dissipated power in terms of discrete and continuous model.

In the discrete circuit, neglecting the inductive term, the average apparent power in the complex formalism is given by

$$S = \frac{1}{2} U I^* = \frac{1}{2} V^2 Y = \frac{1}{2} V^2 \left(G + j\omega C\right)$$

In the continuous model,

$$\mathrm{S} = \frac{1}{2} \int_{\Omega} \boldsymbol{J}^* \cdot \boldsymbol{E} \, \mathrm{d}\Omega$$

Consequently, one can write

$$S = \frac{1}{2} V^2 \left(G + j\omega C\right) = \frac{1}{2} \int_{\Omega} J^* \cdot E \, d\Omega$$

As the extraction of parameters is performed by applying a source of 1 V at each port, the admittance can be obtained by integrating the power along the domain:

$$Y = G + j\omega C = \int_{\Omega} J^* \cdot E \, d\Omega$$

This process is repeated N (number of excitations) times and the parameters are obtained as follows:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$G = Re\left\{\int_{\Omega} J^* \cdot E \, d\Omega\right\}$$

$$C = \frac{1}{\omega} Im\left\{\int_{\Omega} J^* \cdot E \, d\Omega\right\}$$

Phasor Notation

> **Note:** The information in this topic is applicable to the 3D AC Magnetic T-Omega and A-Phi solvers and to the 3D AC Conduction solver.

Time varying quantities that have the form

$$F(t) = F_m \cos(\omega t + \theta)$$

can be represented as rotating phasors in the complex plane. Using Euler's formula,

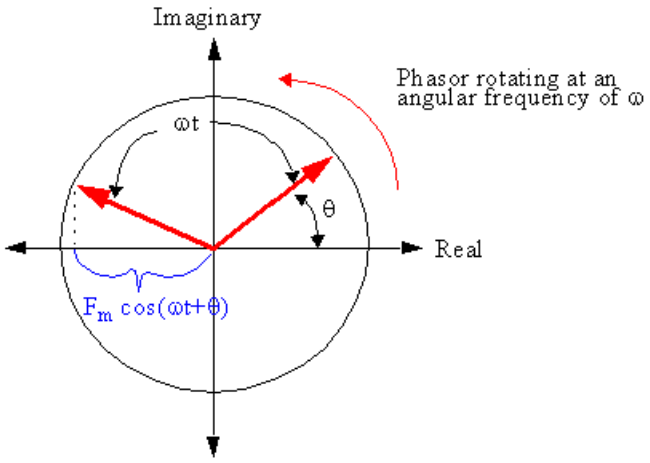
$$e^{j\alpha} = \cos\alpha + j \sin\alpha$$

If $\alpha = \omega t + q$, **F**(t) equals the real portion of $e^{j(\omega t+\theta)}$:

$$F(t) = \Re\left[F_m e^{j(\omega t+\theta)}\right] = \Re\left[F_m(\cos(\omega t+\theta) + j\sin(\omega t+\theta)\right] = F_m \cos\left(\omega t+\theta\right)$$

Each time-varying quantity has the form $F_m e^{j\theta} e^{j\omega t}$. The component $F_m e^{j\theta}$ is a complex constant that can be represented by a stationary phasor in the complex plane. The $e^{j\omega t}$ component is a complex number that depends on *t* and can be represented as a rotating phasor in the complex plane. The phasor's projection on the real axis oscillates sinusoidally. It reaches a peak when parallel with the real axis, and crosses zero when parallel with the imaginary axis. Therefore, a phasor with $\theta = 90^\circ$ represents a quantity that peaks 90 degrees after a phasor with $\theta = 0^\circ$.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Iterative Matrix Solver Technical Details

Two iterative solvers (PCG and QMR) are supported in Maxwell. The PCG is supported in the magnetostatic and electrostatic solvers and QMR is supported in the AC Magnetic solver.

For large simulations, the iterative solvers save significant memory – easily a factor two or more, and may be also faster than the direct solver.

Consider the matrix equation:

$$Ax = b \quad (1)$$

where *A* is a matrix, *b* is the right-hand side and *x* is the solution.

When A^{-1} is computationally expensive or the exact solution *x* is impossible, an alternative is to seek an approximation $\tilde{x}$ to *x*, with an error $e = x - \tilde{x}$. The exact solution can therefore be rewritten as

$$e = x - \tilde{x} \quad (2)$$

Substituting (2) into (1) results in the so-called residual equation:

$$Ae = r \quad (3)$$

where *r* is called the residual defined by

$$r = b - a\tilde{x} \quad (4)$$

As mentioned above, the exact solution for *e* in (3) is impossible because it requires A^{-1}. However, if an approximation $M \approx A$ is available, the error *e* can be approximated in (3) by

$$\overline{e} = M^{-1}r \quad (5)$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Finally, the approximation $\tilde{x}$ is updated by

$$\tilde{x} \leftarrow \tilde{x} + \bar{e} \quad (6)$$

It is (4)-(6) that form the foundation of the iterative solution method. A matrix solver using the iterative solution method is called an *iterative matrix solver*. The method starts with an initial guess $\tilde{x} = x_0$ and repeats (4)-(6) until the approximation $\tilde{x}$ to x is within tolerance, or the number of iterations exceeds a given number. In the former case, it is said the solution converges; while in the latter, it doesn't.

The residual r is used for measuring the closeness of $\tilde{x}$ to x. Because A and b in (1) can be scaled by the same factor without altering x, so does the residual r in (4). It typically makes more sense to replace $\|r\|$ as the stopping criterion with the relative residual:

$$res = \frac{\|r\|}{\|b\|} \quad (7)$$

where $\|\ \|$ stands for vector norm.

M in (5) is called a preconditioner of A. A good preconditioner greatly reduces the number of iterations. The following makes M a good preconditioner: M is a good approximation to A in some sense and M^{-1} is computationally cheap.

Some of the classic iterative matrix methods include

- The Jacobi method where M is the diagonal of A.
- The Gauss-Seidel method where M is the lower triangular or upper triangular matrix of A.
- The successive over-relaxation method (SOR) where M is a weighted combination of the lower triangular and upper triangular matrix of A.

Refer to the following reference for the details of iterative solvers:

Henk A. van der Vorst, *"Iterative Krylov Methods for Large Linear System"* Cambridge University Press, 2003.

3D Transient Solver

In the **3D transient (time domain)**, the default solver uses the $\vec{T} - \Omega$ (Omega) formulation. The default order of the magnetic scalar potential is 2 and the default order of the electric vector potential is 1. Motion (translational or cylindrical/non-cylindrical rotation) is allowed, excitations – currents and/or voltages – can assume arbitrary shapes as functions of time, nonlinear BH material dependencies are also allowed. The support of voltage excitations for the windings has as consequence the fact that the winding currents are unknown and thus the formulation has to be modified slightly to allow the Maxwell solver to account for source fields due to unknown currents in voltage-driven solid conductors (where eddy effects are evaluated) and in voltage-driven stranded conductors (where the eddy effects, such as skin and proximity effects, are ignored). Also for a simpler formulation of problems where motion is involved, Maxwell uses a

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

particular convention and uses the fixed coordinate system for the Maxwell's equations in the moving and the stationary part of the model. Thus, the motion term is completely eliminated for the translational type of motion while for the rotational type of motion a simpler formulation is obtained by using a cylindrical coordinate system with the z axis aligned with the actual rotation axis.

The formulation used by the Maxwell transient module supports Independent-Dependent boundary conditions and motion-induced eddy currents everywhere in the model, in the stationary as well as in the moving parts of the model. Mechanical equations attached to the rigid-body moving parts allows a complex formulation with the electric circuits being strongly coupled with the finite element part and also coupled with the mechanical elements whenever transient mechanical effects are included by users in the solution. In this case the electromagnetic force / torque is calculated using the virtual work approach. For problems involving rotational type of motion a "sliding band" type of approach is followed and thus no remeshing is done during the simulation. For translational type of motion, the mesh in the band object (surrounding the part in motion) needs to be recreated at each timestep with a degree of refinement which is dependent upon the mesh size in the moving object. In this later case, the mesh in both stationary and moving objects remains unchanged as initially created by the user. For the transient type of electromagnetic field analysis (with or without motion) the user is responsible for creating the mesh that is capable to "catch" the respective physics, such as skin and proximity effects, if any, are to be present in the resulting fields.

The following three Maxwell's equations are relevant for transient (low frequency) applications:

$$\nabla \times \boldsymbol{H} = \sigma(\boldsymbol{E})$$

$$\nabla \times \boldsymbol{E} = -\frac{\partial \boldsymbol{B}}{\partial t}$$

$$\nabla \cdot \boldsymbol{B} = 0$$

The following two equations directly result from the above equations:

$$\nabla \times \frac{1}{\sigma} \nabla \times \boldsymbol{H} + \frac{\partial \boldsymbol{B}}{\partial t} = 0$$

$$\nabla \cdot \boldsymbol{B} = 0$$

The final result is a formulation where vector fields are represented by first-order edge elements and scalar fields are represented by second order nodal unknowns.

Field equations are coupled with circuit equations for both solid and stranded conductors because, in the case of applied voltage supplies, the currents are unknown. For the case of voltage-driven solid conductors, the following equation is used to account for the ohmic drop across the i-th conductor loop:

$$V_{Ri} = \int\limits_{R_C(i)} \iint \boldsymbol{J}_{0i} \left(\boldsymbol{E} + \nu \times \boldsymbol{B} \right) dR$$

where J_{0i} represents the current density.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The current density J_{0i} corresponds to 1 A of net current in loop *i* and vanishes outside loop *i*.

The induced voltage can be derived from the following equation:

$$e_i = -\iiint_{R_{C(i)}} H_i \bullet B dR$$

where the integration is performed over the whole conductor region.

Stranded conductors are considered to be without induced eddy currents and, thus, are placed in the non-conducting region. This means that, for the purpose of calculating the ohmic voltage drop, the procedure for solid conductors cannot be used. Instead, a lumped parameter is used to represent the DC resistance of the winding. The induced voltage due to the total flux linkage is obtained in a similar way as for solid conductors. In both cases, it is also possible to add an external inductance and capacitance.

For the time discretization, a backward time-stepping scheme is used:

$$\left\{\frac{dx}{dt}\right\}^{t+\Delta t} = \frac{\{x^{t+\Delta t}\} - \{x\}^t}{\Delta t}$$

For the nonlinearities allowed in 3D transient applications, the classical Newton-Raphson algorithm is used.

The transient solver in Maxwell supports the **coil terminals** and **winding** definitions. Thus it is possible to specify the number of turns of coils in models which is necessary for the calculation of global quantities with high engineering value such as flux linkage and back emf of coils. Thus for the 3D transient solver a number of quantities are automatically calculated and displayed as 2D plots (functions of time): voltage (current), flux linkage, back emf. Other global quantities can be also calculated by the 3D transient solver and displayed as 2D plots such as power loss, core loss, stranded loss, electromechanical quantities such as force/torque, speed and displacement.

A few types of sources can be used in 3D transient applications. The *Coil Terminal* type is discussed here.

Coil Terminals

A coil terminal excitation is the preferred excitation for Maxwell transient applications. This type of excitation can be specified as functional (arbitrary functions of time) and is very flexible; six different combinations can be used: current, voltage, and external circuit, with either solid or stranded conductor options.

The **current, stranded conductor** option should be used when setting coils of constant cross-section made of strands of wire (with no eddy current effects to be taken into account in the coil). The current can be functional and the specific variation as a function of time can be specified by the user.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When defining the function, already defined primitives such as `sin()`, `cos()`, `exp()`, *etc.* can be used. For arbitrary variations, a piecewise linear capability can also be used by defining the corresponding dataset. You should also specify the number of winding turns.

> **Note:** When using the stranded option, coils and, more generally, each individual current path must have a constant cross-section. However, coils and conductors that may have different individual cross-sections can be connected in windings.

The **current, solid conductor** option does not carry the restriction of constant a cross-section. The current path can have varying cross-section, and can also split as long as in the end all parallel branches merge into a unique current path. You can also define a functional excitation and apply it to the winding.

The **voltage, strand** option is similar to the corresponding current setting, with the obvious difference being that now the voltage is known, while the current remains unknown. In many situations when the voltage setup is used, other terminal characteristics that influence the current calculation are also specified: for instance, series resistance, inductance, and capacitance. No eddy current effects are taken into account, so a series resistance (DC resistance) must be specified. For this type of source, the cross-section of each conductor needs to be constant, but the cross-section can differ from one coil to another, and it is possible to construct windings using coils with different cross-sections. After the solver calculates the total current, the current is uniformly distributed on the cross-section, as it always is with a stranded option.

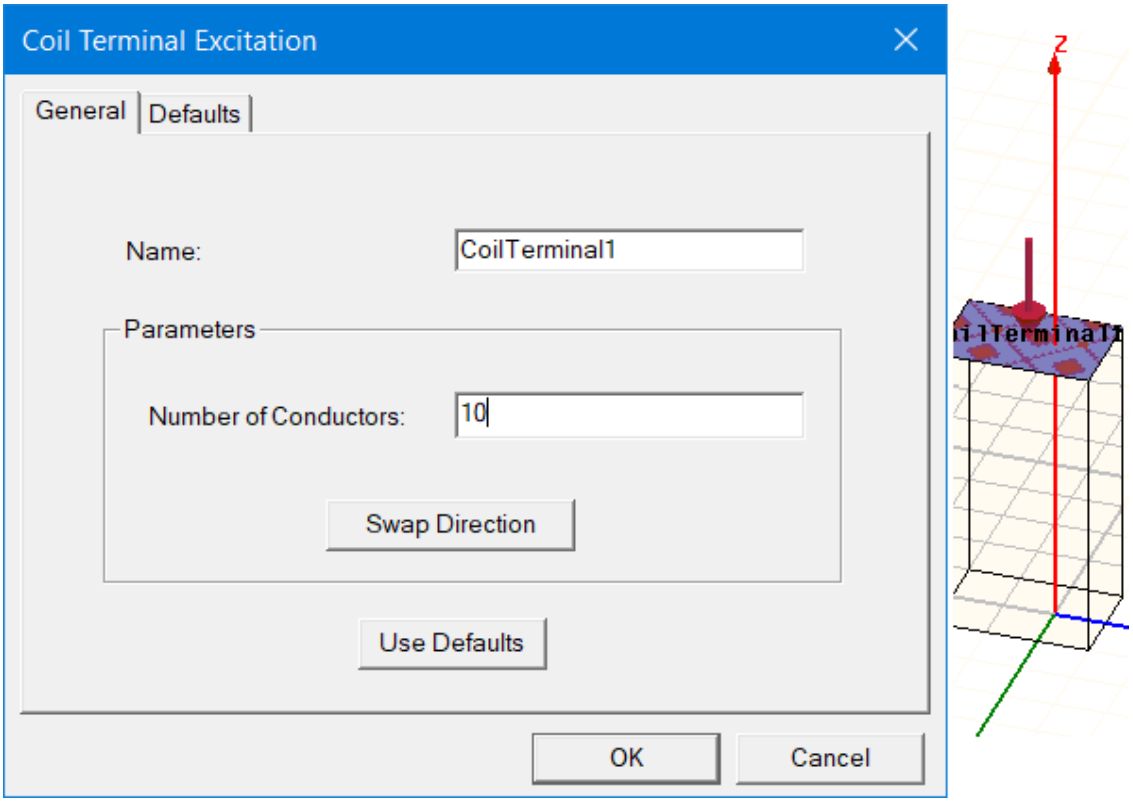
The **voltage, solid** option is chosen when solid conductors with eddy effects are part of the winding. In this situation, eddy current effects are taken into account. If required by the application, you should also include other characteristics of the source, such as series resistance, inductance, and capacitance.

The **external strand** and **external solid** settings are used when the circuits attached to the windings have an increased degree of complexity. In the case of **external sources**, solid or stranded options are used, depending on whether or not eddy effects are to be taken into account.

There are normally two phases in the process of defining windings.

1. First, define the terminal(s) by selecting the planar surface(s) as appropriate for the model, and then assign the coil terminal excitation by specifying the orientation of the current with respect to the terminal (current in or current out, graphically represented by an arrow). Specify the *actual* number of conductors (turns) intersecting *the chosen* planar surface (coil terminal), regardless of the symmetry (if any) of the problem.

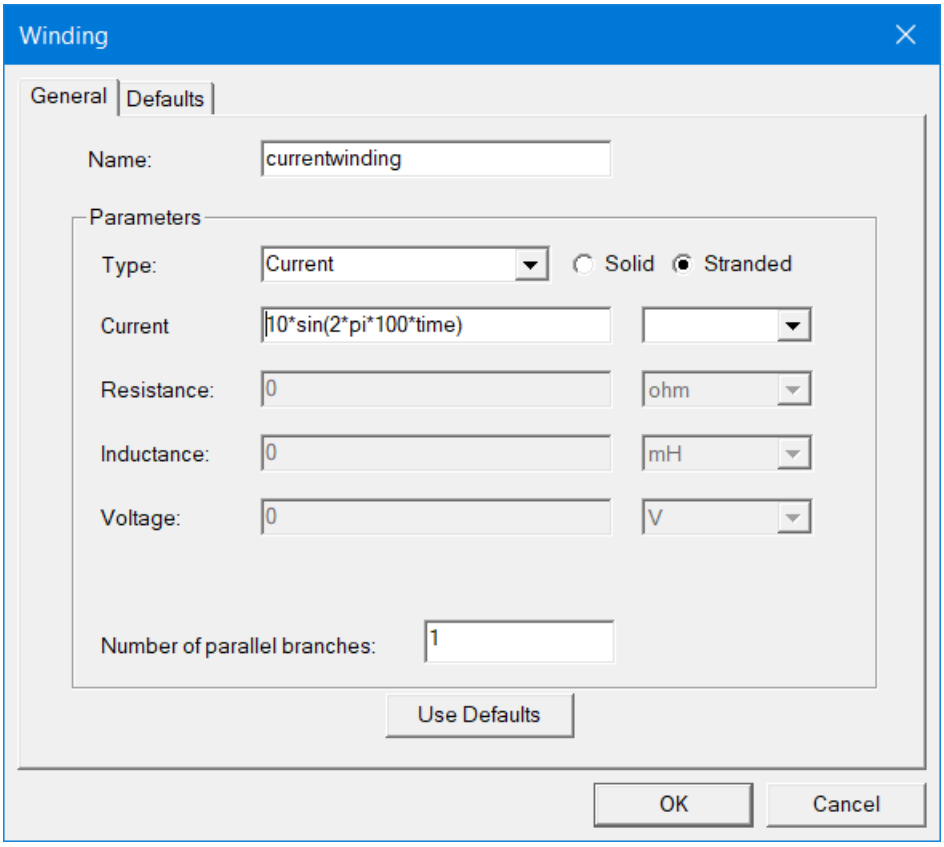
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Right-click in the 3D Modeler window, add a new winding from the excitation menu (no need to select any geometry), and specify the remaining details of the setup (the screen shot below shows a sinusoidal current with 10 A amplitude and 100 Hz frequency being used in the setup.

> **Note:** If external circuits are used in the excitation setup, they need to be created using the Maxwell Circuit Editor before they can be assigned to windings.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Right-click the winding in the property window, and specify the coil terminal(s) that belong to the winding (unassigned coil terminals are listed and can be selected from the respective window).

Solution Process

The solution process for 3D transient applications poses several challenges not apparent for other solver types. For instance, the behavior of fields is more complex than it is for static or steady-state applications, and a special finite element mesh structure is required in order for the software to accurately represent the physics.

Inductance Computation for 2D and 3D Transient Solutions

Because both 2D and 3D transient solutions are nonlinear, for inductance computation, Maxwell provides two options: *apparent inductance* and *incremental inductance*. After the FEA normal solution is completed at each timestep, the permeability associated with *apparent inductance*, or the differential permeability associated with *incremental inductance*, of each element, is frozen – which is equivalent to freezing the coefficient matrix (the left side of the equation to be solved). In order to obtain winding self-inductance and mutual inductance, the solver sets an excitation current of 1 A in the first winding, while setting all other winding currents to zero (permanent

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

magnet effects are excluded). This excitation assignment corresponds to one source vector on the right-hand side of the equation to be solved. As a result, the calculated flux linkage provides the self-inductance for the first winding with 1 A excitation current, and the calculated flux linkages represent mutual inductance for all other windings with zero current. Next, the solver excites the second winding with a current of 1 A, while setting all other winding currents to zero (permanent magnet effects are excluded). This excitation assignment corresponds to another source vector on the right-hand side of the equation to be solved. The solver continues this process until all windings have been assigned 1 A current in turn.

Flux Linkage

For the 3D transient solver using the **T-Omega** formulation, the turn density vector is defined as

$$\boldsymbol{J_k} = \frac{N_k}{S_k}\boldsymbol{t}$$

where

N_k is the number of turns;

S_k is the cross-sectional area of winding k; and

$\boldsymbol{t}$ is a unit vector tangent to the direction of winding wires.

In practice, this turn density vector is derived from the introduced basis function of the source field, $\boldsymbol{H_k}$, as

$$\boldsymbol{J_k} = \nabla \times \boldsymbol{H_k}$$

where $\boldsymbol{H_k}$ is the field corresponding to a 1 ampere current in winding k and is obtained by solving a conduction problem:

$$\nabla \times \boldsymbol{H_k} = J \qquad \nabla \times \boldsymbol{E} = 0 \qquad J = \sigma \boldsymbol{E}$$

The turn density vector $\boldsymbol{J_k}$ corresponds to 1 ampere of net current in the winding k, and vanishes outside winding k. Thus, the induced voltage due to the total time derivative of flux linkage in winding k can be obtained by projecting the induced electric field $-dA/dt$ onto the turn density vector $\boldsymbol{J_k}$ and integrating over the region of winding k:

$$e_k = -\iiint_{R_k} \boldsymbol{J_k} \bullet \frac{d\boldsymbol{A}}{dt} dR = -\iiint_{R_k} \nabla \times \boldsymbol{H_k} \bullet \frac{d\boldsymbol{A}}{dt} dR$$

$$= -\frac{d}{dt} \iiint_{R_k} \boldsymbol{H_k} \bullet \nabla \times \boldsymbol{A} dR - \oint_{\Gamma} \boldsymbol{H_k} \times \boldsymbol{A} \bullet \boldsymbol{n} d\Gamma$$

where Γ is the boundary of a simply connected region, which is composed of the winding region $\boldsymbol{R_k}$ and the cutting region $\boldsymbol{R_\Sigma}$ that is introduced to fill the holes of the winding to avoid multi-valued problem of scalar potential. In such a case, because $\boldsymbol{H_k}$ has zero tangential component on boundary Γ, the equation is

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\oint\!\!\!\oint_{\Gamma} \boldsymbol{H_k} \times \boldsymbol{A} \bullet \boldsymbol{n} d\Gamma = 0$$

Then the induced voltage in winding *k* can be derived in terms of the ***H*** field as

$$e_k = -\frac{d}{dt} \iiint_{R_k} \boldsymbol{H_k} \bullet \boldsymbol{B} dR$$

and therefore, the flux linkage in winding *k* be derived by

$$\lambda_k = \iiint_{R_k} \boldsymbol{H_k} \bullet \boldsymbol{B} dR$$

3D Transient A-Phi Formulation

For the Maxwell 3D A-Phi transient solver, solid and stranded conductors, and flux linkage, are defined as described below.

- **Solid Conductors**

  The Maxwell 3D A-Phi transient solver is formulated using the magnetic vector potential *A* and electric scalar potential Φ. The relations between the potentials and the magnetic field *B* and electric field *E* are

  $$B = \nabla \times A$$

  $$E = -\frac{dA}{dt} - \nabla\Phi$$

  Using the generalized Ampere's law, the current continuity equation, and the constitutive relations,

  $$\nabla \times \mathrm{H} = J$$

  $$\nabla \cdot J = 0$$

  $$J = \sigma E + \frac{dD}{dt}$$

  $$B = \mu H$$

  $$D = \varepsilon E$$

  the time-dependent magnetic equation is expressed as

  $$\nabla \times \nu \nabla \times A = -\sigma \frac{dA}{dt} - \sigma\nabla\Phi - \frac{d}{dt}\left(\varepsilon \frac{dA}{dt}\right) - \frac{d}{dt}\left(\varepsilon\nabla\Phi\right) + \nabla \times H_c$$

  $$\nabla \cdot \left(-\sigma \frac{dA}{dt} - \sigma\nabla\Phi - \frac{d}{dt}\left(\varepsilon \frac{dA}{dt}\right) - \frac{d}{dt}\left(\varepsilon\nabla\Phi\right)\right) = 0$$

  where

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- H is the magnetic field intensity
- J is the total current density including source, eddy, and displacement current contributions
- σ is the conductivity
- μ is the permeability
- ε is the permittivity
- H_c is the coercivity of the permanent magnet
- $\nu = \mu^{-1}$ is the reluctivity.

Maxwell 3D is used mostly for low-frequency simulations, so the radiation term $\frac{d}{dt}\left(\varepsilon\frac{dA}{dt}\right)$ in the above equations is ignored in the A-Phi Solver. The first order edge elements for the magnetic vector potential and the second order nodal elements for the electric potential are used. The *tree-cotree* method is used to ensure the uniqueness of the magnetic vector potential *A*. The Degrees of Freedom (DOFs) of edge elements on the spanning tree of the finite element mesh are set to zero. The formulation above is used in the entire problem domain except in the stranded conductors.

- **Stranded Conductors**

  For stranded conductors, the current density is assumed to be constant over the conductor domain, where the eddy and displacement effects are ignored. Therefore, only Ampere's law is considered in stranded conductors:

  $$\nabla \times \nu\nabla \times \boldsymbol{A} = \boldsymbol{J}$$

  The solver approximates the constant current density distribution over each conductor for a unit current before the transient solution starts. The approximated unit current density is

  $$\hat{\mathbf{J}} \approx \hat{t}\frac{\mathbf{N}}{S}$$

  where

  $\hat{t}$ is the current (winding) direction

  I is the total terminal current

  S is the area of the coil terminal into which current is entering

  N is the number of turns

  J is then written as $J = \hat{J}I$.

  For winding voltage excitation, the formulation is

  $$\nabla \times \nu\nabla \times \mathbf{A} - \hat{\mathbf{J}}\mathbf{I} = 0$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\iiint_{\Omega_k} \hat{\mathbf{J}} \cdot \frac{\mathrm{dA}}{\mathrm{dt}} d\Omega + \left(R + R_{ext} \right) I + \mathrm{L_{ext}} \frac{\mathrm{dI}}{\mathrm{dt}} = V$$

where R is the DC resistance of the winding, and R_{ext} and L_{ext} are, respectively, the external resistance and inductance.

- **Flux Linkage**

  Flux linkage calculation for windings is relatively easy compared to T-Omega formulation. It is calculated using the equation

  $$\lambda_k = \iiint_{\Omega_k} J_k \cdot A \, d\Omega$$

  where J_k is the current density in the winding under 1 A excitation, A is the total magnetic vector potential in the conductor, and Ω_k is the volume of the winding.

Complex Field Behavior for a Transient Solution

For 3D transient applications, the behavior of the fields is more complex than it is for static or steady-state applications. A diffusion of the magnetic field into the materials occurs in 3D transient situations. The distribution of the magnetic field inside objects typically has a number of spatial harmonics, which usually means the timestep used in the analysis should be less (sometimes much less) than the magnetic diffusion time constant. These time constants depend upon the geometry of objects and also upon their respective material properties. For example, for a cylinder, the magnetic diffusion time constant of the fundamental spatial harmonic is given by the following equation:

$$\tau = \frac{\mu \sigma a^2}{2.4048^2}$$

where μ is the permeability of the cylinder, σ is the conductivity, and *a* is the radius.

Special Mesh Structure for a Transient Solution

Because eddy currents are usually considered in conductive objects, a special finite element mesh structure is required in order to accurately capture the physics. In general, a careful planning of the (manual) meshing process is required in order to achieve an accurate solution with the available hardware resources.

Overcoming Challenges for a Transient Solution

There are a number of applications where the combined effects of magnetic diffusion and strong eddy effects represent a significant challenge. Add the effect of nonlinearities that may also be present, and you have the full picture of a difficult to solve application.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following suggestions can help you overcome these challenges presented by some 3D transient applications:

- Use symmetry whenever the problem allows it.
- Set the eddy effects only on objects where it really counts.
- In general, all unnecessary details, such as details of a geometric nature, should be eliminated, particularly in cases where a large problem is the expected task.

Boundary Conditions for a Transient Solution

The behavior of the **H**-field at the edges of the problem space is controlled through boundary conditions. The following boundary conditions can be assigned for a 3D transient model:

- Natural boundaries (T-Omega and A-Phi) are assigned to the surface between objects.
- Neumann boundary conditions (homogeneous) (T-Omega and A-Phi) are assigned to the outside faces of the problem region.
- Odd Symmetry (Magnetic Flux Tangential)
- Even Symmetry (Magnetic Flux Normal)
- Matching
- Insulating

Symmetry for a Transient Solution

Symmetry boundary conditions take advantage of the symmetry of a problem. When deciding upon the symmetry, you must consider the geometry, as well as material properties and source distribution in space. Using symmetry boundary conditions when possible is very useful, particularly in 3D transient applications, which typically require large computing resources. Two types of symmetry are available:

- Odd Symmetry (Magnetic Flux Tangential)
- Even Symmetry (Magnetic Flux Normal)

These boundary conditions can be assigned to the faces of the problem region.

Odd Symmetry (Magnetic Flux Tangential)

For a transient solution, use an odd symmetry boundary to model a plane of symmetry in a symmetric problem in which current on one side of the plane flows in the opposite direction than current on the other side of the plane.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

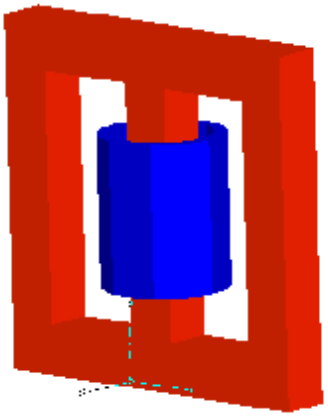
> **Note:** If the odd symmetry plane cuts through an object with eddy currents that tend to flow normal to the magnetic flux tangent boundary, then the odd symmetry condition must be explicitly applied to the model. In this situation, you cannot rely on the default boundary condition.

Even Symmetry (Magnetic Flux Normal)

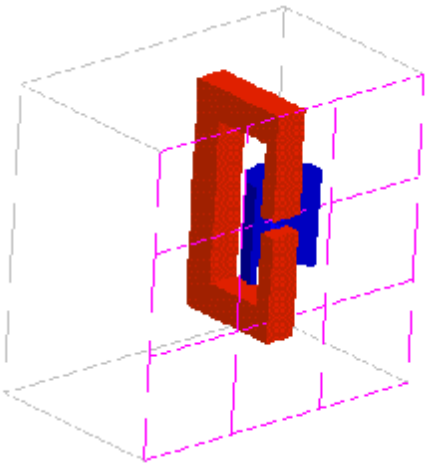
For a transient solution, use an even boundary to model a plane of symmetry in a symmetric problem in which the direction of current flow on both sides of the plane is the same. The currents on both sides of the plane are also assumed to have the same variation in time.

Examples of symmetry are shown in the following four pictures, where the inductor model presented has three-fold symmetry:

Full Model

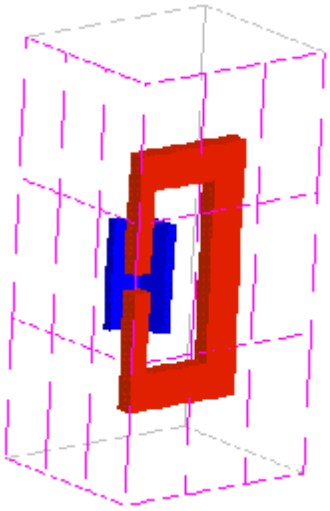


Half Model, Magnetic flux tangent Boundary condition on the symmetry plane

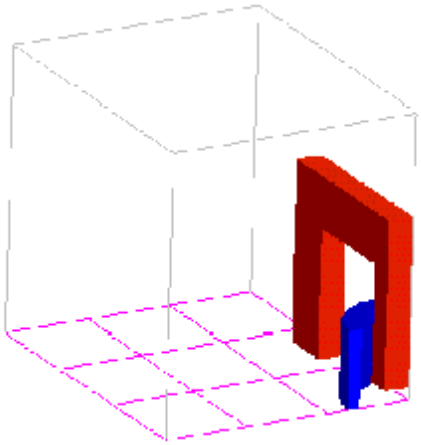


Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Quarter Model, 2 magnetic flux tangent boundary conditions



1/8 Model, 2 magnetic flux tangent boundary conditions (as in 1/4 model), one magnetic flux normal boundary condition (bottom plane)



Matching Boundary for a Transient Solution

Matching boundaries allow you to model planes of periodicity where the **H** field on one surface exactly matches the **H** field on another by forcing the magnetic field at each location on one surface (the "dependent" boundary) to match the magnetic field at the corresponding location on the other surface (the "independent" boundary). Matching boundaries are used in periodic structures and decrease the resources used in the computational process.

For matching boundaries, you need to set up both an independent and a dependent boundary. Unlike symmetry boundaries on independent and dependent boundaries, the **H** field does not need to be either tangential or normal to these boundaries. However, the **H** field on the two boundaries must have the same magnitude and direction (or the same magnitude and opposite

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

direction) at each timestep. The variation in time of the fields at corresponding locations is the same on matching (independent and dependent) boundaries.

Insulating Boundary for a Transient Solution

An insulating boundary prevents current from flowing across a surface - for instance, the interface between two adjacent conductors. Use an insulating boundary to model very thin layers of insulating material between conductors. Modeling thin insulating sheets saves you from the complications associated with the geometry of the insulation and also speeds up the solution process.

Meshing and Band Setting Recommendations for 3D Transient Applications With Motion

Maxwell supports three types of 3D transient problems with motion:

- Translational motion (motion along a user specified, linear direction).
- Rotational motion (non-cylindrical such as the pivoting rotation around an axis encountered in the armature of a relay).
- Rotational motion (cylindrical, such as the type of rotation encountered in an electric machine type of application).

Regardless of the particular type of motion involved, all types of motion applications require a band object that must contain the moving part(s). If there are multiple moving objects, all of the moving parts must be included in one all-inclusive object — this is because they all must be moving as one rigid body, with a single force acting on the assembly. Also, regardless of the type of motion, the user must create a mesh density capable of capturing the physical effects characteristic for the specific application, such as field gradients, skin and proximity effects, *etc*.

In terms of meshing, the main characteristic of the first two types of motion is that, during the analysis, a new mesh is created at each timestep as the motion occurs. This remeshed region is always inside the band object between the moving part(s) and the stationary part(s). The mesh corresponding to the moving part(s) and the stationary part(s) is kept fixed during the analysis. The mesh density inside the band object is always controlled during the solution process: the edges of any element created inside the band object during remeshing will never be larger than the average element edge on the entire surface of the band object where it is in contact with both the moving and stationary parts. Thus, both the mesh density created for the moving part (in the area where it touches the band) and the mesh density in the stationary part (in the area where it touches the band) are used to actively control the mesh density in the remeshed part throughout the entire solution process. This ensures good solution quality for any aspects conditioned by the quality of the mesh.

In maximizing the quality of the solution for 3D transient applications with motion, a number of observations and recommendations are available regarding the band object.

For applications with translational and non-cylindrical rotation types of motion:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The band object can touch the symmetry plane if any exists.
- The moving object cannot touch a stationary object during motion (the gap between the moving object and the band can never become zero during analysis). The only exception to this rule is for the particular case when both the band and moving object surfaces touch a symmetry plane (if any exists in the respective setup) and the moving object slides along the symmetry plane (in this case the symmetry plane, respective band, and moving object surfaces coincide);
- The band cannot have true surface faces; all faces must be segmented (for example created with the regular polyhedron primitive).
- The band object cannot separate the stationary part into unconnected subregions.

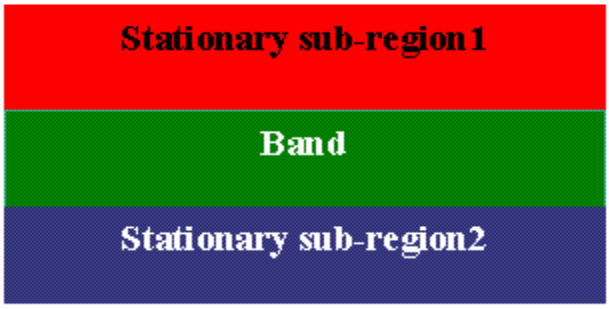


Fig.1 Band cannot separate the stationary part into unconnected regions

- Hollow objects cannot be used for band objects.
- Subtractions cannot be performed within the band object.

For the cylindrical type of rotational motion applications:

For this type of application, the band object must by somewhat larger than the rotating part(s) in ALL directions with only one exception, at the boundaries, which can just be touched. Thus the band object must effectively enclose the moving part, not just by touching it, in the radial and axial directions; so users have to leave some "room" between band and surrounding objects. The only exceptions allowed to the requirement mentioned above are: symmetry plane (if any exists), M-S or other boundaries as applicable for the application, can be touched, that is, at those planes it is allowed for the band surface(s) to coincide with the respective planes.

Additionally, the following observations are also applicable:

- For the band object, always use a faceted (regular polyhedron) type of cylindrical object or a wedge object if symmetry is used. The angular aperture of each facet depends on the problem; however, an opening of 2-3 degrees per facet is usually sufficient.
- Hollow cylinders cannot be used for band objects.
- Subtractions cannot be performed within the band object.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The band object can separate the stationary part into unconnected sub-regions (a rotor sandwiched between two stators is allowed).
- Non-cylindrical band objects are allowed.

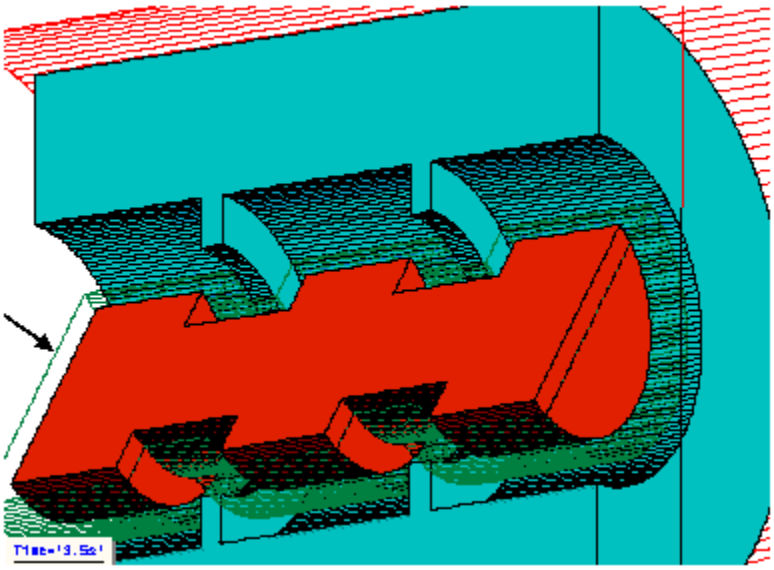


Fig. 2 Rotor and stator assembly with band object in between

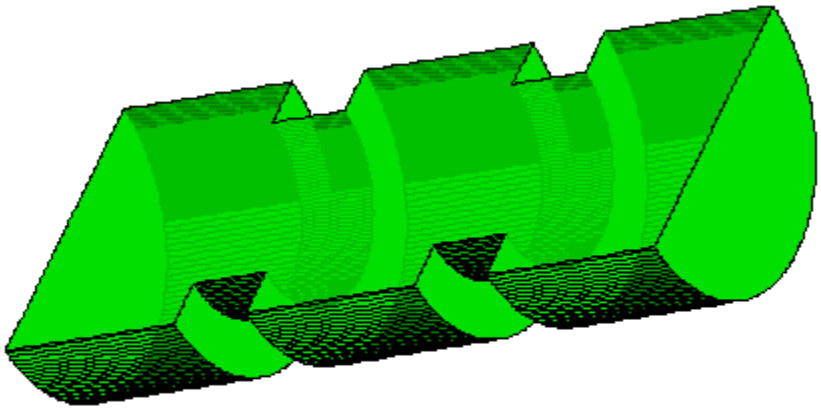
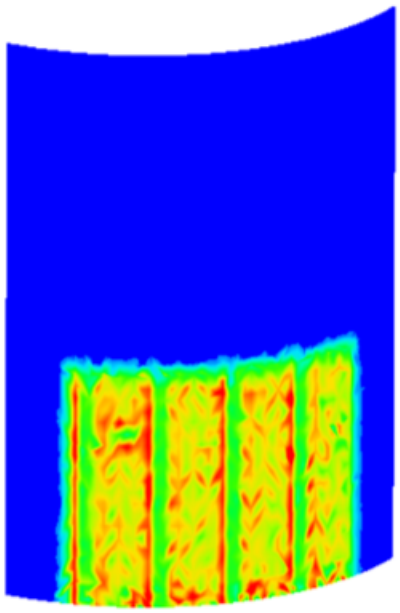


Fig. 3 Band object

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Improving Reliability of Results for Air-Gap Modeling of Electric Motors (3D)

When there is motion in the simulation, the mesh on the interface of the moving and stationary domains can become non-conformal (*i.e.*, the nodes at the two sides of the interface are not in the same positions). Transient motion simulation in Maxwell 3D requires mapping of fields across the interface between moving and stationary domains. Depending on the method used for this mapping, hot spots of the field on the interface can occur due to numerical error as shown in the figure below.



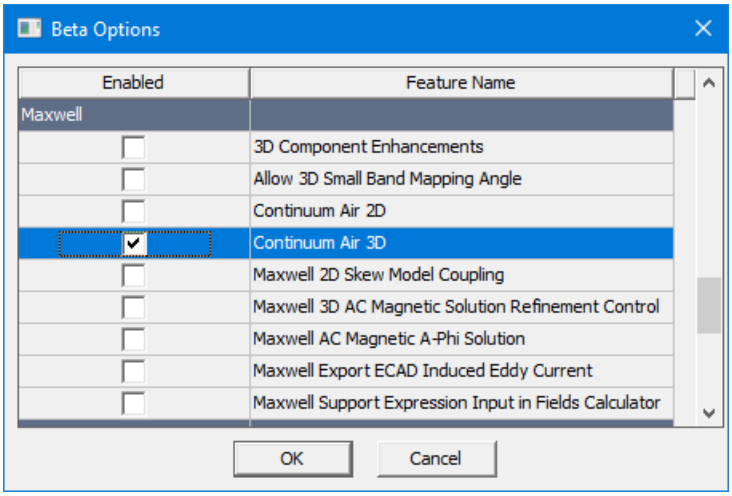
The **Continuum Air 3D** beta feature delivers improved smoothness, accuracy, robustness, and efficiency for 3D transient electric machine simulation and meshing. Continuum Air technology dramatically improves the interface mapping, reducing hot spots, numerical noise, and other simulation artifacts. As a result, solution quantities will be noticeably smoother and more accurate, especially sensitive quantities such as core loss, cogging torque, and forces for NVH.

In addition, this beta feature enables multi-domain meshing, a new meshing approach that provides improved mesh efficiency, speed, and robustness. Multi-domain meshing is automatically activated for most motor types and settings, and its use is documented in the Profile tab of the Solutions window.

To turn on this feature:

1. Open the **Tools** > **Options** > **General Options** window.
2. Click the **Beta Options** button.
3. Scroll down to the Maxwell options, and select **Continuum Air 3D**.

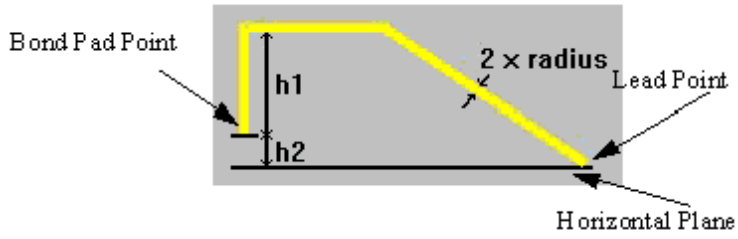
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Click **OK** in the Beta Options window and in the General Options window.
5. Click **Yes** to restart Ansys Electronics Desktop to activate the feature.

Bondwires

A bondwire is a thin metal wire that connects a metal signal trace with a chip. You can choose to draw a standard JEDEC four-point bondwire, as shown below:



where

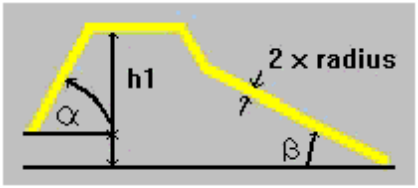
h1 = the height between the bond pad point and the top of the loop.

h2 = the height between the lead point and the bond pad point.

radius = half the diameter, or thickness of the wire.

You can draw a standard JEDEC five-point bondwire, as shown below:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



where

α = the angle between the horizontal plane and the wire at the bond pad point.

β = the angle between the horizontal plane and the wire at the lead point.

When drawing the bondwire, first select the bond pad point, a point in 3D space that defines the bond pad position in a horizontal plane. Then select the lead point, which indicates the distance the wire covers in the horizontal plane. Maxwell uses the distance between the bond pad and lead points to calculate the height between the bond pad and the lead point, or *h2*, a value that you can modify in the **Bondwires** dialog box.

Partial Simulation for Full Rotational Machine Models

A rotational machine can be simulated by using either a 360 deg full model, or a partial model with matching boundary. Full model simulation has higher memory and time cost than partial model simulation; while partial model would require manual operations to cut the model and assign a matching boundary. The manual cut on the geometry could introduce geometry issues which lead to poor mesh quality near matching boundaries.

Partial simulation for full rotational machine models is designed to combine the benefits of full model simulation and partial model simulations. Without any manual cutting of the geometry, Maxwell can perform partial simulation for full rotational machines.

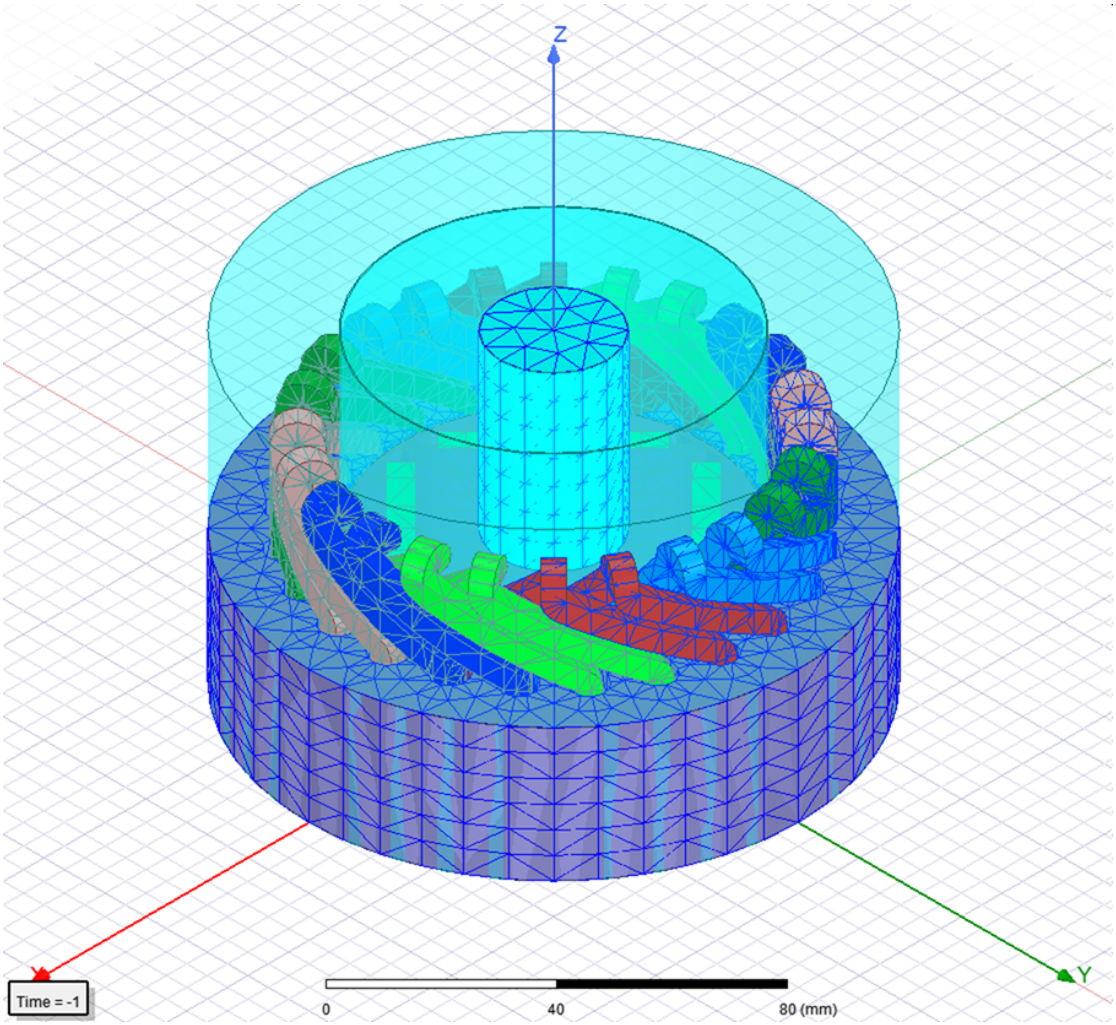
Background Information

Partial simulation for a full rotational machine model generates and solves a partial mesh on the full model. Instead of a manual cutting of the geometry, this feature allows the mesher to decide where to make the cut. The mesher will also automatically create virtual matching boundaries for the partial mesh.

The following example uses the **ipm_2.aedt** model in the *[install_dir]***\Examples\RMxprt\ipm** folder to generate a Maxwell 3D model with **Total circumferential fractions in created 360 deg geometry** on the **Symmetry Multiplier** tab set to *4*.

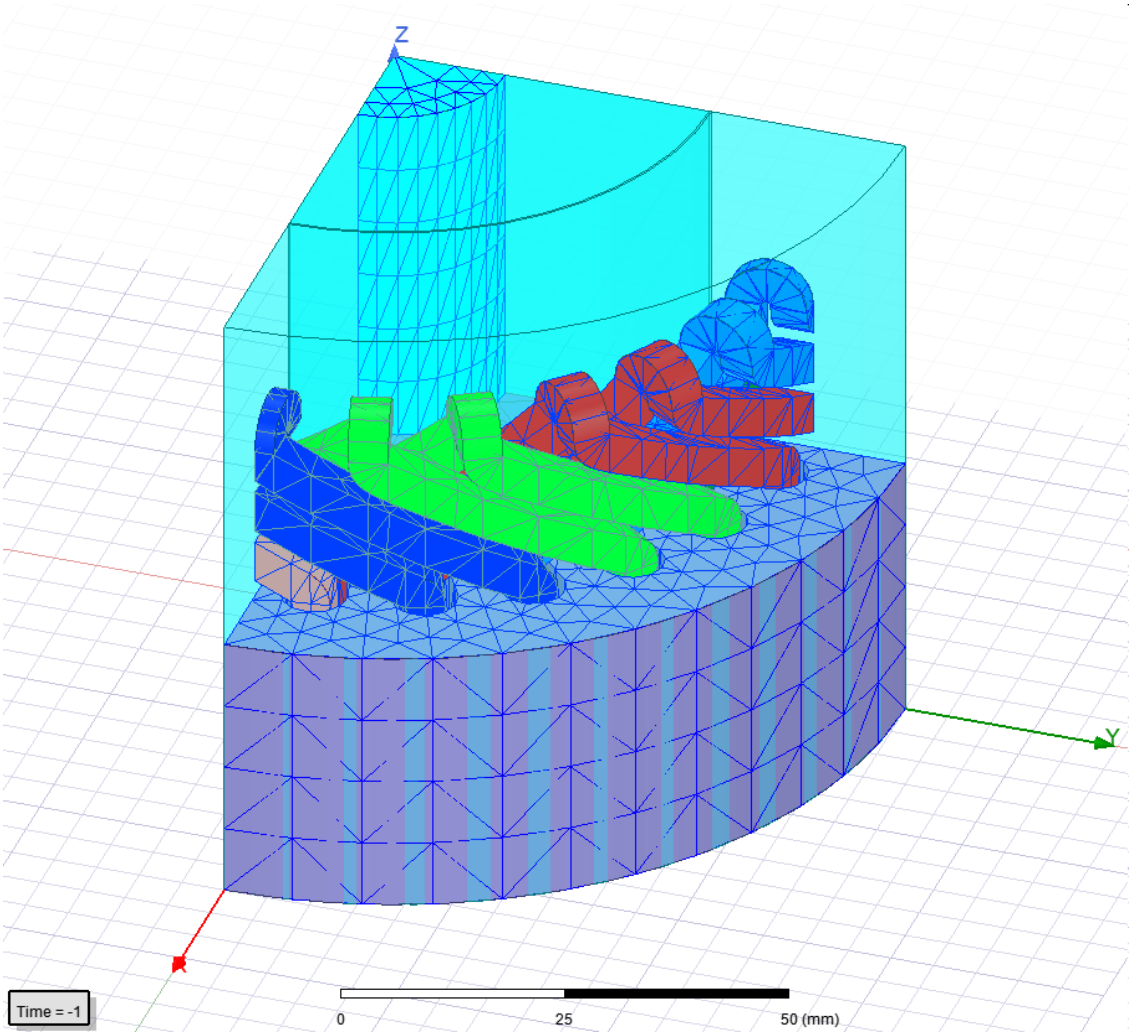
The figure below shows a full mesh on the full ipm_2 model with an element count of 95759:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



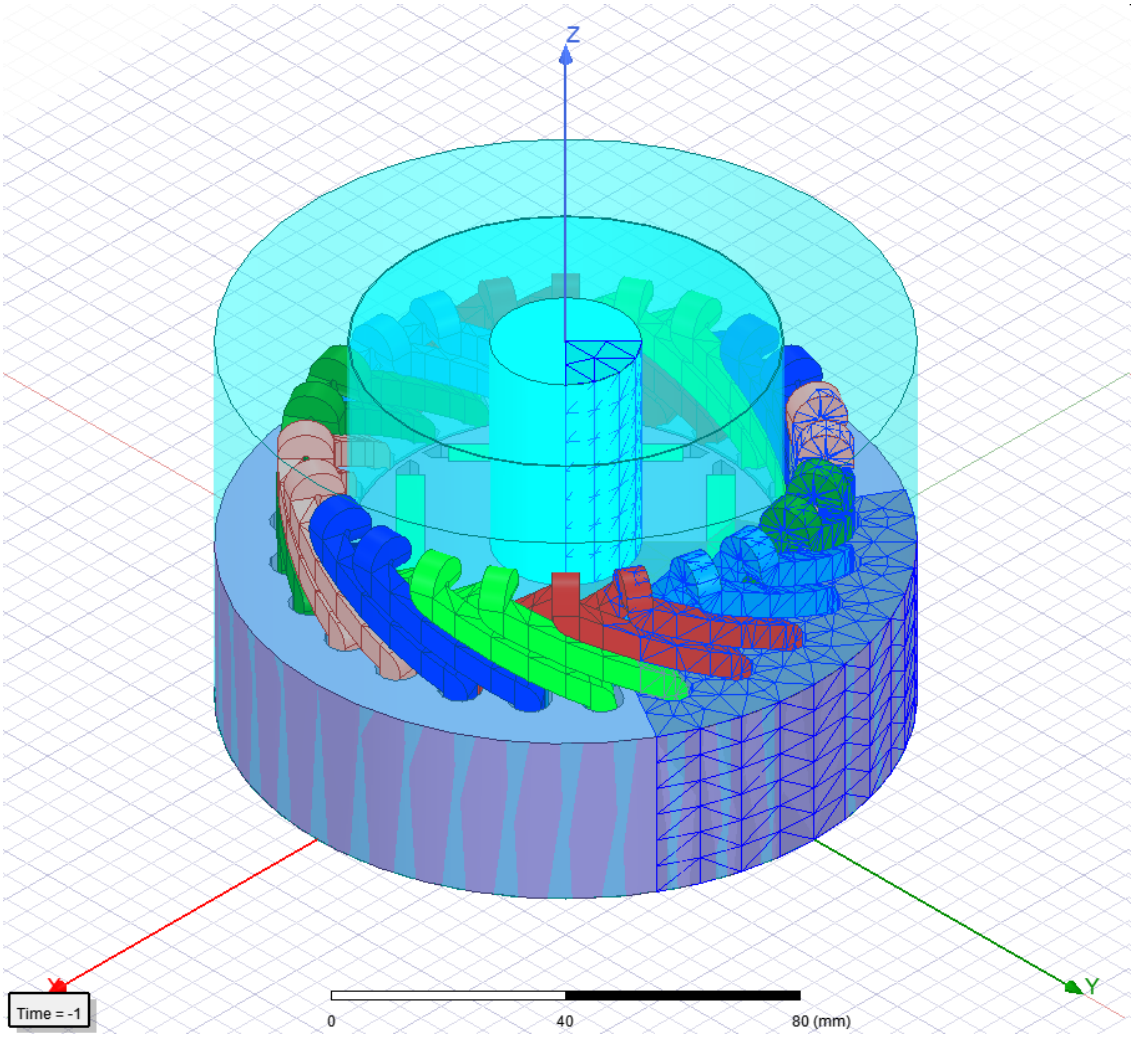
The following figure shows a partial mesh on a manually cut partial model for which the element count is 22194:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



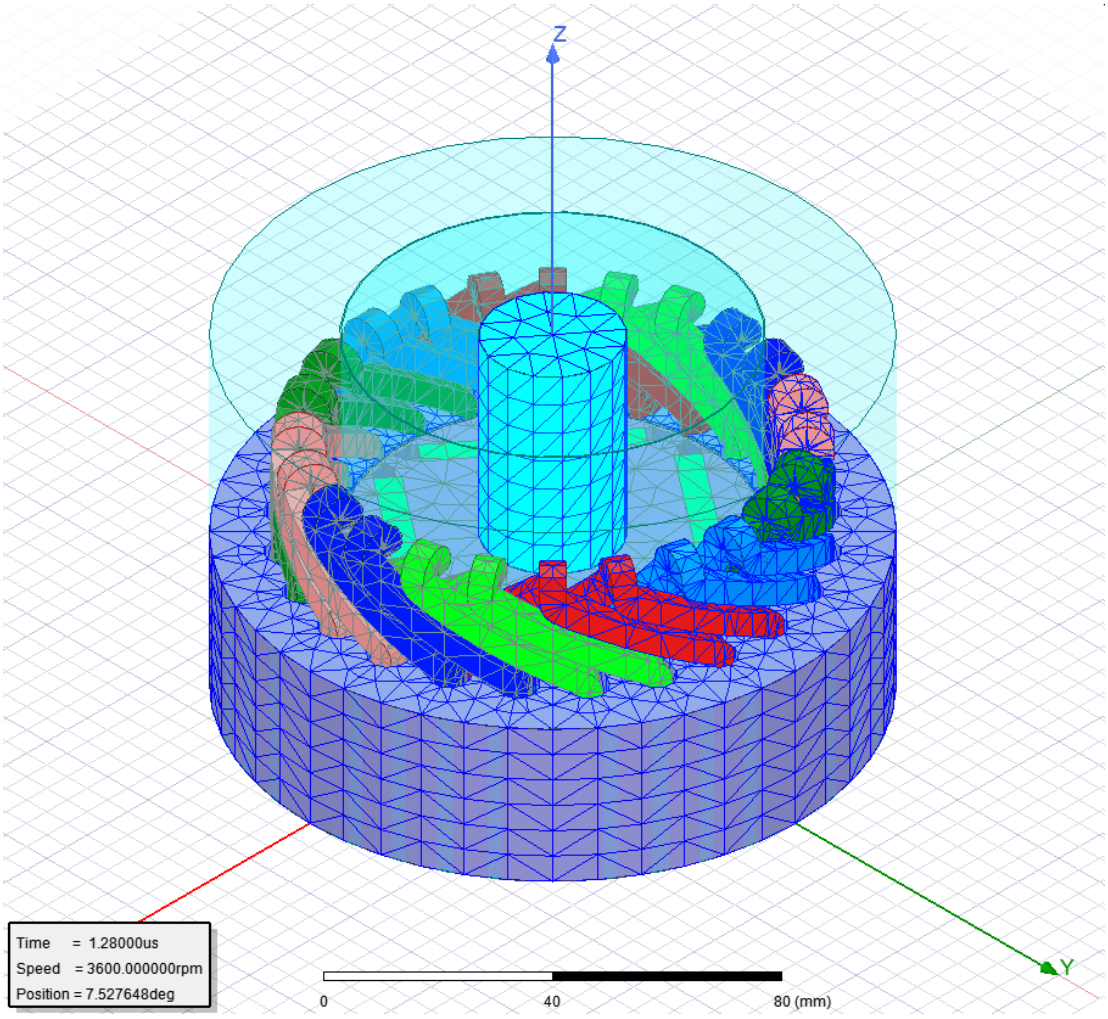
The following figure shows a behind-the-scene partial mesh on a full model for which the element count is 20298. Note that the element count for the partial mesh on a full model is comparable to partial mesh on a partial model, for which neither a manual model cut, nor matching boundary assignment is required.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

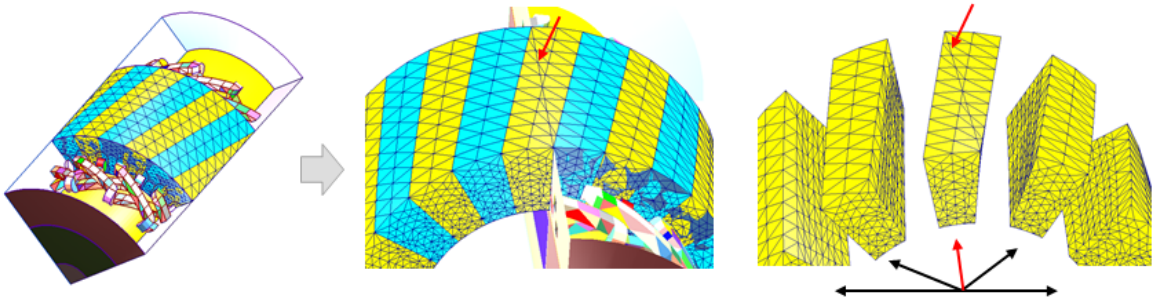


When plotting the partial mesh on the full model, although behind-the-scene mesh only covers part of the model, the mesh plot will be copied and cover the full 360 degree model. Thus, the following model will actually be displayed:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

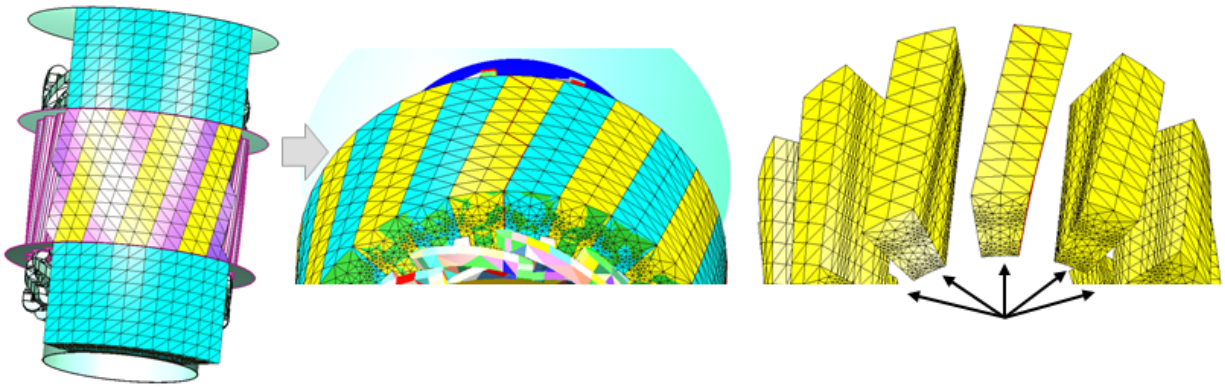


A model with a skewed stator gets even more benefits with partial clone mesh. As shown below, because of the manual cut on the geometry, the skewed model cannot get a perfectly repeatable clone mesh near the matching boundary with partial mesh for a partial model.



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

On the other hand, with non-planar virtual matching boundary in partial mesh for the full model, identical mesh can be generated for all repeatable geometry patterns.



Workflow for Partial Simulation for a Full Rotational Machine Model

1. After solving the RMxprt machine design, create a full rotational Maxwell 3D machine design with 360 deg geometry.
2. Select **Solve one circumferential fraction** in Symmetry Multiplier settings.
    a. Enter an integer value for the **Total circumferential fractions in created 360 deg geometry**. For example, if a full model can be cut into a 1/4 partial model and matching boundary can be assign at 0 deg and 90 deg of the model, the integer value entered should be *4*.
    b. Select either **Full** or **Half** for **Create geometry in axial direction** depending on whether the created model is the half model or the full model in axial direction.
    c. For a rotating machine, the simulation solution is periodic in the circumferential direction. Thus, select either Periodic or Half-periodic for **Field in one fraction** depending on whether the simulation solution in one fraction is of one period or half period physically.
    d. Click **OK** to close the dialog box.
3. Run simulation on the model.

Healing Models

The underlying solid modeling technology used by Ansys Electromagnetics products is provided by the Parasolid kernel geometric modeler. Users can directly create models using primitives and operations on primitives. In addition, users can import models saved in a variety of formats (Step, IGES, *etc.*) All the models are stored internally in Parasolid native format. When users import models into Ansys Electromagnetics products, translators are invoked that convert the models to this format. Parasolid regularly upgrades their solid modeling system. Third-party

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

vendors also create files in Parasolid format, but in some cases, the data in the files is not robust. For example, when translating from IGES, some accuracy is lost.

All of the above contribute to errors when a model is read. Not all third-party vendors write files to the supported formats with a high level of accuracy. Third-party vendors sometimes create invalid files – the files are either targeted at earlier versions of the kernel, or incorporate invalid elements in the file. Sometimes there are errors in reading a file generated by an earlier version with the current version.

The modeling kernel is a highly specialized software component underlying the 3D Modeler in AEDT. The kernel represents the fundamental geometry and implements the operations that create and modify it. Simple geometry and operations are likely to be identical in different kernels and will migrate cleanly. More complex models may have discrepancies due to differences in geometry representation, tolerances, algorithms, or other implementation details. These differences may be subtle, but they can lead to challenges in the migration process.

After migrating to Parasolid, some objects may have invalid geometry or topology, leading to validation or meshing failures. In this case, customers are encouraged to take advantage of the 3D Modeler's built-in healing commands. The Parasolid kernel has very strong healing algorithms and it is often possible to automatically repair such geometry. In the worst-case scenario where healing is not successful, it may be necessary to redraw the object in Ansys.

Error Types

When models are imported there are two types of errors – geometry errors and topology errors. Geometry errors are errors in definition of the underlying geometry while topology errors are errors in how the underlying components like faces, edges and vertices are connected. These have to be fixed before mesh analysis can be performed.

When models pass the initial validity checks, mesh generation could still fail. If bodies in the model overlap, mesh will not be invoked. If bodies are very close to each other, mesh might fail. Small features (small edges, small edges, sliver edges) might be present in the bodies which might cause mesh to fail.

The following are common errors present in models:

1. api_check_entity() errors – These are errors detected by the kernel and are geometry and topology errors.
2. non-manifold topology – These are non-manifold edges and vertices that are present in the model.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** One of the requirements for valid mesh generation is that the surface mesh must be ***manifold***, that is, a plot of all the surface triangles of a 3D solid object should not have any "holes." Such a mesh is often referred to as a ***watertight*** mesh. If there is a hole and you were to fill the object with water, the water would leak out of that hole – the mesh would not be ***watertight***. Zero-area holes (often called cracks) will also "leak." Having multiple triangles cover the same patch also results in a mesh that is not manifold. Refer to *Detecting Errors* for information on detecting and correcting such errors.

3. Body pair intersection – This detects if pairs of bodies intersect.
4. Small feature detection – small edge length, small face area and sliver face detection.
5. Misaligned entities detection – detects pairs of faces from bodies that can be aligned to remove interbody intersections. This improves the odds of mesh success.
6. Mesh failure error display – This is available for single body, body pairs and last simulation run (all bodies in model). Errors reported by the meshing module are reported to the user.

Errors of type 1, 2, and 3 must be resolved before the mesh can be applied to the model.

Detecting Errors

Errors in the model can be detected by the following methods:

1. Perform a validation check (**Maxwell 3D** > **Validation check**). This returns the following types of errors: api_check_entity() errors, non-manifold errors, and intersection errors. It works on all the bodies present in the model.
2. Run an object analysis (**Modeler** > **Model Analysis** > **Analyze Objects**) on a subset of bodies in the model. This returns api_check_entity() errors and non-manifold edge/vertex errors, and optionally, small edge, small face, and sliver face detection errors.
3. Run an interobject misalignment analysis (**Modeler** > **Model Analysis** > **Analyze Interobject Misalignment**). This yields face pairs from different bodies that are slightly misaligned with respect to each other. This misalignment causes the faces to intersect each other or to create small volumetric portion between the bodies that might cause mesh to fail. If these faces are aligned – that is, the faces are made to share the same surface definition – then the gap between the faces is eliminated, and there is a higher likelihood that the mesh will succeed.
4. Run a surface mesh analysis (**Modeler** > **Model Analysis** > **Analyze Surface Mesh**). For the selected bodies, a mesh is invoked on each individual body and optionally, on body pairs. Errors from running the mesh analysis are displayed, helping you focus on bodies/body pairs that fail meshing. If these are fixed, there is a higher probability that the mesh will succeed.
5. Mesh errors from the last simulation run are displayed.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Errors for 2, 3, 4 and 5 are displayed in the **Model Analysis** dialog box. Using the menu items for 2, 3 and 4 displays the dialog box. This dialog box can also be opened by clicking **Modeler** > **Model Analysis** > **Show Analysis Dialog**.

Healing and Meshing

This section lists problems that can prevent a 3D model from being meshed successfully. Subsequent sections will describe how these problems can be detected and addressed.

Kernel Errors

The underlying solid modeling technology used by Ansys Electromagnetics 3D Maxwell, HFSS, and Q3D products is provided by the Parasolid kernel. You can create models directly in the drawing environment of these Ansys Electromagnetics products using primitives, such as boxes and cylinders, and operations on primitives, such as Boolean operations. In addition, you can import models produced by other CAD tools in a variety of formats such as STEP, IGES, *etc*. In Ansys Electromagnetics 3D products, all models are stored internally in the kernel's native format. When you import models into Ansys Electromagnetics products, translators are invoked that convert the models to this format. Often, models that were created in other CAD tools were initially created for other purposes than electromagnetic analysis, for example, for mechanical design or just for display purposes. They may have imperfections that make them illegal to the kernel. Furthermore, there can be compatibility issues between different versions and even flavors of modeling tools. All this can lead to errors in imported 3D models.

If you use Ansys Electromagnetics products to create geometry models, and thereby avoid model import and translation, you are unlikely to encounter such problems.

Mixed Dimensionality

Even if a model is imported and translated without errors, there is a restriction to be aware of: The kernel cannot handle mixed-dimensionality models. One of the goals of Ansys Electromagnetics' use of the kernel modeling system is to create a valid volumetric mesh for simulation. Mixed-dimensionality models will not yield a valid volumetric mesh. Therefore, the Ansys Electromagnetics tools will not mesh objects with mixed dimensionality, so-called non-manifold objects. For instance, imagine a 3D object representing a curved metal plate with a small but finite thickness. If it reaches zero thickness somewhere while having non-zero thickness elsewhere, it has mixed dimensionality: 2D as well as 3D. You will get an error message saying that the object is non-manifold. Of course, 2D and 3D objects can co-exist in a model, but any one object cannot be both 2D and 3D.

Intersecting Objects

Another restriction is that Ansys Electromagnetics 3D tools do not allow partial intersections (also known as partial overlaps) between 3D objects. Each element of the mesh has to belong

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

unambiguously to one object. There is no problem if one object is enclosed completely inside a bigger object, but partial intersections lead to ambiguities. As long as there are partial object intersections, the mesh generator will not attempt to create a mesh. Instead, you will get an error message notifying you which objects are intersecting. You must remove the intersections before you can proceed. You can do this by changing the shapes of objects slightly or by subtracting one object from the other.

> **Warning:** If, as a result of a subtraction, the model has pairs of true surfaces that are coincident (that is, smooth curved surfaces that fit exactly one inside the other), you make it harder for the mesh generator to create a mesh. This is because the kernel creates segmentations on each of these surfaces, and these segmentations are not guaranteed to fit. In such a case, setting a small value for Surface Deviation under **Mesh** > **Assign Mesh Operation** > **Surface Approximation** increases your chance of success, but it is better to avoid such situations if you can.

Small Features and Misalignment

When there are no kernel errors in the model, no non-manifold objects, and no partial object intersections, the mesh generator can be invoked to create a valid mesh for the electromagnetic analysis. Even if the geometry is valid, mesh generation can still fail. Possible causes are the presence of very short edges, very small faces, long and thin sliver faces, and slight misalignments between faces that are supposed to be coincident.

Detecting and Addressing Model Problems to Improve Meshing

The following sections describe a systematic procedure to detect and address model problems that can interfere with the meshing process:

Technical Notes: *Healing During Geometry Import*

Technical Notes: *Healing After Geometry Import*

Technical Notes: *Removing Object Intersections*

Technical Notes: *Removing Small Features*

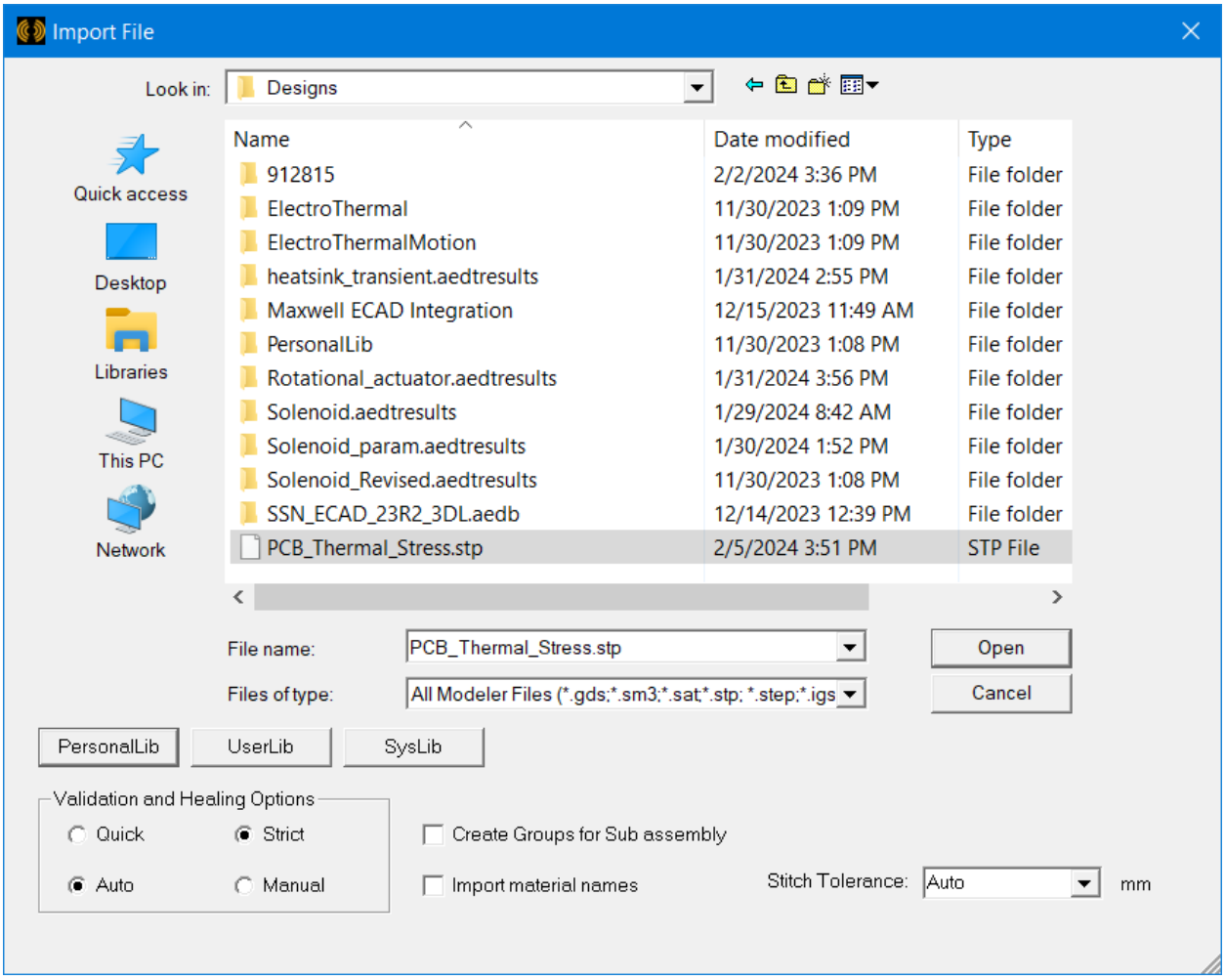
Technical Notes: *Aligning Objects*

Technical Notes: *Troubleshooting if Meshing Still Fails*

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stage One: Healing During Geometry Import

If you do not draw your entire geometry in the Ansys Electromagnetics environment but wish to import (part of) it, in the **Import File** window you select which geometry file to import. At the bottom of this window are the **Validation and Healing Options**. If **Quick** is selected, healing is switched off. If **Strict** is selected, there are two modes of healing available: **Auto** and **Manual**. Auto healing will try to address geometry kernel errors and non-manifold errors, the first two classes of potential problems listed earlier. It will also fix surface normals in the body and update the orientation of body to avoid having a body with negative volume.



Manual healing adds small-feature removal to this. You can remove small features at this stage if you wish. However, the usual approach is to apply auto-healing at this stage and leave small-feature removal until later.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stage Two: Healing after Geometry Import

Healing can only be performed on objects that have no drawing history other than "import". If necessary, object history can be deleted through **Modeler** > **Purge History**. If that causes a warning that another object will be deleted, you may need to purge the history of that other object first, or purge the histories of several objects simultaneously.

At any time after import, you can perform a Validation Check: **Maxwell 3D** > **Validation Check**. This will enable you to focus on bodies and body pairs that need attention before a mesh can be created.

1. Select the objects that have geometry kernel errors, such as `failing api_check_entity()`, and the objects that have non-manifold features, *i.e*. mixed dimensionality.
2. Invoke **Modeler** > **Model Analysis** > **Analyze Objects**.

This will bring up an **Options** dialog to set thresholds for small feature detection and on completion, the **Model Analysis** dialog is displayed. All bodies in the model are shown in the objects grid along with their status. Bodies can have the following status:

- Good
- Null Body
- Analysis not performed
- Invalid entities found
- Small-entity errors

Invalid-entity errors are geometry kernel errors and non-manifold errors. Small-entity errors are small faces, sliver faces and small edges that are optionally detected based on user-defined parameters.

> **Note:** Invalid-entity errors must be fixed before a mesh can be generated.

To fix invalid entity errors:

1. Choose the bodies that have the "Invalid Entities Found" warning.
2. In the same window, choose **Perform** > **Heal Objects**, with or without an optional setting for small-feature removal.

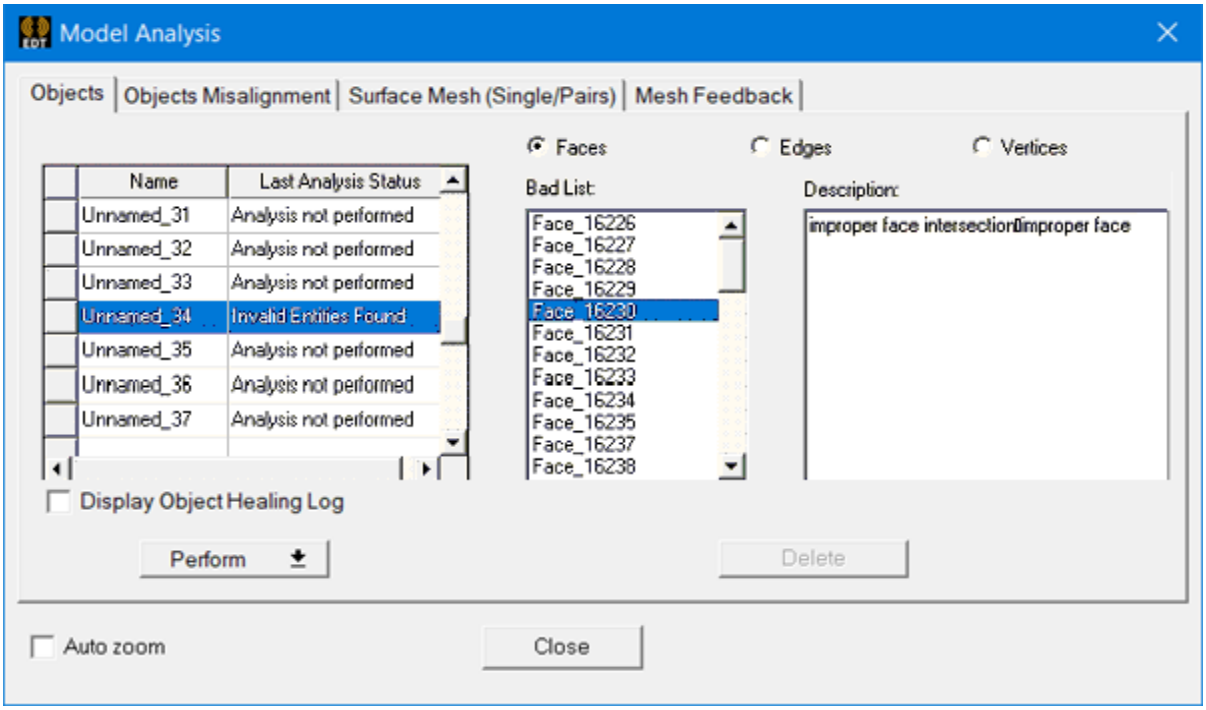
   In most cases, the bodies will be healed and the errors fixed.
3. If errors still persist, select the offending faces and edges and click on **Delete**.

   This will replace the selected face/edge entity by a tolerant edge/vertex respectively.

In order to avoid unintended changes, it is good practice to do the following:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. At the bottom of the **Model Analysis** window, check the **Auto Zoom to Selection** option.
2. Select one face or edge at a time.
3. Decide for each face and edge whether you want to delete it.



> **Note:** Healing causes changes to the geometry and topology of the body being healed. Validation check has to be rerun after healing is done to identify body pairs that intersect. It is possible that after healing, bodies that were previously disjoint now overlap.

In some cases the replacement of the face/edge by tolerant edge/vertex will fail. If the object remains invalid, you know at this point what parts of the object are invalid. You will need to change that part of the object manually, either in Ansys Electromagnetics drawing environment or in the original CAD tool, to make it pass. Often, the invalid entities are in small details that can be changed without noticeably affecting the results of the electromagnetic analysis. For example, it may be possible to create a small object, well placed in the "offending" region, and to unite it with or subtract it from the problematic object, such that the "offending" details no longer exist.

Stage Four: Removing Small Features

Even though, in principle, the geometry may be ready for a mesh to be created, it is possible that small features in the geometry lead to a mesh that is unnecessarily large and contains long and thin tetrahedra that make the simulation converge slower. Small features may even cause the

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

mesh generation to fail. By small, we mean details on an object that are thousands of times smaller than the main features of the object, and that, in most cases, are unintended consequences of the drawing history in another CAD tool. Therefore, it is advantageous to remove small features.

To do this, you may need to purge the history of objects because healing and related operations can only be performed on objects without history beyond import. You may have noticed that you could have invoked small-feature removal at several earlier stages. There is no objection to doing it earlier. The reason why it is presented here as stage four is that the previous stages were necessary while this one is optional.

To start the small-feature removal:

1. Select objects and invoke object analysis through **Modeler** > **Model Analysis** > **Analyze Objects**.

   Alternatively, without objects selected, use **Modeler** > **Model Analysis** > **Show Analysis Dialog** > **Objects** and select objects from the list. In the **Model Analysis window**, invoke **Perform** > **Analyze Objects**.

   The software will report the smallest edge length and the smallest face area, and enable you to set thresholds for the detection of short edges, small faces and sliver faces.

2. Click **OK** to start the analysis.

   As a result of the analysis, the software presents a list of all faces and edges that do not meet the thresholds set by you.

3. Check the **Auto-Zoom to Selection** option at the bottom of the Model Analysis window, and click on small faces and short edges in the list.

   Inspect them visually and decide whether they can be deleted. It is good practice to delete them one by one rather than deleting many at once in order to prevent unintended changes.

   Sometimes, an edge or face cannot be deleted and you get a message notifying you. In that case, either ignore it, or revisit it after deleting some other details first, or revisit it later manually in the 3D drawing environment.

   At this point, the geometry has no errors, no non-manifold objects and no partial object intersections. Furthermore, there are fewer small features that were unintended or unimportant for the electromagnetic analysis, so the quality of the model has improved.

Stage Five: Aligning Objects

Objects that touch each other in imported geometries do not always have well-aligned faces. Often, this is a consequence of the limited level of precision in the imported file. Misaligned faces can cause tiny object intersections or tiny gaps between objects, which in turn can lead to an inefficient mesh or even a failure to create the mesh.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To repair such occurrences in an automated way, you can select groups of objects and invoke **Modeler** > **Model Analysis** > **Analyze Interobject Misalignment**. This will yield face pairs from different bodies that are slightly misaligned with respect to each other.

In the window that shows this list, check the **Auto-Zoom to Selection** option, and select face pairs from the list. When you decide that faces should be aligned, click **Align Faces**. In some cases, face alignment will fail if the topology of the body would change by a large amount after alignment. In that case, you can decide to ignore it, as it may not be a problem, or revisit it later manually in the 3D Modeler environment.

> **Note:** In complicated models, the **Interobject Misalignment** analysis can take a long time if you select all objects before launching the analysis. If you do not know which pairs of objects to analyze, just let the mesh generator try to make a mesh. If the mesh fails, a list is presented to you of misalignments that the mesh generator finds suspicious but did not want to adjust without permission. Not every misalignment in the list is always a problem: this is a list of features that might need your attention.

> **Note:** As face misalignments between touching objects can cause small object intersections, this alignment capability can already serve a useful role in Stage Three.

Stage Six: Troubleshooting if Meshing Still Fails

If mesh generation fails, information about the reasons for the failure is presented under **Modeler** > **Model Analysis** > **Show Analysis Dialog** > **View Mesh Feedback**.

Again, check the **Auto Zoom to Selection** option, and click on the errors in the list. This can give you hints about which parts of the model are causing difficulties.

For instance, there may be self-intersecting bodies or faces. Such errors can have a variety of causes, such as a face that is supposed to be planar, but of which the vertices do not quite lie in the same plane. When you zoom and search you are likely to see what causes the problem.

Also, there may be face misalignments. Once you know they exist, you can inspect them and decide whether to align them under the **Objects Misalignment** tab.

One of the tabs of the **Model Analysis** window is the **Surface Mesh** tab. Under that tab, you can try to create surface meshes for objects and pairs of objects. A surface mesh on selected objects is easier to create than a volume mesh for the whole model, so this can help you to identify quickly which objects are causing difficulties and why.

Also, in order to determine which objects are causing difficulties, you can exclude objects temporarily from the model. If the mesh succeeds without them, this helps to identify the reason for failure. To exclude an object temporarily, select it and uncheck **Model** in its properties window. Then try to create the mesh again.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Once you know which objects make the mesh fail, you can try to make small changes to them that do not affect the electrical properties noticeably but help the mesh maker succeed. For example,

- Zoom in on details and consider removing details;
- Find coincident true surfaces and move one of the faces over a very short distance so the pair of faces is not coincident anymore;
- Split very complicated objects into multiple less-complicated objects;
- Delete a complicated 3D ground object and create a 2D ground through a boundary condition on the appropriate faces of a dielectric;
- Replace imported objects by objects drawn in Ansys Electromagnetics 3D modeling environment. For instance, some CAD tools produce cylinders that consist of two half cylinders that have a seam where they join. The fit is not always perfect.

Finally, for coincident true surfaces, set a very small value for **Surface Deviation** under **Mesh** > **Assign Mesh Operation** > **Surface Approximation**. Parasolid will give them more segments, but you can compensate for that with **Model Resolution**. In a parametric sweep, you can experiment with settings for **Surface Deviation** and **Model Resolution**.

Handling Complicated Models

Complicated models, often imported from a CAD tool or layout tool, may slow down the interface, use a lot of RAM during file I/O and other operations, contain imperfections and object overlaps. After analysis, postprocessing of such models may be time consuming. Maxwell has several options and features that address these problems, which are detailed in the following topics:

- Interface Options for Complicated Models
- Geometry Imperfections and Complicated Models
- Object Overlap Settings for Complicated Models
- Postprocessing Settings for Complicated Models

Interface Options for Complicated Models

To improve the speed of the interface when dealing with complicated geometries, do one or all of the following:

Under **Tools** > **Options** > **Modeler Options**, on the **Display** tab:

1. Set **Default View Render** to **Wire Frame**. Wire-frame rendering is faster than shaded rendering.
2. Turn off **Display UV Isolines**. For models with curved faces, this will simplify the wire-frame display, so the rendering will be faster.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Turn off **Visualize History of Objects**. This will remove visualization of objects that are part of the model history. For large models, this is faster and uses less memory.

Under **View** > **Visualization Settings:**

- Use larger deviations to view curved objects in less detail.

Under **Modeler** > **Import:**

- Uncheck **Check Model** and **Heal Imported Objects**. This helps for complicated models: Validation and healing take considerable time for such models. Use this option to defer checking to a later stage (especially in cases where you know that you want to mesh the model as is).

Under **Tools** > **Options** > **General Options**, on the **Project Options** tab:

- Turn off **Do Autosave** or set the autosave interval to a larger value, for example, 50. Autosave can be time consuming.

Geometry Imperfections and Complicated Models

Modeler > Import

Many formats can be handled, but Ansys recommends importing a version of the geometry that is as close as possible to its source rather than geometries that have been translated before from one format to another or that have been imported into another computational tool and later exported from it.

Modeler > Validation Settings

Geometry imperfections are listed as kernel errors when executing **Maxwell** > **Validation Check** and when starting an analysis. Ansys recommends attempting to heal objects with such errors. However, Maxwell enables you to bypass the errors (not the check itself) by choosing a setting under **Modeler** > **Validation Settings**. Check **Warning Only** to ignore all errors. Check **Basic** to bypass all but the most severe errors. The Maxwell mesh generator has been enhanced to handle many geometry errors.

Object Overlap Settings for Complicated Models

Complicated geometries often have small object overlaps. The **Maxwell 3D** > **Design Settings** > **Set Material Override** tab has an **Enable material override** setting that addresses this issue. This setting will allow overlaps between dielectrics and metals. In the overlap region, the metal will locally take priority over the dielectric, as if this part of the dielectric has been subtracted. Overlaps between two dielectrics and overlaps between two metals are still not allowed.

Postprocessing for Complicated Models

Under **Tools** > **Options** > **General Options** > **Desktop Performance**:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Turn off **Update reports on file open**. This will disable expensive updating of reports and plots when opening a project.
- Turn off **Dynamically update postprocessing data during edits**. This will disable expensive updating of existing reports and plots.

Materials

This section of the *Technical Notes* includes information on the following material properties:

Simple Materials

- Relative Permeability
- Relative Permittivity
- Bulk Conductivity
- Dielectric Loss Tangent
- Magnetic Loss Tangent

Frequency Dependent

- Frequency-dependent material properties.

Anisotropic materials

- Anisotropic materials
- Anisotropic relative permeability tensors
- Anisotropic relative permittivity tensors
- Anisotropic conductivity tensors
- Anisotropic dielectric loss tangent tensors
- Anisotropic magnetic loss tangent tensors
- Anisotropic materials and ports

Ferrite materials

- Magnetic Saturation
- Lande G Factor
- Delta H

Magnetostrictive materials

- Magnetostriction Modeling of Nonlinear Magnetic Material

Relative Complex Permeability

The permeability of dielectrics is assumed to be complex, as follows:

$$\mu = \mu' - j\mu''$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The relative permeability is a dimensionless quantity, defined as follows:

$$\mu' = \mu_r \mu_0$$

where μ_0 is the permeability of free space.

The permeability can also be expressed as

$$\mu = \mu' \left(1 - j\frac{\mu''}{\mu'}\right)$$

where μ' is the real portion of μ and μ''/μ' is the magnetic loss tangent.

Relative Complex Permittivity

The permittivity of dielectrics is assumed to be complex, as follows:

$$\varepsilon = \varepsilon' - j\varepsilon''$$

which can also be expressed as

$$\varepsilon = \varepsilon' \left(1 - j\frac{\varepsilon''}{\varepsilon'}\right)$$

where ε' is the real portion of ε and from which the relative permittivity is defined as

$$\varepsilon' = \varepsilon_r \varepsilon_0$$

where ε_0 is the permittivity of free space and $\varepsilon''/\varepsilon'$ is the dielectric loss tangent.

If a material's losses due to bulk conductivity will be significant, such as in semiconductor dielectric materials, an additional bulk conductivity value, σ, must be added. From the time harmonic form of Maxwell's equations, the complex permittivity, ε_c is defined as

$$\nabla \times \boldsymbol{H} = (\sigma + j\omega\varepsilon)\boldsymbol{E} = j\omega\varepsilon_c \boldsymbol{E}$$

where

$$\varepsilon_c = \varepsilon\left(1 - jtan\delta - j\frac{\sigma}{\omega\varepsilon}\right)$$

Bulk Conductivity

Maxwell is capable of including conductivity in the model either as a bulk material loss factor, similar to dielectric loss tangent, or as an impedance boundary condition applied to the outer surfaces of the object. The choice of bulk material loss instead of the boundary condition is made by selecting **Solve Inside** in the **Properties** window and by adding an Impedance Boundary on the surface of the object.

The choice between bulk material loss and the surface impedance boundary condition is problem dependent. The boundary condition can be applied whenever the conductor is much

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

thicker than the skin depth at the solution frequency. In this case, the unknowns within the conductor are not included in the unknown vector, resulting in a smaller matrix and a faster analysis. However, if the conductor is not thick relative to the skin depth, the bulk material conductivity must be used to arrive at an accurate solution. With this assumption, the wave equation reduces to

$$\nabla \times \left(\frac{-1}{j\omega\mu}\nabla \times \boldsymbol{E}\right) = \left(j\omega\varepsilon + \sigma\right)\boldsymbol{E}$$

Dielectric Loss Tangent

To represent a dielectric that dissipates the power of a high-frequency electric field, enter a dielectric loss tangent, $\varepsilon''/\varepsilon'$ property value for the material. The smaller the loss tangent, the less lossy the material.

The dielectric loss tangent may vary with frequency. To simulate the variances, define a function for the dielectric loss tangent.

Magnetic Loss Tangent

To represent a magnetic material such as ferrite that dissipates the power of a high-frequency magnetic field, enter a magnetic loss tangent, μ''/μ', property value for the material. The smaller the loss tangent, the less lossy the material.

> **Note:** If you plan to do a fast-frequency sweep for a design that includes dielectrics, make sure that the dielectric or magnetic loss tangent does not vary significantly over the requested frequency range. If they do, the results may not be what you expect. In cases where the loss tangent does vary significantly over the frequency range in which you are interested, copy and solve the design several times, adjusting the loss tangent and associated frequency range for the copied design so that the loss tangent is relatively stable over the design's requested frequency range.

> **Note:** When magnetic loss tangent is used, the resulting hysteresis core loss will be reflected in the resistance term for both the impedance matrix and the Simplorer Dynamic AC Magnetic ROM.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Imaginary Permeability

AC Magnetic

Some materials exhibit a permeability that includes both a real and imaginary component. The imaginary component is used to model magnetic losses in a time-varying field using the relationship:

$$B = (\mu_r' - j(\mu_r''))\mu_0 H$$

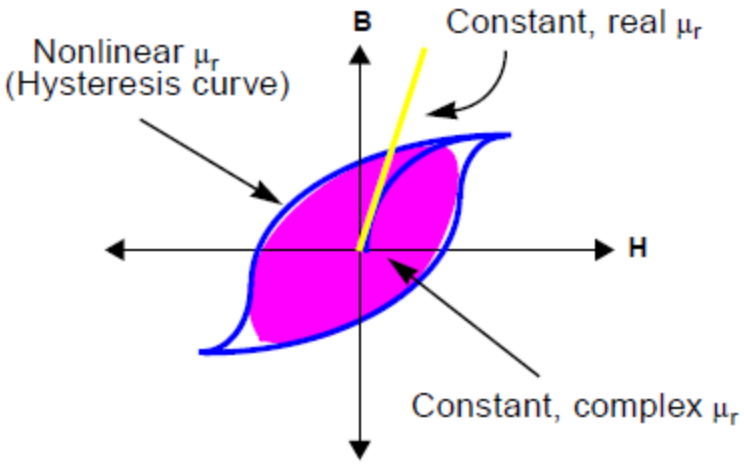
where

μ_r' is the real component of the relative permeability.

μ_r'' is the imaginary component of the relative permeability.

μ_0 is the permeability of free space.

As shown below, a complex relative permeability causes the B field to lag behind the H field – similar to the behavior of a nonlinear, lossy material. The power loss during this cycle (the color shaded area) is approximately equal to the hysteresis loss (the area within the blue lines). The hysteresis curve for a material with a constant, real permeability (the straight yellow line) is shown as a reference.



When the nature of losses is also due to magnetic hysteresis, the Maxwell application allows you to model such effects using complex permeability by allowing the specification of the tangent of magnetic loss. When a linear material exhibits hysteresis effects, it should be characterized using a complex permeability, as below:

$$\underline{\mu} = \mu' - j\mu'' = \mu'\left(1 - j\tan\delta\right)$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where tan δ is the magnetic loss tangent, $\frac{\mu''}{\mu'}$

Enter the imaginary relative permeability of a material, μ'', in the **Imag. Permeability** field. The default imaginary permeability of zero is that of a material that exhibits no magnetic loss in a time-varying field.

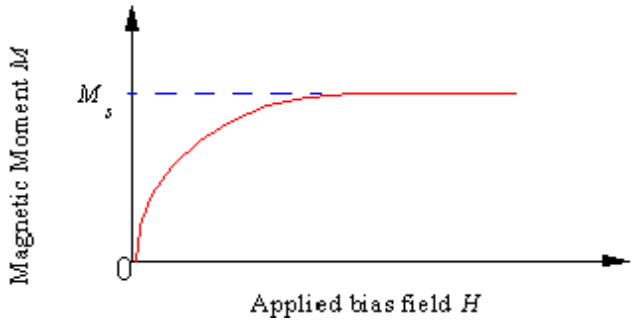
Ferrite Materials

Ferrite materials are used to model the interaction between a microwave signal and a material whose magnetic dipole moments are aligned with an applied bias field. The gyrotropic quality of the ferrite is evident in the permeability tensor, which is Hermitian in the lossless case. The Hermitian tensor form leads to the non-reciprocal nature of the devices containing microwave ferrites. If the microwave signal is circularly polarized in the same direction as the precession of the magnetic dipole moments, the signal interacts strongly with the material. When the signal is polarized in the opposite direction to the precession, the interaction will be weaker. Because the interaction between the signal and material depends on the direction of the rotation, the signal propagates through a ferrite material differently in different directions.

If you assign a ferrite material to an object, you must assign a magnetic bias source to the object.

Magnetic Saturation

A material with a non-zero magnetic saturation is considered to be a ferrite. When a ferrite is placed in a uniform magnetic field, the magnetic dipole moments of the material begin to align with the field. As the strength of the applied bias field increases, more of the dipole moments align. The magnetic saturation, M_S, is a property that describes the point at which all of the magnetic dipole moments of the material become aligned. At this point, further increases in the applied bias field strength do not result in further saturation. The relationship between the magnetic moment, M and the applied bias field, H, is shown below.



The magnetic saturation, $4\pi M$, is entered in gauss.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lande G Factor

The Lande *g* factor is a ferrite property that, on a microscopic level, describes the total magnetic moment of the electrons according to the relative contributions of the orbital moment and the spin moment. When the total magnetic moment is due entirely to the orbital moment, *g* is equal to one. When the total magnetic moment is due entirely to the spin moment, *g* is equal to two. For most microwave ferrite materials, *g* has a range from 1.99 to 2.01. The Lande *g* factor is dimensionless.

Delta H

Delta H is the full resonance line width at half-maximum, which is measured during a ferromagnetic resonance measurement. It relates to how rapidly a precessional mode in the biased ferrite will damp out when the excitation is removed. The factor ΔH doesn't appear in the permeability tensor; instead, the factor α appears. The factor α is computed from

$$\alpha = -\frac{\gamma \mu_0 \Delta H}{2\omega}$$

The factor α changes the κ and χ terms in the permeability tensor from real to complex, which makes the tensor complex non-symmetric (where it had been Hermitian for lossless ferrites).

Enter the full resonance line width at half maximum in the **Delta H** value box. Delta H is measured at a specific frequency. That frequency needs to be entered by the user at the interface. The frequency where Delta H is typically measured is 9.4 GHz.

Anisotropic Materials

Anisotropic materials have characteristics that vary with direction. These characteristics are defined by their anisotropy tensors. You must define three diagonals each for anisotropic permittivity, electric loss tangent, conductivity, permeability, and magnetic loss tangent. Each diagonal represents a tensor of your model along an axis. The alignment of the materials axis with respect to the object is discussed in Assigning Material Property Types.

The following anisotropic characteristics can be defined:

- Anisotropic relative permeability tensors
- Anisotropic relative permittivity tensors
- Anisotropic conductivity tensors
- Anisotropic dielectric loss tangent tensors
- Anisotropic magnetic loss tangent tensors
- Anisotropic materials and ports

Anisotropic Relative Permeability Tensors

The relative permeability tensor for an anisotropic material is described by

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$[\mu] = \begin{bmatrix} \mu_1\mu_0 & 0 & 0 \\ 0 & \mu_2\mu_0 & 0 \\ 0 & 0 & \mu_3\mu_0 \end{bmatrix}$$

where

- μ_1 is the relative permeability along one axis of the material's permeability tensor.
- μ_2 is the relative permeability along the second axis.
- μ_3 is the relative permeability along the third axis.
- μ_0 is the permeability of free space.

The relationship between *B* and *H* is

$$\begin{matrix} x \to \mu \\ y \to \nu \\ z \to w \end{matrix} \begin{bmatrix} B_x \\ B_y \\ B_z \end{bmatrix} = [\mu] \begin{bmatrix} H_x \\ H_y \\ H_z \end{bmatrix}$$

To specify the relative permeability for an anisotropic material, enter the μ_1, μ_2, and μ_3 values in the **Value** boxes of the **T(1,1)**, **T(2,2)**, and **T(3,3)** rows, respectively. If the relative permeability is the same in all directions, use the same value for μ_1, μ_2, and μ_3. These values can also be entered as variables.

Anisotropic Relative Permittivity Tensors

The relative permittivity tensor for an anisotropic material is described by

$$[\varepsilon] = \begin{bmatrix} \varepsilon_1\varepsilon_0 & 0 & 0 \\ 0 & \varepsilon_2\varepsilon_0 & 0 \\ 0 & 0 & \varepsilon_3\varepsilon_0 \end{bmatrix}$$

where

- ε_1 is the relative permittivity of the material along one tensor axis.
- ε_2 is the relative permittivity along the second axis.
- ε_3 is the relative permittivity along the third axis.
- ε_0 is the permittivity of free space.

The relationship between *E* and *D* is then

$$\begin{matrix} x \to \mu \\ y \to \nu \\ z \to w \end{matrix} \begin{bmatrix} D_x \\ D_y \\ D_z \end{bmatrix} = [\varepsilon] \begin{bmatrix} E_x \\ E_y \\ E_z \end{bmatrix}$$

To specify the relative permittivity for an anisotropic material, enter the ε_1, ε_2, and ε_3 values in the **Value** boxes of the **T(1,1)**, **T(2,2)**, and **T(3,3)** rows, respectively. These values can also be entered as variables.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Anisotropic Conductivity Tensors

The conductivity tensor for an anisotropic material is described by

$$[\sigma] = \begin{bmatrix} \sigma_1 & 0 & 0 \\ 0 & \sigma_2 & 0 \\ 0 & 0 & \sigma_3 \end{bmatrix}$$

where

- σ_1 is the relative conductivity along one axis of the material's conductivity tensor.
- σ_2 is the relative conductivity along the second axis.
- σ_3 is the relative conductivity along the third axis.

The relationship between *J* and *E* is then

$$\begin{matrix} x \to \mu \\ y \to \nu \\ z \to w \end{matrix} \begin{bmatrix} J_x \\ J_y \\ J_z \end{bmatrix} = [\sigma] \begin{bmatrix} E_x \\ E_y \\ E_z \end{bmatrix}$$

To specify the conductivity for an anisotropic material, enter the σ_1, σ_2, and σ_3 values in the **Value** boxes of the **T(1,1)**, **T(2,2)**, and **T(3,3)** rows, respectively. The values of σ_1 and σ_2 apply to axes that lie in the xy cross-section being modeled. The values of σ_3 apply to the z-component. These values affect current flowing in dielectrics between the conductors. These values can also be entered as variables.

Anisotropic Dielectric Loss Tangent Tensors

The dielectric loss tangent tensor for an anisotropic material is described by

$$[\overline{\varepsilon}] = \begin{bmatrix} \varepsilon_1'(1 - jtan\delta_1) & 0 & 0 \\ 0 & \varepsilon_2'(1 - jtan\delta_2) & 0 \\ 0 & 0 & \varepsilon_3'\left(1 - jtan\delta_3\right) \end{bmatrix}$$

where

- tan δ_1 is the ratio of the imaginary relative permittivity to the real relative permittivity in one direction.

$$\tan\delta_1 = \frac{\varepsilon_1''}{\varepsilon_1'}$$

- tan δ_2 is the ratio of the imaginary relative permittivity to the real relative permittivity in the second direction.

$$\tan\delta_2 = \frac{\varepsilon_2''}{\varepsilon_2'}$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- tan δ_3 is the ratio of the imaginary relative permittivity to the real relative permittivity in the third orthogonal direction.

$$\tan\delta_3 = \frac{\varepsilon_3''}{\varepsilon_3'}$$

- ε'_1, ε'_2, and ε'_3 are the real relative permittivities specified earlier.

The relationship between *D* and *E* will then be

$$\begin{matrix} x \to \mu \\ y \to \nu \\ z \to w \end{matrix} \begin{bmatrix} D_x \\ D_y \\ D_z \end{bmatrix} = [\overline{\varepsilon}] \begin{bmatrix} E_x \\ E_y \\ E_z \end{bmatrix}$$

To specify the electric loss tangent for an anisotropic material, enter the $\tan\delta_1$, $\tan\delta_2$, and $\tan\delta_3$ values in the **Value** boxes of the **T(1,1)**, **T(2,2)**, and **T(3,3)** rows, respectively. These values can also be entered as variables.

Anisotropic Magnetic Loss Tangent Tensors

The magnetic loss tangent tensor for an anisotropic material is described by

$$[\overline{\mu}] = \begin{bmatrix} \mu_1'(1 - j\,tan\delta_{M1}) & 0 & 0 \\ 0 & \mu_2'(1 - j\,tan\delta_{M2}) & 0 \\ 0 & 0 & \mu_3'(1 - j\,tan\delta_{M3}) \end{bmatrix}$$

where

- tan δ_{M1} is the ratio of the imaginary relative permeability to the real relative permeability in one direction.

$$\tan\delta_{M1} = \frac{\mu_1''}{\mu_1'}$$

- tan δ_{M2} is the ratio of the imaginary relative permeability to the real relative permeability in the second direction.

$$\tan\delta_{M2} = \frac{\mu_2''}{\mu_2'}$$

- tan δ_{M3} is the ratio of the imaginary relative permeability to the real relative permeability in the third orthogonal direction.

$$\tan\delta_{M3} = \frac{\mu_3''}{\mu_3'}$$

- μ'_1, μ'_2, and μ'_3 are the real relative permeabilities specified earlier.

The relationship between *B* and *H* will then be

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\begin{matrix} x \rightarrow \mu \\ y \rightarrow \nu \\ z \rightarrow w \end{matrix} \begin{bmatrix} B_x \\ B_y \\ B_z \end{bmatrix} = [\overline{\mu}] \begin{bmatrix} H_x \\ H_y \\ H_z \end{bmatrix}$$

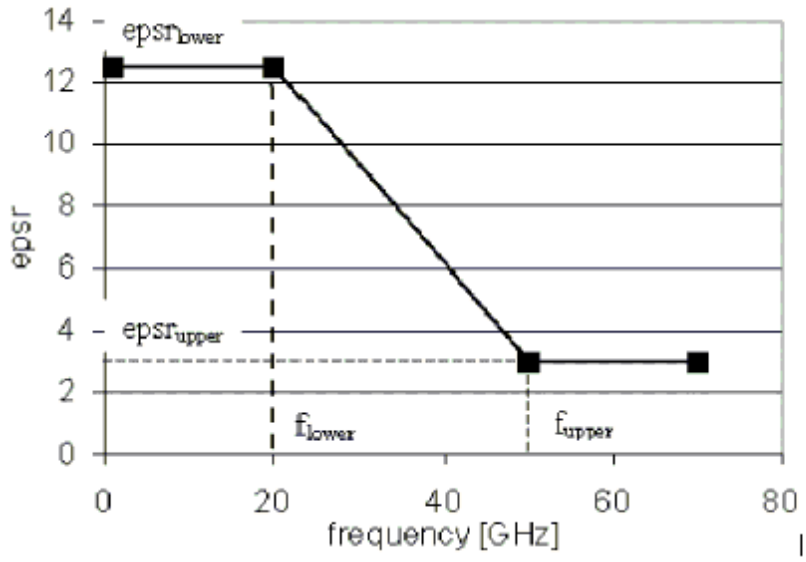
To specify the magnetic loss tangent for an anisotropic material, enter the tan δ_{M1} , tan δ_{M2}, and tan δ_{M3} values in the **Value** boxes of the **T(1,1)**, **T(2,2)**, and **T(3,3)** rows, respectively. These values can also be entered as functions.

Anisotropic Materials and Ports

An anisotropic material can be in contact with a port if one principal axis of the anisotropic material is aligned normal to the port.

Frequency-Dependent Material Properties

The properties of some materials vary with the frequency of the field excitation. This frequency dependence is often linear within a certain frequency range and constant outside of the frequency range, as shown below:



where

- $epsr_{lower}$ is the relative permittivity of a material below the frequency range 20 - 50 GHz.
- $epsr_{upper}$ is the relative permittivity of a material above the frequency range 20 - 50 GHz.
- f_{lower} is the lower frequency, below which the material property is constant.
- f_{upper} is the upper frequency, above which the material property is constant.

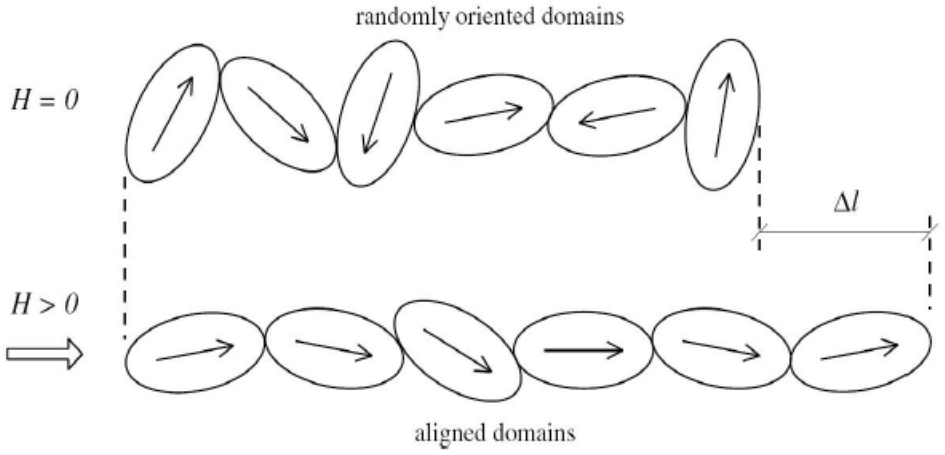
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In general, to account for this variance within a given frequency range, use the **Piecewise Linear Material Input** window to specify a property's values at frequencies below and above the frequency range. Based on these values, Maxwell automatically creates a linear dataset that specifies the property's values at the desired frequencies during solution generation. This dataset can be modified with additional points if desired.

> **Note:** The 3D AC conduction solver only solves the function inserted in the material property field and does not check the upper and lower limits.

Magnetostriction Modeling of Magnetic Material

Magnetostriction was first observed by J. Joule in 1842. Currently, it is typically described as "deformation of a body" as a result of its magnetization. The elongation or contraction in the direction of applied magnetic field is usually between 10^{-5} and 10^{-3} and accompanied by the opposite sign changing in the transverse direction, so that the volume remains almost the same. The phenomenon of magnetostriction can be regarded as an energy transduction (or transformation) from mechanical to magnetic and vice-versa. It can be described as a bidirectional magnetomechanical coupling between the mechanical and magnetic fields in the magnetostrictive material. The origin of this phenomenon can be traced to the alignment of the magnetic domains inside the material. The following figure is a schematic of the effect simplified to one dimension.



When no magnetic field is applied to the system, the series of domains have randomly oriented magnetic moments. Whereas when a magnetic field is applied, the series of domains rotate to partially align themselves to the magnetic field direction, resulting in a change in length Δl. Applying a sufficiently high magnetic field will result in perfectly aligned domains, in which case the material achieves the maximum magnetically induced strain (that is, peak magnetostriction). Magnetostriction requires the magnetic domains to be longer in one dimension than the other

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

two to obtain a change in length when the domain rotates. This, in general, results in anisotropy of the crystal structure of the material.

Magnetostriction is considered to be one of the main sources of noise in transformers. When an alternating voltage is applied to one or more windings of a transformer, a magnetic flux is generated in the transformer core laminations made of grain-oriented electrical steel. The nonlinear anisotropic property of magnetostriction creates alternating changes of the core dimensions due to the varying magnetic flux in the laminations. Those magnetostrictive forces cause core vibrations that are transmitted to the tank via the insulation oil and the core clamping points. Part of the mechanical energy is eventually radiated by the tank walls as noise. In electric motors, magnetostrictive forces also contribute to motor vibration and noise.

On the other hand, magnetostrictive materials convert energy between the mechanical and magnetic domains (inverse magnetostriction). They deform in response to applied magnetic fields and change their magnetic state when stressed. Some materials (such as Terfenol-D, Galfenol) have the ability to produce large magnetostrictive strains at moderate fields. Short response times (in the millisecond range) combined with resolutions in the order of microstrains make these materials well-suited to precision sensing and actuation mechanisms.

Due to the complexity of the mechanism of magnetostriction, it is very difficult to predict it with analytic models. In general, empirical methods based on statistics and dimensional basic parameters are used by most transformer manufacturers and sensor producers. Those approaches present limitations when applied to new designs and do not enable accurate parametric studies. Therefore, prediction models based on finite element formulations will be utilized to accurately describe the complex interactions of the various design parameters and the coupling of the physical fields.

The finite element method (FEM) is widely used in engineering practice because using the irregular grids, it can model complex inhomogeneous and anisotropic materials and represent complicated geometry. The FEM discretization produces a set of matrix differential equations. Because of the nonlinearity, the matrices generally are dependent of the solution vectors, so an iteration method such as Newton-Raphson method should be used to solve these nonlinear matrix equations. Namely, the nonlinear matrix equations are linearized for each nonlinear iteration. The linearized matrix equations may be solved by either a direct or iterative matrix solver.

The governing equation of magnetic field is given by

$$\sigma\frac{\partial A}{\partial t} + \nabla \times \left(\frac{\nabla \times A}{\mu}\right) = J \qquad \text{(Equation 1)}$$

Where **A** is the magnetic vector potential, **σ** is the electrical conductivity, and **μ** is the magnetic permeability of material. **J** represents the current density.

The magnetic flux density **B** can be expressed as

$$B = \nabla \times A \qquad \text{(Equation 2)}$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The relationship between **B** and **H** (magnetic field) is based on the constitutive law:

$B = \mu \boldsymbol{H}$ (Equation 3)

For a discretized mechanical system, the governing equilibrium equation can be written as

$Kr + R_0 = R$ (Equation 4)

where K is the system stiffness matrix, R_0 is the body force vector, and R the vector of external force, r is vector of node displacement. The strain-displacement relationship can be written as

$S = \nabla r$ (Equation 5)

where **S** is the strain tensor. And the stress-strain relationship can be described by

$T = CS$ (Equation 6)

where **T** is the stress tensor, and **C** is stiffness matrix.

When a magnetic field is applied to a magnetostrictive material, strain occurs within the material placed in a magnetic field in response to the flux density within the material, while the stress gives feedback to affect the magnetic field. For coupled method, we have

$$\begin{bmatrix} B \\ S \end{bmatrix} = \begin{bmatrix} \mu & d \\ d^T & C^{-1} \end{bmatrix} = \begin{bmatrix} H \\ T \end{bmatrix}$$ (Equation 7)

$$d^T = \frac{\partial \lambda}{\partial H}\Big|_T$$ (Equation 8)

$$d = \frac{\partial B}{\partial T}\Big|_H$$ (Equation 9)

Where **d** and $\mathbf{d^t}$ are magnetomechanical coupling coefficients, **λ** is magnetostriction induced by magnetic field **H**. The magnetic field and mechanical stress field are solved together, which means both fields are solved on the same mesh simultaneously by using the one matrix solver.

For decoupled method, the magnetostriction may be determined once the magnetic field distribution is obtained from FEM simulation. Hence, an elastic analysis is reduced to the problem of finding the stress due to an external load caused by a force applied to a material with a previously known magnetostriction, the latter unrelated to the stress. Magnetostriction is transformed into stress with the usage of Hooke's law. Equation (7) can be rewritten as

$B = \mu H + dT$ (Equation 10)

$S = C^{-1}T + d^T H$ (Equation 11)

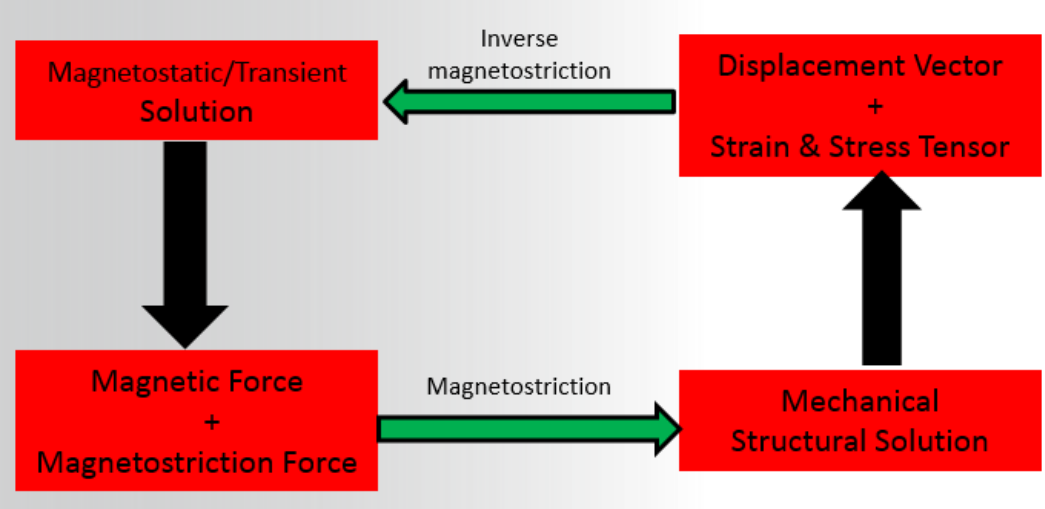
where

$d^T H$: magnetostrictive, deformation induced by magnetic field **H**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

dT: inverse magnetostriction, impact of stress **T** on magnetic field **B**.

By using the decoupled method, the magnetic field and stress field can be solved independently with different mesh and different solver, and magnetostriction and inverse magnetostriction can be involved in either one or both. Compared to the coupled method, the decoupled method is more flexible and general. The process for the decoupled method is shown in the following figure:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29 - Maxwell 2D Technical Notes

The Ansys Maxwell® 2D application is an interactive software package that uses the finite element method (FEM) analysis to simulate and solve two-dimensional electromagnetic field problems. The Maxwell application integrates with other Ansys software to perform complex tasks while remaining easy to use.

The Maxwell 2D application is used for analyzing electric and magnetic fields in structures with uniform cross-sections or full rotational symmetry — where the field patterns in the entire device can be analyzed by modeling the field patterns in its cross-section.

In general, 2D modeling of electromagnetic fields solves for the fields in a 2D cross-section of a device which is considered to be placed far away from the extremities of the devices, thus ignoring end effects. While it can provide excellent accuracy for all applicable modeling tasks, 2D solution sequences cannot be always used to accurately model electromagnetic devices. Devices where end effects are significant or that may not exhibit complete rotational symmetry must be modeled in 3D for greatest accuracy.

These technical notes contain background information on the theory behind the Maxwell module, including

- A list of all Maxwell software modules
- Discussion of background theory; capacitance, inductance, impedance, conductance, and admittance matrices; virtual force and torque; flux linkage; and current flow for the following field solvers:
- Electrostatic
- Magnetostatic
- AC Magnetic
- DC conduction
- AC conduction
- Transient
- The differences between field solutions in Cartesian (XY) and axisymmetric (RZ) models
- An overview of the phasor notation used in this guide to represent complex field quantities

Solvers

Maxwell 2D consists of the following solver modules:

Electrostatic Field Solver	Computes the static electric field that exists in a structure given a distribution of DC voltages and static charges. A capacitance matrix, force, torque, and flux linkage may also be computed from the electric field.
Magnetostatic Field Solver	Computes the static magnetic field that exists in a structure given a distribution of DC currents and permanent magnets. The magnetic field

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	may be computed in structures with both nonlinear and linear materials. An inductance matrix, force, torque, and flux linkage may also be computed from the energy stored in the magnetic field.
AC Conduction Field Solver	Computes the AC voltages and current density distribution in a material having both conductive and dielectric properties given a distribution of AC voltages. An admittance matrix and current flow may also be computed from the calculated fields.
DC Conduction Field Solver	Computes the DC currents that flow in a lossy dielectric given a distribution of DC voltages. A conductance matrix and current flow may also be computed from the computed electric field solution.
AC Magnetic Field Solver	Computes the oscillating magnetic field that exists in a structure given a distribution of AC currents. Also computes current densities, taking into account all eddy current effects (including skin effects). An impedance matrix, force, torque, core loss, and current flow may also be computed from the computed field solution.
Transient Magnetic Solver	Computes transient (Time Domain) magnetic fields caused by permanent magnets, conductors, and windings supplied by voltage and/or current sources with arbitrary variation as functions of time, position and speed. It can also be coupled with external circuits. Rotational or translational motion effects can be included in the simulation.

Axisymmetric Field Simulation

If you selected **Cylindrical About Z** as the geometry mode when setting the solution type, a special version of the selected field solver is used. It assumes that the 2D geometry being studied sweeps 360° around the z-axis of a cylindrical coordinate system, enabling you to model devices like solenoids and insulators that have an axis of rotational symmetry. All electric or magnetic fields in the problem must also be rotationally symmetric so that the solution in any RZ-plane is the same as any other.

> **Note:** All cylindrical about Z models must be drawn in the XZ plane with $X \geq 0$.

Axisymmetric field simulation is available for the following solvers:

- Electrostatic
- Magnetostatic
- AC Magnetic
- DC Conduction
- AC Conduction
- Transient with and without motion

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Any field solution involving a long plank-shaped object is different from a solution involving an annular object — even though the same equation is solved for both types of geometries. Mathematically, the difference between Cartesian and axisymmetric field solutions arises from the difference in how gradients, curls and divergences are expressed in the two coordinate systems.

- In a Cartesian (xyz) coordinate system, these operators are defined as follows (where F is a scalar quantity and **F** is a vector quantity):

$$\nabla \mathrm{F}\left(\mathrm{x},\mathrm{y},\mathrm{z}\right) = \frac{\partial F}{\partial x}\hat{x} + \frac{\partial F}{\partial y}\hat{y} + \frac{\partial F}{\partial z}\hat{z}$$

$$\nabla \bullet \mathrm{F}\left(\mathrm{x},\mathrm{y},\mathrm{z}\right) = \frac{\partial F_x}{\partial x} + \frac{\partial F_y}{\partial y} + \frac{\partial F_z}{\partial z}$$

$$\nabla \times \mathrm{F} = \begin{bmatrix} \hat{x} & \hat{y} & \hat{z} \\ \frac{\partial}{\partial x} & \frac{\partial}{\partial y} & \frac{\partial}{\partial z} \\ F_x & F_\mathrm{y} & F_z \end{bmatrix}$$

If **F** has a z-component only, the curl is defined as

$$\nabla \times \mathrm{F} = \frac{\partial F_z}{\partial y}\hat{x} - \frac{\partial F_z}{\partial x}\hat{y}$$

- In a cylindrical (r, Φ, z) coordinate system, these operators are defined as

$$\nabla F\left(r,\Phi,z\right) = \frac{\partial F}{\partial r}\hat{r} + \left(\frac{1}{r}\right)\frac{\partial F}{\partial \Phi}\hat{\Phi} + \frac{\partial F}{\partial z}\hat{z}$$

$$\nabla \bullet F\left(r,\Phi,z\right) = \left(\frac{1}{r}\right)\frac{\partial (rF_r)}{\partial r} + \left(\frac{1}{r}\right)\frac{\partial F_\Phi}{\partial \Phi} + \frac{\partial F_z}{\partial z}$$

$$\nabla \times \mathrm{F} = \begin{bmatrix} \hat{r} & \widehat{r\Phi} & \hat{z} \\ \frac{\partial}{\partial r} & \frac{\partial}{\partial \Phi} & \frac{\partial}{\partial z} \\ F_r & rF_\Phi & F_z \end{bmatrix}$$

If **F** has a Φ-component only, the curl is defined as

$$\nabla \times \mathrm{F} = \left(\frac{1}{r}\right)\left[\frac{\partial (rF_\Phi)}{\partial z}\hat{r} + \frac{\partial F_\Phi}{\partial r}\hat{z}\right]$$

In the cylindrical case, an extra *r* shows up in places where there is no analogous *x* in the Cartesian case, resulting in a different field solution. The calculations involving the differences in mathematics between axisymmetric and Cartesian models are implicitly handled by the Maxwell and are transparent to you.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Electrostatic Field Simulation

The electrostatic field simulator computes static electric fields arising from potential differences and charge distributions.

Electrostatic Theory

The electrostatic field simulator solves for the electric potential, $\Phi(x,y)$, in this field equation:

$$\nabla \bullet (\varepsilon_r \varepsilon_0 \nabla \Phi (x, y)) = -\rho$$

where

- $\Phi(x,y)$ is the electric potential.
- ε_r is the relative permittivity. It can be different for each material.
- ε_o is the permittivity of free space, 8.854 x 10^{-12} F/m.
- $\rho(x,y)$ is the charge density.

This equation is derived from Gauss's Law and from Faraday's law of induction. Gauss's Law indicates that the net electric flux passing through any closed surface is equal to the net positive charge enclosed by that surface. In differential form, Gauss's Law is

$$\nabla \bullet D = \rho$$

where **D**(x,y) is the electric flux density. Because $D = \varepsilon_r \varepsilon_o E$, then

$$\nabla \bullet (\varepsilon_r \varepsilon_0 E(x, y)) = \rho$$

In a static field, as a consequence of Faraday's law, $E = -\nabla\Phi$. Therefore,

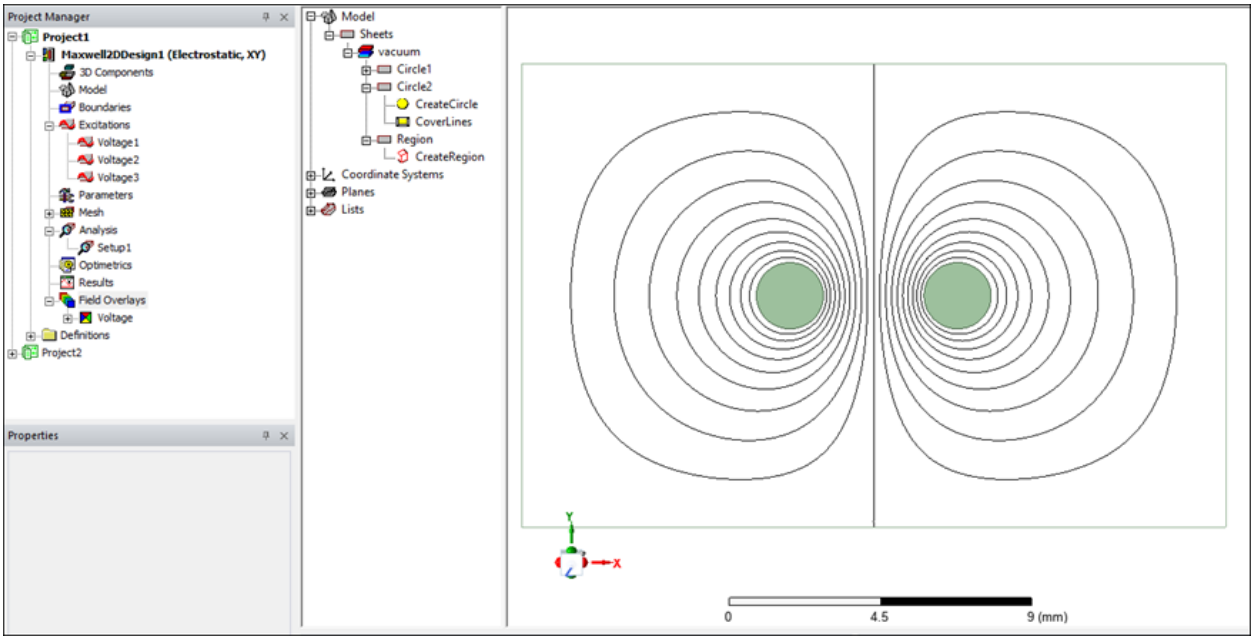
$$\nabla \bullet (\varepsilon_r \varepsilon_0 \nabla \Phi (x, y)) = -\rho$$

which is the equation that the electrostatic field simulator solves using the finite element method.

After the solution for the potential is generated, the system automatically computes the E-field and D-field using the relations $E = -\nabla\Phi$ and $D = \varepsilon_r \varepsilon_o E$.

A contour plot of electric potential generated by the electrostatic field simulator is shown below:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Capacitance

At the simplest level, capacitance relates to the amount of energy stored in the electric field in and surrounding a structure. In a single circuit, the capacitance represents the amount of energy stored in the electric field that arises due to a voltage differential across a dielectric:

$$W_e = \frac{1}{2}Cv^2$$

where W_e is the energy stored in the electric field, C is the capacitance, and v is the voltage across the dielectric.

Maxwell computes the capacitance between two conductors by simulating the electric field that arises when a voltage differential is applied. By computing the energy stored in the field, the corresponding capacitance can be computed:

$$C = \frac{2W_e}{v^2}$$

To compute capacitance using this method, the E-field and D-field associated with a given distribution of voltages must first be computed. The electrostatic field simulator, which computes the electric potential at all points in the problem region, does this.

Capacitance in Terms of Charges and Voltages

A capacitance matrix represents the charge coupling within a group of conductors — that is, the relationship between charges and voltages for the conductors. Given the three conductors

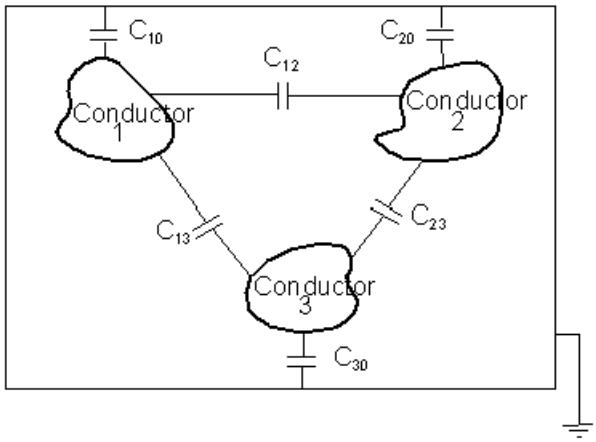
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

shown below, with the outside boundary taken as a reference, the net charge on each object is

$$Q_1 = C_{10} V_1 + C_{12}(V_1 - V_2) + C_{13}(V_1 - V_3)$$

$$Q_2 = C_{20} V_2 + C_{12}(V_2 - V_1) + C_{23}(V_2 - V_3)$$

$$Q_3 = C_{30} V_3 + C_{13}(V_3 - V_1) + C_{23}(V_3 - V_2)$$



This can be expressed in matrix form as

$$\begin{bmatrix} Q_1 \\ Q_2 \\ Q_3 \end{bmatrix} = \begin{bmatrix} C_{10} + C_{12} + C_{13} & -C_{12} & -C_{13} \\ -C_{12} & C_{20} + C_{12} + C_{23} & -C_{23} \\ -C_{13} & -C_{23} & C_{30} + C_{13} + C_{23} \end{bmatrix} = \begin{bmatrix} V_1 \\ V_2 \\ V_3 \end{bmatrix}$$

The capacitance matrix above gives the relationship between *Q* and *V* for the three conductors and ground. In a device with *n* conductors, this relationship would be expressed by an *n* x *n* capacitance matrix. Capacitance matrix values are specified in farads (coulombs/volt).

If one volt is applied to Conductor 1 and zero volts is applied to the other two conductors, the capacitance matrix becomes

$$\begin{bmatrix} Q_1 \\ Q_2 \\ Q_3 \end{bmatrix} = [C] \begin{bmatrix} 1 \\ 0 \\ 0 \end{bmatrix} = \begin{bmatrix} C_{10} + C_{12} + C_{13} \\ -C_{12} \\ -C_{13} \end{bmatrix}$$

The diagonal elements in the matrix (such as $C_{(1,1)}$) are the sum of all capacitances from one conductor to all other conductors. These terms represent the self-capacitance of the conductors. Each is numerically equal to the charge on a conductor when one volt is applied to that conductor and the other conductors (including ground) are set to zero volts. For instance, for

$$C_{(1,1)} = C_{10} + C_{12} + C_{13}$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the off-diagonal terms in each column (such as $C_{(1,2)}$, $C_{(1,3)}$) are numerically equal to the charges induced on other conductors in the system when one volt is applied to that conductor. For instance, in column one of the example capacitance matrix, $C_{(1,2)}$ is equal to $-C_{12}$. This is equal to the charge induced on Conductor 2 when one volt is applied to Conductor 1 and zero volts are applied to Conductor 2.

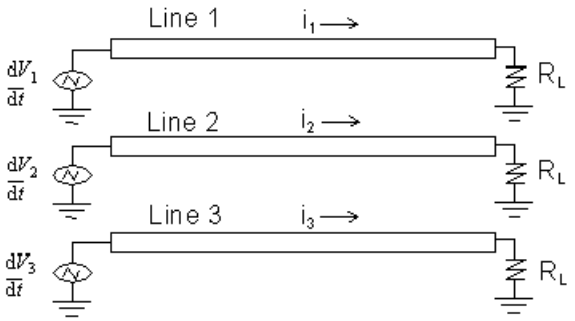
The off-diagonal terms are the negative values of the capacitances between the corresponding conductors (the mutual capacitances). In column one of the example capacitance matrix, the off-diagonal terms represent the capacitances between Conductor 1 and the other two conductors; in column two, the terms represent the capacitance between Conductor 2 and the other conductors; and so forth.

Note that the capacitance matrix is symmetric about the diagonal. This indicates that the mutual effects between any two objects are identical. For instance, $C_{(1,3)}$, the capacitance between Conductor 1 and Conductor 3 ($-C_{13}$), is equal to $C_{(3,1)}$, the capacitance between Conductor 3 and Conductor 1.

Capacitance in Terms of Currents and Time Varying Voltages

A capacitance matrix can also represent the relationship between currents and time varying voltages in a system of conductors.

Given the three transmission lines shown here,



the currents caused by the time varying voltage source on each line are given by the following relationship:

$$\begin{bmatrix} i_1 \\ i_2 \\ i_3 \end{bmatrix} = \begin{bmatrix} C_{10}+C_{12}+C_{13} & -C_{12} & -C_{13} \\ -C_{12} & C_{20}+C_{12}+C_{23} & -C_{23} \\ -C_{13} & -C_{23} & C_{30}+C_{13}+C_{23} \end{bmatrix} \begin{bmatrix} \frac{dV_1}{dt} \\ \frac{dV_2}{dt} \\ \frac{dV_3}{dt} \end{bmatrix}$$

If dV_2/dt and dV_3/dt are set to zero, this relationship becomes

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\begin{bmatrix} i_1 \\ i_2 \\ i_3 \end{bmatrix} = [C] \begin{bmatrix} \frac{dV_1}{dt} \\ 0 \\ 0 \end{bmatrix} = \begin{bmatrix} C_{10} + C_{12} + C_{13} \\ -C_{12} \\ -C_{13} \end{bmatrix} \frac{dV_1}{dt}$$

This gives the currents that are induced on Line 2 and Line 3 when a time varying voltage source is applied to Line 1 — that is, the capacitive coupling between the three lines, or the short circuit capacitance.

Computing Capacitance

To compute a capacitance matrix for a structure, Maxwell performs a sequence of electrostatic field simulations. In each field simulation, one volt is applied to a single conductor and zero volts is applied to all other conductors. Therefore, for an *n*-conductor system, *n* field simulations are automatically performed.

The energy stored in the electric field associated with the capacitance between two conductors is given by the following relation:

$$W_{ij} = \frac{1}{2} \int_{\Omega} D_i \bullet E_j d\Omega$$

where

- W_{ij} is the energy in the electric field associated with flux lines that connect charges on conductor *i* to those on conductor *j*.
- D_i is the electric flux density associated with the case in which one volt is placed on conductor *i*.
- E_j is the electric field associated with the case in which one volt is placed on conductor *j*.

Therefore, the capacitance between conductors *i* and *j* is

$$C = \frac{2W_{ij}}{v^2} = \int_{\Omega} D_i \bullet E_j d\Omega$$

Virtual Forces (Electrostatic)

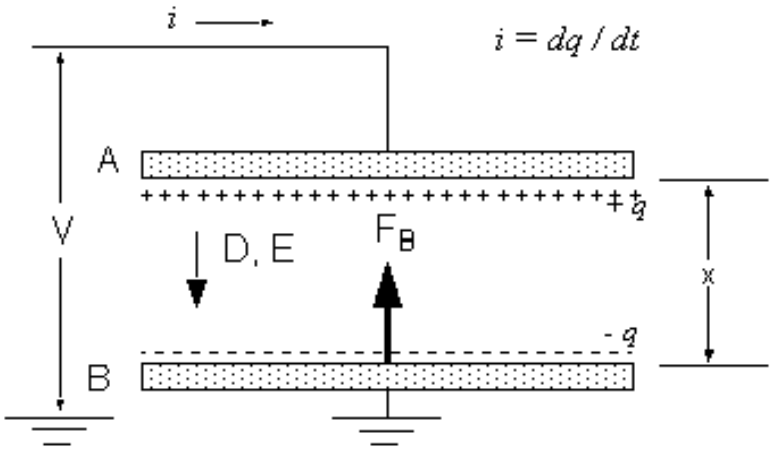
To compute the virtual force on an object, the electrostatic field simulator uses the principle of virtual work. In the structure shown below, the force on the bottom plate (plate B) in the direction of the displacement, *x*, is given by the following relationship:

$$F_B = \left. \frac{dW(v,x)}{dx} \right|_{V=\text{Constant}}$$

where *W* is the stored energy of the system,

$$W(v,x) = \frac{1}{2} \int_{Vol} \boldsymbol{E} \bullet \boldsymbol{D} dVol$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Unlike the classical virtual work method, the plate is not actually moved during the numerical process of the force computation. Instead, only the triangles that lie along the outside surface of the object are virtually distorted. *W* and its derivative, dW/dx, are calculated from a single field solution using finite element interpolation functions.

The virtual work method is only valid for objects surrounded by the materials air or vacuum. In the electrostatic solver, a threshold of tolerance = 1.e-7 (compared to the permittivity of vacuum/air) is used to determine if the material is vacuum/air. When an object is surrounded by materials other than air or vacuum the Maxwell stress tensor method is applied to calculate the force acting on the object.

Virtual Torques (Electrostatic)

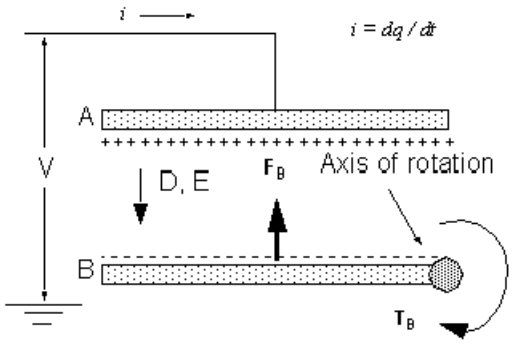
Similar to the virtual force calculation, the system uses virtual work principles to compute the torque on an object. In the structure shown below, the virtual torque on the bottom plate (plate B) about the axis of rotation is given by the following relationship:

$$T_B = \frac{dW(v,\theta)}{d\theta}\Big|_{V=Constant}$$

where *W* is the stored energy of the system,

$$W(v,\theta) = \frac{1}{2}\int_{Vol} \boldsymbol{E} \bullet \boldsymbol{D} dVol$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Unlike the classical virtual work method, the plate is not actually rotated during the numerical process of the torque computation. Instead, only the triangles that lie along the outside surface of the object are virtually distorted. *W* and its derivative, $dW/d\theta$, are calculated from a single field solution using finite element interpolation functions.

Flux Linkage (Electrostatic)

To compute the electric flux linkage, the electrostatic field solver uses the following relationship:

$$\Psi = \int E \bullet dA$$

where *E* is the electric field and *A* is the area over which flux density is computed.

- In Cartesian (XY) models, the area is found by sweeping the flux line you've drawn in the xy-plane into the z direction — forming a 3D surface. The electric flux value computed is the flux per meter depth in the z direction.
- In axisymmetric (RZ) models, the area is found by rotating the flux line you've drawn in the rz-plane 360 degrees about the z axis. The electric flux computed is the total flux that passes through this surface.

A separate flux linkage value is computed for each line you draw.

Magnetostatic Field Simulation

The magnetostatic field simulator lets you compute static magnetic fields arising from DC currents and other sources like permanent magnets and external magnetic fields. Magnetic fields in both linear and nonlinear materials can be simulated.

Magnetostatic Theory

The magnetostatic field simulator solves for the magnetic vector potential, $\mathbf{A}_Z(x,y)$ in this field equation:

$$J_z\left(x,y\right) = \nabla \times \left(\frac{1}{\mu_r \mu_0}\left(\nabla \times A_z\left(x,y\right)\right)\right)$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where

- $A_z(x,y)$ is the z component of the magnetic vector potential.
- $J_z(x,y)$ is the DC current density field flowing in the direction of transmission.
- μ_r is the relative permeability of each material.
- μ_0 is the permeability of free space.

Given $J_z(x,y)$ as an excitation, the magnetostatic field simulator computes the magnetic vector potential at all points in space.

> **Note:** In general, both **J** and **A** are vectors. However, **J** is assumed to only have a z-component. A consequence of this is that **A** only has a z-component as well. Both quantities can therefore be treated as scalars.

The equation that the magnetostatic field solver computes is derived from Ampere's law, which is

$$\nabla \times H = J$$

and from Maxwell's equation, $\nabla \cdot \mathbf{B} = 0$.

Because $H = \mathbf{B}/\mu_r\mu_0$, then

$$\nabla \times \left(\frac{B}{\mu_r \mu_0}\right) = J$$

Because $\mathbf{B} = \nabla \times A$, due to $\nabla \cdot \mathbf{B} = 0$, then

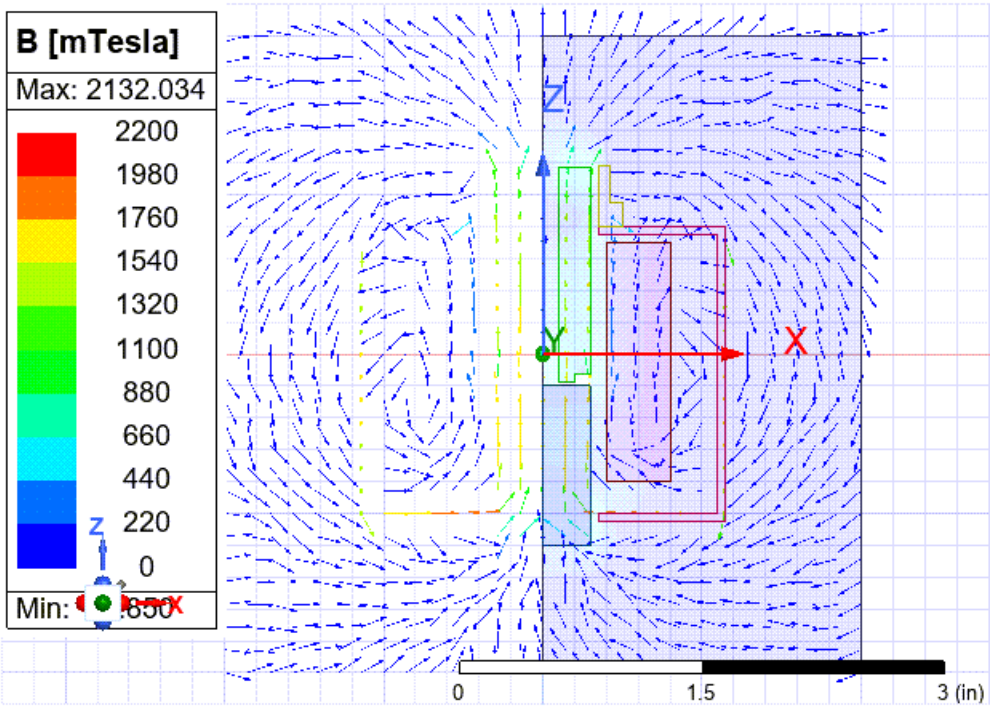
$$\nabla \times \left(\frac{1}{\mu_r \mu_0} \nabla \times A\right) = J$$

The magnetostatic field simulator solves this equation using the finite element method.

After $\mathbf{A}_z(x,y)$ is computed, the magnetic flux density, **B**, and the magnetic field, **H**, can then be computed using the relationships $\mathbf{B} = \nabla \times A$, and **H** for linear materials is $\mathbf{H} = \mathbf{B}/\mu_r\mu_0$.

Both **B** and **H** lie in the xy cross-section being analyzed. An arrow plot of a B-field generated by the magnetostatic field simulator is shown below:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Inductance

At the simplest level, inductance relates to how much energy is stored in the magnetic field when current flows:

$$W_m = \frac{1}{2}Li^2$$

where:

- W_m is the energy stored in the magnetic field
- L is the inductance
- i is the current flowing in the circuit

The Maxwell computes inductances associated with a structure by simulating the magnetic field that arises when various voltages and currents are applied. Then, by computing the energy stored in those fields, it can then compute the necessary inductances:

$$L = \frac{2W_m}{i^2}$$

To compute inductances using this method, the B-field and H-field associated with a distribution of currents must first be computed. The magnetostatic field simulator, which computes the magnetic vector potential at all points in the problem region, performs this task.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

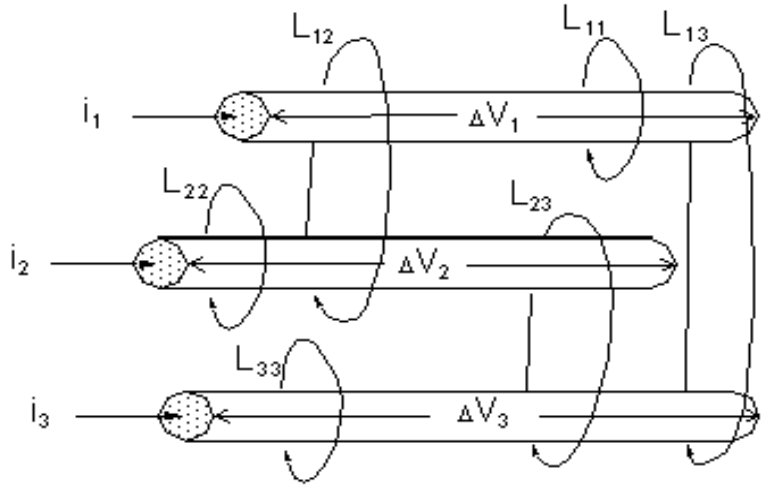
Inductance in Terms of Flux Linkage and Currents

An inductance matrix represents the magnetic flux linkage between the current loops in a system. Given the three current loops below, the relationship between induced flux and currents is as follows:

$$\lambda_1 = L_{11}\, i_1 + L_{12}\, i_2 + L_{13}\, i_3$$

$$\lambda_2 = L_{12}\, i_1 + L_{22}\, i_2 + L_{23}\, i_3$$

$$\lambda_3 = L_{13}\, i_1 + L_{23}\, i_2 + L_{33}\, i_3$$



This can be expressed in matrix form as

$$\begin{bmatrix} \lambda_1 \\ \lambda_2 \\ \lambda_3 \end{bmatrix} = \begin{bmatrix} L_{11} & L_{12} & L_{13} \\ L_{12} & L_{22} & L_{23} \\ L_{13} & L_{23} & L_{33} \end{bmatrix} \begin{bmatrix} i_1 \\ i_2 \\ i_3 \end{bmatrix}$$

The inductance matrix above gives the relationship between λ and *i* for the three independent current loops. In a device with *n* current loops, this relationship would be expressed by an $n \times n$ inductance matrix. Inductance matrix values are specified in henries.

If one ampere is applied to Current Loop 1 and zero amperes are applied to the other two loops, the inductance matrix becomes

$$\begin{bmatrix} \lambda_1 \\ \lambda_2 \\ \lambda_3 \end{bmatrix} = [L] \begin{bmatrix} 1 \\ 0 \\ 0 \end{bmatrix} = \begin{bmatrix} L_{11} \\ L_{12} \\ L_{13} \end{bmatrix}$$

The diagonal terms in the matrix (such as L_{11}) represent the self-inductance of each current loop. Self-inductance is numerically equal to the flux linkage in a current loop when one ampere

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is flowing in it, and no current is flowing in the other loops. For example, L_{11} is equal to the flux in Current Loop 1 when one ampere is flowing in that current loop, and no current is flowing in the other loops.

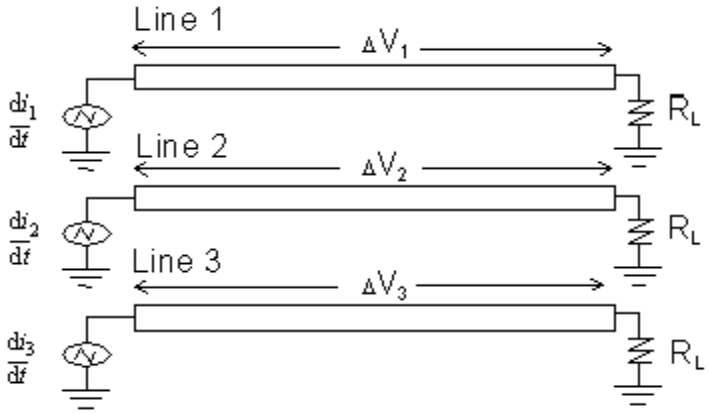
The off-diagonal terms (such as L_{12}, L_{13}) represent the mutual inductances between the current loops. Mutual inductance is numerically equal to the flux linkage in a current loop when one ampere is flowing through another loop, and no current is flowing anywhere else. For example, L_{12} is equal to the flux linkage in Loop1 when one ampere is applied to Loop 2 and no current is flowing in the other loops.

Note that the inductance matrix is symmetric about the diagonal. This indicates that the mutual effects between any two loops are identical. For instance, L_{13}, the inductance between Current Loop 1 and Current Loop 3, is equal to the inductance between Current Loop 3 and Current Loop 1.

Inductance in Terms of Voltages and Time Varying Currents

An inductance matrix can also represent the relationship between voltage and current fluctuations in a system. Given the three transmission lines shown below, the voltage changes caused by the time varying current source on each line are given by:

$$\begin{bmatrix} \Delta V_1 \\ \Delta V_2 \\ \Delta V_3 \end{bmatrix} = \begin{bmatrix} L_{11} & L_{12} & L_{13} \\ L_{12} & L_{22} & L_{23} \\ L_{13} & L_{23} & L_{33} \end{bmatrix} \begin{bmatrix} (di_1)/(dt) \\ (di_2)/(dt) \\ (di_3)/(dt) \end{bmatrix}$$



The inductance matrix above gives the relationship between ΔV and *di/dt* for the three independent transmission lines.

If di_2/dt and di_3/dt are set to zero, this relationship becomes

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\begin{bmatrix} \Delta V_1 \\ \Delta V_2 \\ \Delta V_3 \end{bmatrix} = [L] \begin{bmatrix} (di_1)/(dt) \\ 0 \\ 0 \end{bmatrix} = \begin{bmatrix} L_{11} \\ L_{12} \\ L_{13} \end{bmatrix} (di_1)/(dt)$$

This gives the voltage changes that are induced on Lines 2 and 3 when a time-varying current source is applied to Line 1 — that is, the inductive coupling between all the loops.

Computing an Inductance Matrix

To compute an inductance matrix, the software performs a sequence of magnetostatic field simulations. In each field simulation, one ampere is allowed to flow in a single conductor. The current returns as defined under **Maxwell 2D** > **Parameters** > **Assign** > **Matrix** — either in the conductor you identified as the return path, or along outside balloon, value (Dirichlet) or odd symmetry boundaries. No current flows in any other conductor.

For an n-conductor system, *n* field simulations are automatically performed. The energy stored in the magnetic field that couples two conductors is

$$W_{ij} = \frac{1}{2} L I^2 = \frac{1}{2} \int_{\Omega} B_i \bullet H_j d\Omega$$

where

- W_{ij} is the energy stored in the magnetic field linking conductor *i* with conductor *j*.
- *I* is the current in conductor *i*.
- B_i is the magnetic flux density where one ampere is allowed to flow through conductor *i*.
- H_j is the magnetic field where one ampere is allowed to flow through conductor *j*.

The inductance coupling conductors *i* and *j* is therefore

$$L_{ij} = \frac{2W_{ij}}{I^2} = \int_{\Omega} B_i \bullet H_j d\Omega$$

For multi-turn conductors, the net value of inductance is the value given by

$$L_{net} = N^2 L_{matrix}$$

where *N* is the number of turns in the coil.

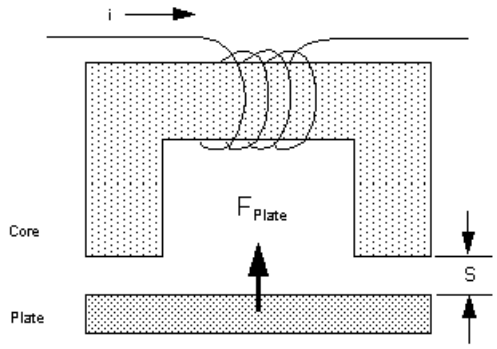
Virtual Forces (Magnetostatic)

To compute the force on an object, the system uses the principle of virtual work. In the structure shown below, the force on the plate in the direction of the displacement, *s*, is given by the following relationship:

$$\boldsymbol{F}_{plate} = \frac{dW(s,i)}{ds}\bigg|_{i = const} = \frac{\partial}{\partial s} \left[\int_V \left(\int_0^H \boldsymbol{B} \bullet d\boldsymbol{H} \right) dV \right]$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where $W_{(s,i)}$ is the magnetic coenergy of the system. The current, *i*, is held constant.



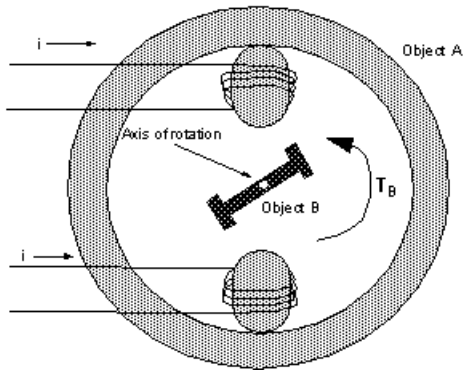
Unlike the classical virtual work method, the plate is not actually moved during the force computation. Instead, only the triangles that lie along the outside surface of the object are virtually distorted. Thus, the force computation only requires one field solution.

Virtual Torques (Magnetostatic)

Similar to the virtual force calculation, the system uses virtual work principles to compute the torque on an object. In the structure shown below, the torque on Object B about the axis of rotation is given by the following relationship:

$$T_B = \frac{dW(\theta,i)}{d\theta}\Big|_{i=const} = \frac{\partial}{\partial\theta}\left[\int_V \left(\int_0^H \boldsymbol{B} \bullet d\boldsymbol{H}\right) dV\right]$$

where $W(\theta, i)$ is the magnetic coenergy of the system. The current, *i*, is held constant.



Unlike the classical virtual work method, Object B is not actually rotated during the torque computation. Instead, only the triangles that lie along the outside surface of the object are virtually distorted. Thus, the change in the system's coenergy (and therefore the virtual torque) is given by the change in the coenergy of these triangles.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Flux Linkage (Magnetostatic)

To compute the magnetic flux linkage, the magnetostatic field solver uses the following relationship:

$$\Psi_i = L_{ii} I_i + \left(\sum_{j=1}^{n} L_{ij} I_j \right) + \Psi_m$$

where

$L_{ii} I_i$ defines the flux linkage due to self-inductance in a coil,

$\sum_{j=1}^{n} L_{ij} I_j$ is the flux linkage due to mutual inductance between coils (i ≠ j), and

Ψ_m is the permanent magnetic flux linkage.

The energy method is used to calculate the inductance of a winding, which is then multiplied by the current flowing in the winding to get the flux linkage.

To obtain a correct flux vector, all current-carrying coils must be made part of the L-matrix computation. If you leave out L-entries that belong to current-carrying coils, the fluxes will be incorrect. (Refer to Inductance in Terms of Flux Linkage and Currents for additional information.)

> **Note:** Maxwell no longer uses the following computation algorithm:
> $\Psi = \int B \bullet dA$

AC Magnetic Field Simulation

The AC Magnetic (Eddy Current) field simulator allows you to simulate the effects of time-varying currents in parallel-conductor structures – including eddy current effects in conductors.

On nonlinear systems, the AC Magnetic solver can be used as an alternative to the transient solver to solve for harmonic losses.

AC Magnetic Solver Theory

Time-varying currents flowing in a conductor produce a time-varying magnetic field in planes perpendicular to the conductor. In turn, this magnetic field induces eddy currents in the source conductor and in any other conductor parallel to it. The AC Magnetic (Eddy Current) field solver calculates the eddy currents by solving for **A** and Φ in the field equation:

$$\nabla \times \frac{1}{\mu} (\nabla \times \mathbf{A}) = (\sigma + j\omega\varepsilon)(-j\omega A - \nabla\Phi)$$

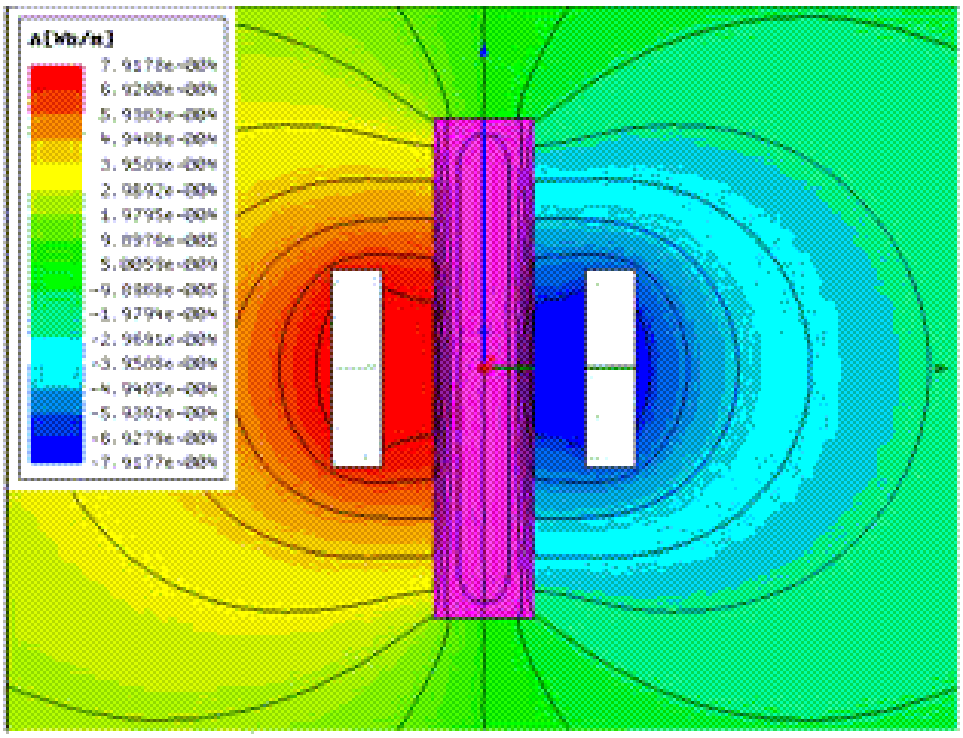
where

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- A is the magnetic vector potential.
- Φ is the electric scalar potential.
- μ is the absolute magnetic permeability.
- ω is the angular frequency at which all quantities are oscillating.
- σ is the conductivity.
- ε is the absolute permittivity.

> **Note:** The AC Magnetic equation is derived from Maxwell's equations. Phasor notation is used to represent complex quantities.

A plot of flux lines produced by eddy currents that were computed in a structure by the AC Magnetic solver is shown below:



Components of Current Density

Notice that the right side of the equation

$$\nabla \times \frac{1}{\mu}(\nabla \times \mathbf{A}) = (\sigma + j\omega\varepsilon)(-j\omega\boldsymbol{A} - \nabla\Phi)$$

consists of a complex conductivity multiplied by the complex value of **E** ($-j\omega A - \nabla\Phi$). Therefore, it is equal to the complex current density, **J**, which has three components:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- $\mathbf{J_s}$, the source current density due to differences in electric potential, $-\sigma\nabla\Phi$.
- $\mathbf{J_e}$, the induced eddy current density due to time-varying magnetic fields, - $j\omega\sigma A$.
- $\mathbf{J_d}$, the displacement current density (time-varying electric fields), $j\omega\varepsilon(-j\omega A-\nabla\Phi)$.

The total current density is the sum of these three components. The $j\omega$ term in the eddy and displacement components indicate that they are a function of frequency and become increasingly significant as the frequency increases.

Integrating the Current Density

When setting up a problem, you specify the total current flowing in any conductor that is connected to an external source. Therefore, the AC Magnetic module is able to make use of a second equation:

$$I_T = \int_{\Omega}(\sigma + j\omega\varepsilon)(-j\omega\boldsymbol{A} - \nabla\phi)d\Omega$$

which reflects the fact the total current in a conductor equals the integral of the current density over the cross-section of the conductor, Ω.

AC Magnetic Assumptions

The AC Magnetic (Eddy Current) field solver makes the following assumptions about the field quantities for which it solves:

- Time-varying electromagnetic quantities are assumed to have the periodic waveform:

$$F(t) = F_m \cos(\omega t + \theta)$$

- All quantities must have the same frequency (angular frequency), but can have different phase angles (θ). If a current is not a pure sinusoid, decompose it into sinusoidal harmonics, and solve separately at each frequency, but only for linear problems.
- All currents (source, eddy, and displacement) are assumed to flow perpendicular to the plane being studied (that is, in the z-direction for XY applications). Therefore, the magnetic fields associated with these currents lie within the xy-plane. As a result, **A**, the magnetic vector potential, has a z-component only.
- Because no currents flow in the xy-plane, the electric field, **E**, has a z-component only. It follows that ϕ is constant over the cross-section of each conductor in the problem.

Deriving the AC Magnetic Equation

The AC Magnetic (Eddy Current) field solver uses the finite element method to compute **A** and Φ using these two relationships:

$$\nabla \times \tfrac{1}{\mu}(\nabla \times \mathbf{A}) = (\sigma + j\omega\varepsilon)(-j\omega\boldsymbol{A} - \nabla\Phi)$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$I_T = \int_{\Omega}(\sigma + j\omega\varepsilon)(-j\omega\boldsymbol{A} - \nabla\Phi)d\Omega$$

where

- A is the magnetic vector potential.
- Φ is the electric scalar potential.
- μ is the magnetic permeability.
- ω is the angular frequency at which all quantities are oscillating.
- σ is the conductivity.
- ε is the permittivity.
- I_T is the total current flowing in conductors.

The following section shows how these equations are derived from Maxwell's equations.

Maxwell's Equations

The AC magnetic field simulator solves for time harmonic electromagnetic fields governed by Maxwell's equations:

$$\nabla \times \boldsymbol{H} = \boldsymbol{J} + \frac{\partial \boldsymbol{D}}{\partial t}$$

$$\nabla \times \boldsymbol{E} = -\frac{\partial \boldsymbol{B}}{\partial t}$$

$$\nabla \bullet D = \rho$$

$$\nabla \bullet B = 0$$

where

- E is the electric field.
- D is the electric displacement, εE.
- B is the magnetic flux density.
- H is the magnetic field intensity, B/μ.
- J is the conduction current density, σE.
- ρ is the charge density.

The AC magnetic solver assumes that all time-varying electromagnetic quantities in the problem have the form

$$\boldsymbol{F}(t) = \boldsymbol{F}_m \cos(\omega t + \theta)$$

Using Euler's formula,

$$e^{j\alpha} = \cos\alpha + j\sin\alpha$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

if $\alpha = \omega t+\theta$, $F(t)$ equals the real portion of $e^{j(\omega t+\theta)}$:

$$F(t) = \Re\left[F_m e^{j(\omega t+\theta)}\right] = \Re\left[F_m(\cos(\omega t+\theta)+j\sin(\omega t+\theta))\right] = F_m\cos(\omega t+\theta)$$

Now, because each time-varying quantity has the form $F_m e^{j\theta} e^{j\omega t}$, $\partial D/\partial t$ and $\partial B/\partial t$ are equal to $j\omega D$ and $j\omega B$.

Therefore, with this simplification and the relations $H = \mu B$, $D = \varepsilon E$, and $J = \sigma E$, Maxwell's equations reduce to

$$\nabla \times \frac{1}{\mu} B = \left(\sigma E + j\omega\varepsilon E\right)$$

$$\nabla \times E = -j\omega B$$

$$\nabla \bullet \varepsilon E = \rho$$

$$\nabla \bullet B = 0$$

Relationship of Magnetic and Electric Field

The quantity that the AC magnetic field simulator actually solves for is A, the magnetic vector potential. It is given by

$$\nabla \times A = B$$

Substituting this into the first of Maxwell's equations, the result is

$$\nabla \times \frac{1}{\mu}(\nabla \times A) = \left(\sigma \mathrm{E} + j\omega\varepsilon \mathrm{E}\right)$$

A solution for E in terms of A is given by

$$E = -j\omega A - \nabla\Phi$$

where Φ is the electric potential. Substituting the right side of this relationship for E into the previous equation results in

$$\nabla \times \frac{1}{\mu}(\nabla \times A) = \left(-j\omega A - \nabla\Phi\right)\left(\sigma + j\omega\varepsilon\right)$$

This equation is one of the two used by the AC magnetic solver to compute A and Φ.

Relationship of Current and Current Density

Notice that the previous equation is in the form of a complex conductivity, $(\sigma+j\omega\varepsilon)$, times the complex value of E. The result is the complex current density, J. Therefore, the integral of this expression over the cross-section of a conductor is constrained to equal the total current that is specified as flowing in that conductor:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$I_t = \int_{\Omega} J d\Omega = \int_{\omega} \frac{1}{\mu} (\sigma + j\omega\varepsilon)(-j\omega A - \nabla\Phi) d\omega$$

This is the second equation that the AC magnetic solver uses in computing *A* and Φ.

The total current, I_t, in this equation is the quantity you specify when setting up a problem. It is the total current flowing in a conductor, and includes

- I_s, the current from an external source,
  $-\int_{\Omega} \sigma \nabla\Phi d\Omega$
- I_e, the induced eddy current,
  $-\int_{\Omega} j\omega\sigma \mathbf{A} d\Omega$
- I_d, the displacement current,
  $\int_{\Omega} j\omega\varepsilon(-j\omega\mathbf{A} - \nabla\Phi) d\Omega$

For problems solved by the AC magnetic solver, *B* is assumed to lie in the xy plane. Therefore, *A* can only have a component in the z-direction. The simulator does not have to solve for the x and y components of *A*. Because *E* only has a z-component, Φ is a constant for each cross-section of a conductor. Therefore, Maxwell does not have to solve for Φ at every node.

> **Note:** For a more detailed discussion of eddy current and skin effect problems in multiconductor systems, see J. Weiss, Z. Cendes. “A One-Step Finite Element Method for Multiconductor Skin Effect Problems”, *IEEE Transactions on Power Apparatus and Systems*, Vol. PAS-101, No. 10 October 1982.

Eddy Currents and Skin Depth

Induced currents allow magnetic fields to penetrate conductors only to a certain depth, which is approximated by the formula

$$\delta = \sqrt{\frac{2}{\omega\sigma\mu_0\mu_r}} \quad \left(\text{in meters}\right)$$

where

- ω is the angular frequency, which is equal to $2\pi f$. (*f* is the frequency at which source currents and voltages oscillate during the solution.)
- σ is the conductor’s conductivity, in siemens/meter.
- μ_r is the conductor’s relative permeability, in amperes/meter.
- μ_0 is the permeability of free space, which is equal to $4\pi \times 10^{-7}$ A/m.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

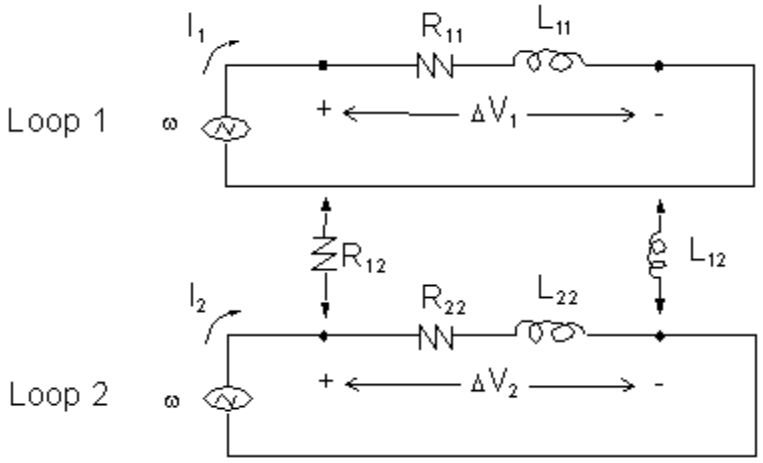
Currents will be concentrated near the surface of the conductor, decaying rapidly past the skin depth. As the formula above indicates, the skin depth gets smaller as the frequency increases.

Impedance Matrix

An impedance matrix summarizes the relationship between AC voltages and AC currents in multi-conductor systems. Given the two current loops below, the relationships between voltages and currents in each loop is as follows:

$$(\Delta V_1 = I_1 R_{11} + I_2 R_{12} + I_1 j\omega L_{11} + I_2 j\omega L_{12}) Size$$

$$\Delta V_2 = I_2 R_{22} + I_1 R_{12} + I_2 j\omega L_{22} + I_1 j\omega L_{12}$$



This can be expressed in matrix form as

$$\begin{bmatrix} \Delta V_1 \\ \Delta V_2 \end{bmatrix} = \begin{bmatrix} Z_{11} & Z_{12} \\ Z_{12} & Z_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix}$$

where

- V_i and I_i are phasors.
- *Z11* = $R_{11} + j\omega L_{11}$ (the self-impedance of Loop 1).
- *Z12* = $R_{12} + j\omega L_{12}$ (the mutual impedance between Loops 1 and 2).
- *Z22* = $R_{22} + j\omega L_{22}$ (the self-impedance of Loop 2).

The impedance matrix above gives the relationship between V and I for the two current loops. In a device with *n* current loops, this relationship would be expressed by an *n* x *n* impedance matrix. The matrix values displayed by the software are resistance and inductance (not resistance and reactance) and therefore do not include jω.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: All impedances are complex numbers in the form $Z = R + j\omega L$
where

- ω is equal to $2\pi f$, where f is the frequency of the AC current source.
- R is the AC resistance, given in ohms/meter (XY) or ohms (RZ).
- L is the AC inductance, given in henries/meter (XY) or henries (RZ).

Computing an Impedance Matrix

Maxwell breaks down the impedance matrix computation into two parts. First, it solves for the inductance matrix (L-matrix) associated with the model. It then solves for the resistance matrix (R-matrix). When it finishes solving for these matrices, the simulator combines them to form the impedance matrix, using the relationship $Z=R + j\omega L$.

To compute the inductance and resistance matrices for the impedance solution, the simulator generates an eddy-current field solution for each conductor in the matrix.

In the first solution, the current in the first conductor is set to one ampere; currents in the other conductors that are included in the impedance matrix are set to zero. This is done by imposing current sources on the conductors.

In the second solution, the current in the second conductor is set to one ampere, and all other conductors that are included in the impedance matrix are set to zero amperes, and so forth. Conductors that are not included in the impedance matrix are not affected.

Matrix Inductance

To compute the inductance of the current loop, the simulator calculates the average energy, W_{AV}, of the system after a field solution is computed:

$$W_{AV} = \frac{1}{4}\int_{V} B \bullet H^{*} dV$$

Because the instantaneous energy of the system is equal to $W_{Inst} = \frac{1}{2}Li^2$, where the instantaneous value of the current is related to the peak value of the current by $i = I_{peak}\cos(\omega t + \theta)$. The average value for the energy can then be found by integrating the instantaneous energy:

$$W_{AV} = \frac{1}{2\pi}\int_{0}^{2\pi} W_{Inst}\, d\omega t = \left(\frac{L}{2}\right)\left(\frac{1}{2\pi}\right)\int_{0}^{2\pi} I_{peak}^2 [\cos(\omega t + \theta)]^2 d\omega t$$

From this, the average energy of the system is equal to

$$W_{AV} = \left(\frac{L}{2}\right) I_{RMS}^2 = \left(\frac{L}{2}\right)\left(\frac{I_{peak}}{\sqrt{2}}\right)^2 = \left(\frac{L}{4}\right) I_{peak}^2$$

The inductance, therefore is

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$L = \frac{4W_{AV}}{I_{peak}^2}$$

The software assumes that the object for which impedance is being computed has a peak current of one ampere per coil turn flowing through it. Thus, the inductance is $4W_{AV}$.

The AC Magnetic simulator includes the eddy effect in conductors when it computes inductance during an impedance solution.

Matrix Resistance

To compute the resistance, the simulator calculates the ohmic loss, *P*, after a field solution has been computed:

$$P = \frac{1}{2\sigma}\int_V J \bullet J^* dV$$

The ohmic loss is related to the resistance by

$$P = RI_{RMS}^2$$

The resistance is therefore

$$R = \frac{P}{I_{RMS}^2} = \frac{2P}{I_{peak}^2}$$

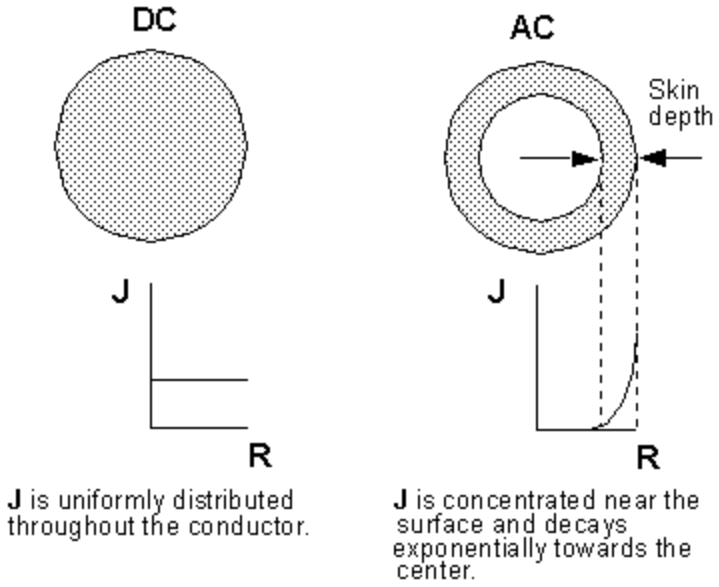
The system assumes that the object for which impedance is being computed has a peak current of one ampere per coil turn flowing through it. Therefore, the resistance is 2*P*.

Note that the resistance for an AC Magnetic problem will be higher than the equivalent DC resistance, due to the skin concentration of currents.

Inductance and Resistance in Impedance Computations

The inductances and resistances computed during an impedance matrix solution are different from those computed for the equivalent DC case. This figure shows how they differ:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The current density, *J*, in the DC case (the conductor on the left) is evenly distributed throughout the cross-section of the conductor. The current density in the AC case (the conductor on the right) is distributed nonuniformly on the surface due to skin effect. Because the area through which current can flow is smaller, it follows that the resistance to the current flow is higher in the impedance matrix than in a resistance matrix computed for the equivalent DC case.

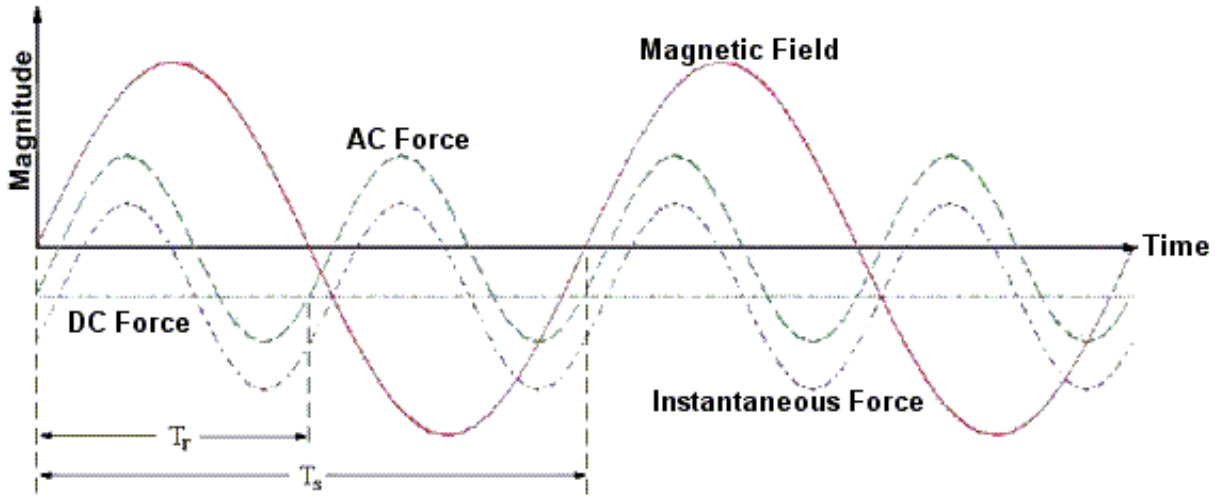
In the DC example, no eddy currents occur. The magnetic field created by the current flowing through the conductor is static. In the AC example, the oscillating magnetic field induces currents in conductors in the model. These induced currents affect the computation of inductance for the impedance matrix, causing it to be different from the equivalent DC computation of inductance.

Virtual Forces

Virtual force in an AC Magnetic (Eddy Current) problem is computed the same way as virtual force in a magnetostatic problem. The only difference is that the average value of force over time is computed — not the instantaneous force at a given time.

The difference between the time-averaged (or DC) force, AC force, and instantaneous force is shown below:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Force oscillates at twice the frequency of the source current and magnetic field:

$$f_F = \frac{1}{T_F} = 2f_S$$

where

- f_F is the frequency of the force.
- f_S is the frequency of the source current and magnetic field.
- T_F is the period of the force.

The time-averaged (or DC) force, AC force, and instantaneous force can be determined by

$$F_{DC} = \frac{1}{2}\int Re\left|\overline{J} \times \overleftarrow{\overline{B}}^{\circ}\right| dV$$

$$F_{AC} = \frac{1}{2}\int Re\left|\overline{J} \times \overline{B}\right| dV$$

$$F_{INST} = F_{DC} + F_{AC}$$

The AC force, F_{AC} must be evaluated at a particular phase (=ωt) in order to determine its magnitude at an instant in time. However, the peak value of the AC force is reported as the “AC Magnitude” in the force and torque solution panel for the Maxwell 2D solver.

Virtual Torques

Virtual torque in an AC Magnetic (Eddy Current) problem is computed the same way as virtual torque in a magnetostatic problem. The only difference is that the average value of the torque over time is computed, not the net torque at a given time.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Current Flow

To compute the current flow, the AC Magnetic (Eddy Current) field solver uses the following relationship:

$$I = \int J \bullet dA$$

where

- *I* is the current.
- *J* is the current density, given by

$$J = (\sigma + j\omega\varepsilon_r)(-j\omega A - \nabla\phi)$$

- *A* is the area over which the current flow is computed.
- In Cartesian (XY) models, the area is found by sweeping the current flow line you've drawn in the xy-plane into the z direction — forming a 3D surface. The current flow computed is the current per meter depth in the z-direction.
- In axisymmetric (RZ) models, the area is found by revolving the flux line you've drawn in the rz-plane 360 degrees around the z-axis, forming a 3D surface. The current flow computed is the total current that passes through this surface.

A separate current flow value is computed for each line you draw.

Nonlinear AC Magnetic Field Simulation

> **Note:** Applicable to both Maxwell 2D and 3D AC Magnetic solvers.

For designs with nonlinear materials, the Maxwell 2D and 3D AC Magnetic solvers allow you to analyze the fundamental components of *B* and *H* at a specified frequency.

Nonlinear AC Magnetic Theory

> **Note:** The information in this topic is applicable to both Maxwell 2D and 3D AC Magnetic solvers.

Unlike linear problems in which the field solution is assumed to be based on the sinusoidal complex peak field values in a steady state, nonlinear problems are usually based on all harmonic components of the fields.

That is to say for nonlinear problems, though the current is sinusoidal at its input, the resulting fields are "harmonically rich." Although the *B*- and *H*-fields are not sinusoidal at output, the Maxwell 2D and 3D nonlinear AC magnetic solvers use equivalent fundamental components of

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

B and *H* to approximate nonlinear field behavior associated with the assigned nonlinear B-H characteristic.

Sinusoidal B

> **Note:** Applicable to the Maxwell 2D AC Magnetic solver and the 3D AC Magnetic A-Phi solver.

Assuming both that *B* is sinusoidal with time, *t*, and that the value of *H* is derived from the original B-H curve, H_e, representing the effective magnetic field, can be calculated.

The derivation of the effective H_e is based on the equivalence of averaged energy between using original, non-sinusoidal *H* field and the effective sinusoidal field $H_e \sin(\omega t)$. That is,

$$\frac{4}{T}\int_0^{T/4} H_e \sin(\omega t) B_m \sin(\omega t) dt = \frac{4}{T}\int_0^{T/4} (H \cdot B) dt$$

with $T = 2\pi/\omega$, the above expression becomes

$$\int_0^{\pi/2} H_e B_m \sin^2(\omega t) d\omega t = \int_0^{\pi/2} (H \cdot B) d\omega$$

Allowing $\alpha = \omega t$ and extracting H_e, simplifying yields

$$H_e B_m \int_0^{\pi/2} \left(\frac{1-cos2\alpha}{2}\right) d\alpha = \int_0^{\pi/2} (H \cdot B) d\alpha$$

or

$$H_e = \frac{4}{\pi}\int_0^{\pi/2} \frac{(H \cdot B)}{B_m} d\alpha$$

where B_m corresponds to the last point on the original B-H characteristic (in the saturation portion) and which is made equal with $\pi/2$.

The above equation is used to derive an effective (B_m, H_e) characteristic for the frequency domain simulation. This process is required to be performed only once for a particular B-H curve. During the simulation, H_e is determined for a specific B_m such that an equivalent permeability can be computed as the ratio B_m / H_e and used in the calculations.

DC Conduction Field Simulation

The DC conduction field simulator allows you to analyze conduction currents due to static electric fields in conductors and lossy dielectrics.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DC Conduction Theory

When a material with a non-zero conductivity is subject to a potential difference, conduction current flows in the material. At all points in the problem space, the current density (*J*) will be proportional to the electric field (*E*) that is established due to the potential difference.

$$J(x,y) = \sigma E(x,y) = -\sigma \nabla \Phi(x,y)$$

where

- *J(x,y)* is the current density.
- *E(x,y)* is the electric field.
- σ is the conductivity of the material in MKS units (S/m).
- $\Phi(x,y)$ is the electric potential.

The equation that the DC conduction field simulator solves is based on the fact that, under steady state conditions, the amount of charge, ρ, leaving any infinitesimally small region must equal the charge flowing into that region:

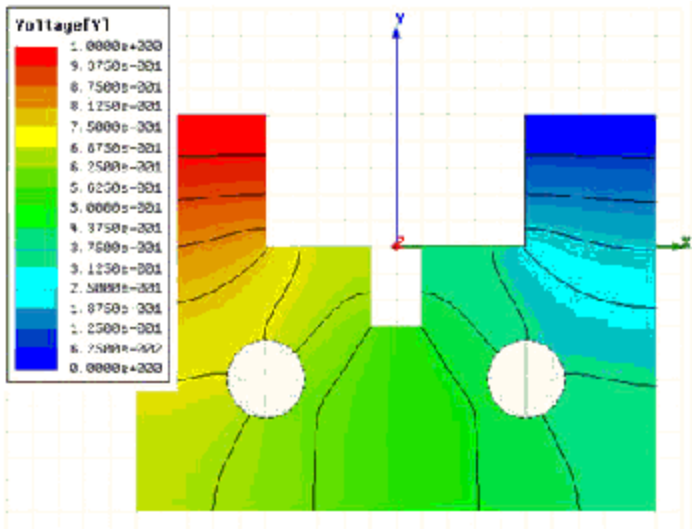
$$\nabla \bullet J = -\frac{\partial \rho}{\partial t} = 0$$

The field quantity that DC conduction actually solves for is the electric potential, Φ, in the following equation:

$$\nabla \bullet (\sigma \nabla \Phi) = 0$$

Note that $-\sigma \nabla \Phi = J$.

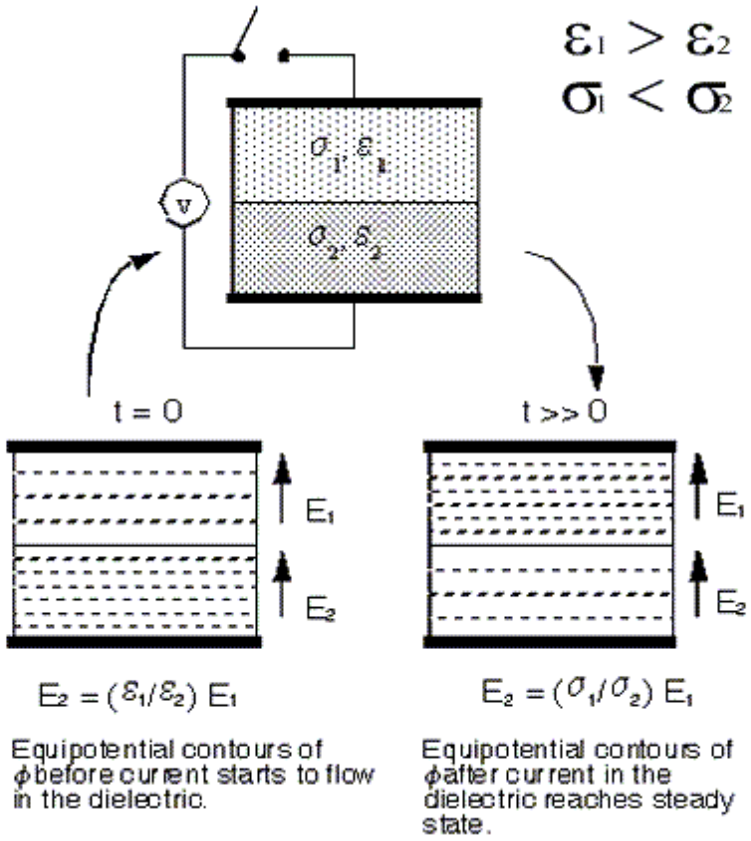
A plot of electric potential that was computed by the DC conduction solver is shown below:



Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Steady-State Conditions

The DC conduction solver assumes that current flow in the conducting material has already reached steady-state conditions. The implication of this assumption becomes clear when you compare the initial and steady-state conditions of a simple problem such as that shown here:



First, consider the interval before steady state is reached. Assume that the switch establishing the potential across the parallel plates in the figure above closes at t = 0. Also assume that the current required to deposit charges on the parallel plates (so that the voltage difference can be supported) occurs instantaneously.

However, it will take a while for current to start flowing in the dielectric. The time it takes for current to flow is determined by the time constant of the material. Therefore, at t = 0, there will be an electric field in the dielectric, but no current and no free charges. Consequently, the relationship that must be satisfied at the interface of the two dielectrics is $\nabla \bullet D = \rho$ or, equivalently, $\nabla \bullet \varepsilon E = \rho$.

At the interface between the two dielectrics, this relationship implies that

$$\varepsilon_1 E_{n1} = \varepsilon_2 E_{n2} \Longrightarrow E_{n1} = \frac{\varepsilon_2}{\varepsilon_1} E_{n2}$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where E_n is the normal component of E. Therefore, at t = 0, before current starts to flow in the two dielectrics, $\Phi(x,y)$ is determined entirely by the permittivity of the dielectrics. Use the electrostatic field solver to solve for $\Phi(x,y)$ in such a case.

After current starts to flow in the dielectric and steady state is reached, free charges are able to accumulate at the boundaries of the two dielectrics. The free charge, ρ, is no longer zero.

The relationship that must be satisfied now is

$$\nabla \bullet J = -\frac{\partial \rho}{\partial t} = 0$$

or, equivalently, $\nabla \bullet \sigma E = 0$.

This relationship implies that

$$\sigma_1 E_{n1} = \sigma_2 E_{n2} \Longrightarrow E_{n1} = \frac{\sigma_2}{\sigma_1} E_{n2}$$

In this case, the solution depends on the conductivity (σ) of the materials rather than on their dielectric constant (ε). The DC conduction solver analyzes the steady state condition. Therefore, use the DC conduction field solver when steady state conditions have been reached.

Relevant Time Constant

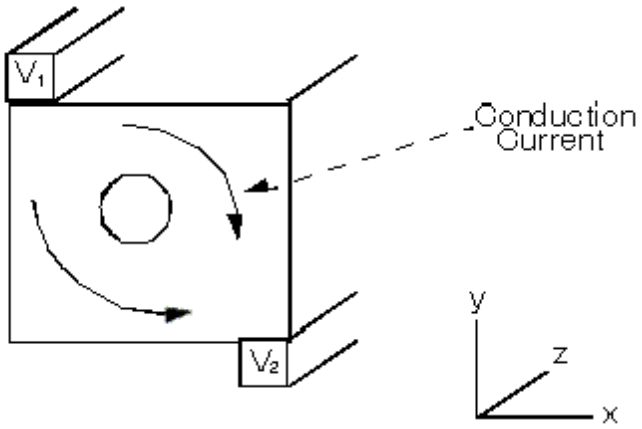
The time constant that determines how long it will take the current flowing in the conductive material to reach steady state is given by $\tau = \varepsilon/\sigma$. In a good conductor such as steel ($\varepsilon = 9 \times 10^{-12}$, $\sigma = 1 \times 10^{7}$), $\tau = 9 \times 10^{-19}$ seconds. Steady state conditions are reached almost instantaneously and in such cases you should use the DC conduction solver to analyze the potential field.

On the other hand, for a good dielectric such as fused quartz ($\sigma = 1 \times 10^{-17}$), the time constant is on the order of 10^5 seconds. There will be a significant period before steady state currents start to flow and you should use the electrostatic solver to analyze the electric field. For many materials in between, a complete analysis may require an electric transient field solution.

Conductance

A conductance matrix gives the relationship between currents and voltage drops. In the following figure, two conductors at voltages V_1 and V_2 are touching a bar with a conductance, G.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Because the bar is not a perfect insulator, current will flow between the two conductors. Given the conductors shown on the previous page, the relationship between the conduction current and the voltage drop, V, in each conductor is $I = GV$.

where

- I is the conduction current,
- G is the conductance, measured in Siemens, and
- V is the voltage drop given by V_1-V_2.

Conductance, as evident from the equation, is the inverse of resistance. Therefore, if a material has a large conductance, it will be a better conductor, and if it has a low conductance, it will be a better resistor.

Current Flow (DC Conduction)

To compute current flow, the DC conduction field solver uses the following relationship:

$$I = \int_A J \bullet dA$$

where

- I is the current.
- J is the current density, given by $J = \sigma E = -\sigma \nabla \Phi$.
- A is the area over which the current flow is computed.
- In Cartesian (XY) models, the area is found by sweeping the current flow line you've drawn in the xy-plane into the z direction — forming a 3D surface. The current flow computed is the current per meter depth in the z direction.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- In axisymmetric (RZ) models, the area is found by revolving the flux line you've drawn in the rz-plane 360 degrees around the z-axis, forming a 3D surface. The current flow computed is the total current that passes through this surface.

A separate current flow value can be computed for each line you draw.

AC Conduction Field Simulation

The AC conduction field solver allows you to analyze conduction currents caused by time-varying electric fields in conductors and lossy dielectrics.

AC Conduction Theory

The AC conduction field simulator solves for Φ in the following equation:

$$\nabla\bullet[\sigma E + j\omega\varepsilon\nabla\Phi(x,y)] = 0$$

where

- $\Phi(x,y)$ is the magnitude and phase of the electric potential at each value of x and y,
- ω is the angular frequency at which the potential is oscillating,
- σ is the conductivity, and
- ε is the permittivity.

The above equation is derived from $\nabla\bullet(J + j\omega D) = 0$

where

- J is the current density, σE,
- D is the electric flux density, εE, and
- E is the electric field, $-\nabla\Phi$.

Complex quantities are represented using phasor notation.

AC Conduction Assumptions

The AC conduction field solver assumes the following conditions about field quantities:

- All time-varying electromagnetic quantities have the periodic waveform:

$$\boldsymbol{F}(t) = \boldsymbol{F_m}\cos(\omega t + \theta)$$

  where all quantities must have the same value of ω, but can have different phase angles (θ).

- If a current is not a pure sinusoid, it is decomposed into sinusoidal harmonics, and solved separately at each frequency.
- The component of **E** due to time-varying magnetic fields caused by conduction currents can be neglected.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Admittance

Admittance can best be explained as the inverse of impedance, and is expressed by $Y = G + j\omega C$,

where

- ω is equal to $2\pi f$, where f is the frequency of the AC voltage source,
- Y is the admittance in Siemens/meter,
- G is the conductance in Siemens/meter, and
- ωC is the susceptance in Siemens/meter.

> **Note:** The matrix values displayed by the software are conductance in ohms/meter and capacitance in farads/meter (not conductance and susceptance) and therefore, do not include jω.

Current Flow

To compute current flow, the AC conduction field solver uses the following relationship:

$$I = \int_A J \bullet dA$$

where

- I is the current.
- J is the current density, given by $J = \sigma E$.
- A is the area over which the current flow is computed. It is found by sweeping the current flow line you've drawn in the xy-plane into the z direction, forming a surface. The current flow computed is the current per meter depth in the z direction.

A separate current flow value can be computed for each line you draw.

Transient Simulation

Transient Solver only

The transient solver allows you to analyze the magnetic fields, energy, force/torque, power loss, core loss, speed, and flux of a model at various time steps of a solution over a specified period of time. This solver allows for non-sinusoidal current or voltage excitation, as well as rotational or translational motion.

Transient Simulation Assumptions

The transient solver assumes the following conditions about the problem:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If motion occurs in the model, no motion occurs outside the band object.
- Rotational motion can be cylindrical or non-cylindrical.
- More than one object can be assigned identical motion within the band object.

Time-Dependent Magnetic Field Simulation

For a 2D XY problem, the vectors have only one component in the Z direction. The time-dependent magnetic equation is expressed as

$$\nabla \times v\nabla \times A = J_s - \sigma\frac{dA}{dt} - \sigma\nabla\Phi + \nabla \times H_c + \sigma\Phi \times \nabla \times A$$

where

- H_c is the coercivity of the permanent magnet.
- v is the velocity of the moving parts.
- A is the magnetic vector potential.
- Φ is the electric potential.
- υ is the reluctivity.
- J_s is the source current density.

The transient solver applies a reference frame that is fixed with respect to the components in the model by setting the velocity, v, equal to zero. This is possible by considering the equation for the magnetic vector potential both in moving part and stationary part in their own reference frame in which velocity is always zero. Because the moving components have now been fixed to their own coordinate system, the partial time derivative becomes the total time derivative of A. Thus, the motion equation becomes

$$\nabla \times v\nabla \times A = J_s - \sigma\frac{dA}{dt} - \sigma\nabla\Phi + \nabla \times H_c$$

which is obtainable at each timestep at every node in the finite element model. In the above equation, the motion is contained implicitly in the total derivative of A.

Stranded Conductors

Stranded conductors lack eddy current behavior and are considered to be filaments too thin to model in a practical finite element grid. Because of this, the transient solver assumes that their contribution to the current density is averaged over the area of problem region.

These filaments can be connected in parallel or series, and carry the same current based on

$$\nabla \times v\nabla \times A = j_s$$

where j_s, the uniformly distributed current density, is given by

$$j_s = d_f \frac{N_f i_f}{S_f \cdot a \cdot p}$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and

- i_f is the total terminal current flowing into a filament coil group (or winding).
- N_f is the total conductor number of the filaments in the winding.
- A is the magnetic vector potential.
- a is the number of parallel branches in the winding.
- d_f is the polarity (+1 or -1) to represent forward or return paths.
- S_f is the total area of the cross-section of the region occupied by the winding.
- p is the ratio of the original full model to the field domain to be solved.

Note that the filaments in the winding may reside at several different locations as long as they are connected in series or parallel.

The voltage seen from the terminal of the winding is the sum of the voltage across all the wires as well as any possible external impedance based on

$$d_f \frac{N_f l}{S_f \cdot a} \iint \frac{dA}{dt} \cdot d\Omega + R' \cdot i_f + L' \cdot \frac{di_f}{dt} = u_s$$

and

$$R' = r_{dc} + r_{end} + r_{ext}$$

$$L' = l_{\text{end}} + l_{ext}$$

where

- r_{dc} is the total DC resistance.
- r_{end} is the end-turn resistance of the winding.
- r_{ext} is the external resistance connected to the winding.
- l_{end} is the end turn inductance of the winding.
- l_{ext} is the external inductance connected to the winding.
- l is the thickness of the model.

> **Note:** The main component of inductance is calculated by Maxwell; end effects – when they can be evaluated – can be included in the model to increase the overall accuracy of the simulation.

Solid Conductors

Solid conductors are large enough to model with finite elements, where skin effects depend not only on the frequency of the system, but on the location of nearby conductors.

Based on Ampere's Law, the total current density, j_t, in the system is given by

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$j_t = -\sigma\frac{dA}{dt} - \sigma\nabla V$$

which reduces to

$$j_t = -\sigma\frac{dA}{dt} + \frac{\sigma}{l}V_b$$

or

$$j_t = j_e + j_s$$

where

- V_b is the voltage difference across the conductors end points.
- j_e is the eddy current density.
- j_s is the source current density.

Solid Conductors with Current Sources

For solid conductors with a current source, the total current is known, while the source component is unknown. The transient solver computes the current source based on the following circuit equation for the n^{th} conductor:

$$\iint_{\Omega_c}\left(-\sigma\frac{dA}{dt} + j_s\right)d\Omega = \iint_{\Omega_c} j_t d\Omega = I_t$$

where

- Ω_c is the width cross-section of the *n*th conductor.
- I_t is the known total current.
- j_S is the source component to be solved for.
- j_t is the total current density.
- A is the magnetic vector potential.

Solid Conductors with Voltage Sources

For solid conductors with a voltage source, the total voltage is known, while the total current density is unknown. The transient solver computes the unknown quantities based on the following circuit equation, which is derived from the solid conductor equations. The resulting equation yields

$$\iint_{\Omega_c}\left(\sigma\frac{dA}{dt} + j_t\right)d\Omega = \iint_{\Omega_c}\frac{\sigma}{l}V_b d\Omega$$

where

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Ω_c is the width cross-section of the *n*th conductor.
- V_b is the known voltage source between the two conductors.
- j_t is the total current density to be solved for.
- σ is the conductivity.
- *l* is the thickness of the model.
- *A* is the magnetic vector potential.

Translational Motion

The transient motion simulator generates translational motion solutions based on the following motion equation:

$$ma + \lambda v = F_{em} + F_{load}$$

where

- *m* is the mass of the object, in kg.
- *a* is the acceleration of the object, in m/s^2.
- *v* is the velocity, in m/s.
- F_{em} is the computed electromagnetic force, in N.
- F_{load} is the external load force, in N.
- λ is the damping, in N·s/m.

Rotational Motion

The transient motion simulator generates rotational motion solutions based on the following motion equation:

$$J\beta + \lambda\omega = T_{em} + T_{load}$$

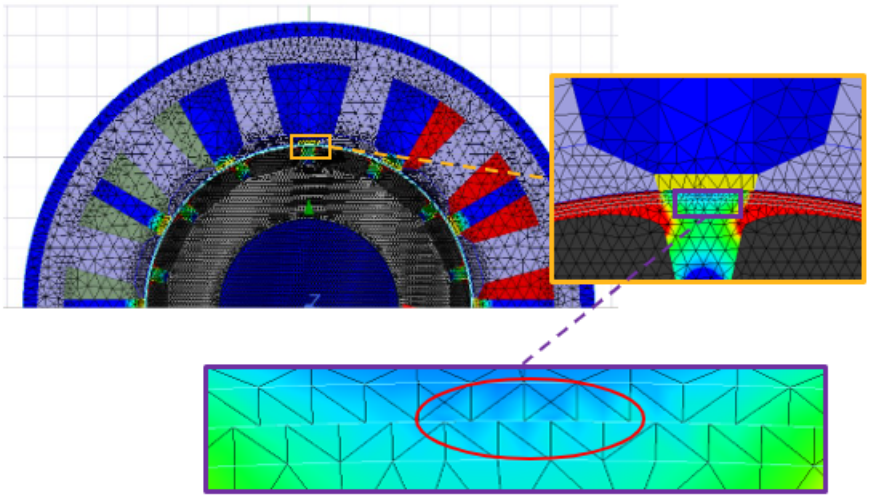
where

- *J* is the moment of inertia, in kg·m^2.
- T_{em} is the computed electromagnetic torque, in N·m.
- T_{load} is the external load torque, in N·m.
- ω is the angular speed, in rad/s.
- β is the angular acceleration, in rad/s^2.
- λ is the damping factor, in N·m·s.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Improving Reliability of Results for Air-Gap Modeling of Electric Motors (2D)

When there is motion in the simulation, the mesh on the interface of the moving and stationary domains can become nonconformal (*i.e.*, the nodes at the two sides of the interface are not in the same positions). This requires some interpolation/mapping technique to map the fields from one side to the other. Depending on the method used for this mapping, hot spots of the field on the interface can occur due to numerical error as shown in the figure below.



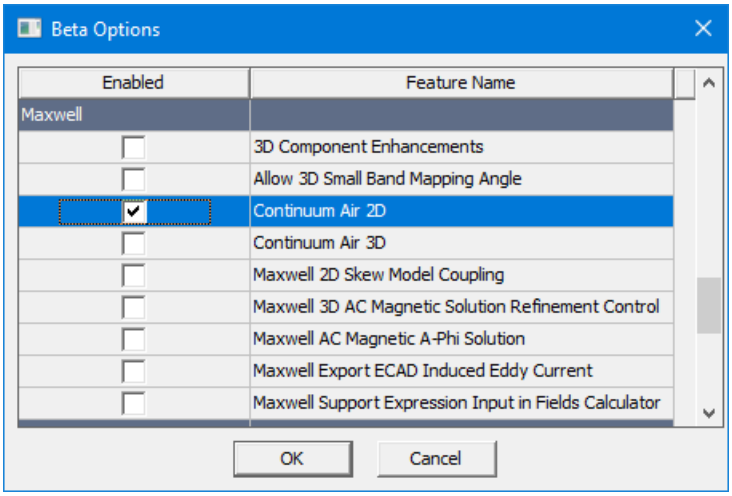
Some typical results of interest around electric machine simulation are considerably sensitive to this numerical error at the interface between moving and stationary domains. One example for such a result is the cogging torque of permanent-magnet synchronous machines. Hot spots of the magnetic field in the interface could lead to the wrong result for cogging torque.

The **Continuum Air 2D** beta feature can be used to improve the smoothness and accuracy of the field solution so that the simulation can be more robust and accurate.

To turn on this feature:

1. Open the **Tools** > **Options** > **General Options** window.
2. Click the **Beta Options** button.
3. Scroll down to the Maxwell options, and select **Continuum Air 2D**.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Click **OK** in the Beta Options window and in the General Options window.
5. Click **Yes** to restart Ansys Electronics Desktop to activate the feature.

Other result quantities of interest that may show considerable improvement in accuracy with this feature include NVH-forces and core losses at the stator (as considerable core loss density occurs at the stator tooth tips, in the vicinity of the interface).

Supported functions for this beta feature include

- All Excitation Winding Types: Current / Voltage / External
- Rotational motion type, including multi-motion rotation
- Full / Partial model
- Serial/General TDM
- Half and full Periodic TDM
- Skew models, including MPI-based distributed multi-slice skew model
- Core loss / hysteresis effect on field calculation

Translational motion is **not** supported in this release. If Continuum Air 2D is turned on when the translational motion function is activated, it will be ignored.

Core Loss for Transient Solvers

Core loss allows the solver to find the total core loss on an object, or group of objects due to the distribution of the electric or magnetic field in the design.

For frequency domain applications:

At a given frequency, the core loss for electrical steel is based on the following expression:

$$p = K_h B_{\max}^2 f + K_c (B_{\max} f)^2 + K_e (B_{\max} f)^{1.5}$$

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where

- K_h is the hysteresis coefficient.
- K_c is the classical eddy coefficient.
- K_e is the excess or anomalous eddy current coefficient due to magnetic domains.
- B_{max} is the maximum amplitude of the flux density.
- f is the frequency.

Similarly, the power ferrite core loss is based on the following expression:

$$p = C_m f^x B_{\max}^y$$

where

- C_m is a constant value determined by experiment.
- f is the frequency.
- B_{max} is the maximum amplitude of the flux density.

For transient (time domain) applications, the above expressions need to be modified in the following ways:

In the frequency domain, the core loss for power ferrites is given by the above formula, where C_m, x, and y are measured based on sinusoidal $B(t)$ waveform. If $x = 1$, $y = 2$, and $C_m = K_h$, the formula for power ferrite is that for hysteresis loss. If $x = y = 2$, and $C_m = K_c$, it corresponds to classic eddy loss. If $x = y = 1.5$, and $C_m = K_e$, it becomes excess loss. Therefore, the formula of power loss for ferrite can be used to describe all individual components for core loss for both electric steel and power ferrite.

Based on the formula for ferrite core loss in the frequency domain, the following expression can be derived to compute core loss in the *time* domain:

$$P_f = C_t |H_{irr}|^{y-x} \left| \frac{dB}{dt} \right|^x$$

where H_{irr} is the irreversible component of the magnetic field strength $H_{irr} = H - H_{rev}$, with H_{rev} being the reversible component of H, which can be obtained from the BH curve based on B. To obtain H_{irr}, an equivalent ellipse is used with the property that the area of the ellipse equals the are of the original hysteresis curve. This explains why, in the process of obtaining the hysteresis loss, the $B(H)$ curve and K_h coefficient are used.

The equation of the equivalent ellipse is given by

$$\left(\frac{H_{irr}}{H_m}\right)^2 + \left(\frac{B}{B_m}\right)^2 = 1$$

where

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$H_m = \frac{K_h \cdot B_m}{x}$$

and B_m is the amplitude of a minor B-H loop which is obtained based on continuously updated history information of the magnetic flux density, which allows the algorithm to account for the contribution of minor loops to the core loss.

With H_{irr} and *dB/dt*, the loss can be easily evaluated. For the other components of the core loss (that is, different combinations of *x* and *y*), unknown coefficients can be derived using the property that the formula for core loss in the time domain gives the same result as the formula for core loss applied for the frequency used in the experiment.

Phasor Notation

> **Note:** The information in this topic is applicable to the 2D AC Magnetic solver and the 2D AC conduction solver.

Time varying quantities that have the form:

$$F(t) = F_m \cos(\omega t + \theta)$$

can be represented as rotating phasors in the complex plane. Using Euler's formula:

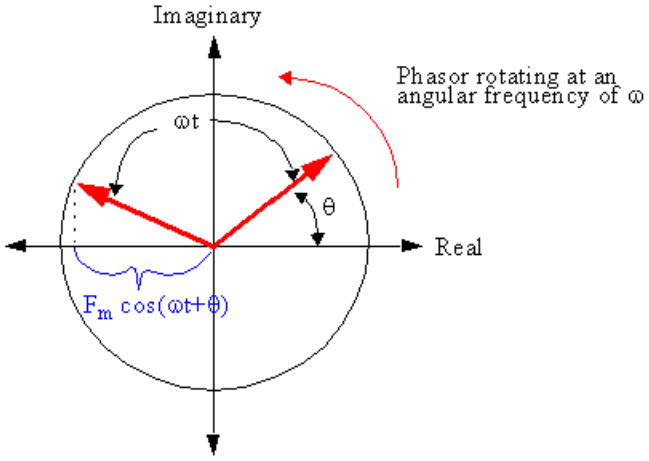
$$e^{j\alpha} = \cos\alpha + j \sin\alpha$$

If $\alpha = \omega t + \theta$, **F**(t) equals the real portion of $e^{j(\omega t+\theta)}$:

$$F(t) = \Re\left[F_m e^{j(\omega t+\theta)}\right] = \Re\left[F_m(\cos(\omega t+\theta) + j\sin(\omega t+\theta)\right] = F_m\cos\left(\omega t+\theta\right)$$

Each time-varying quantity has the form $F_m e^{j\theta} e^{j\omega t}$. The $F_m e^{j\theta}$ component is merely a complex constant that can be represented by a stationary phasor in the complex plane. The $e^{j\omega t}$ component is a complex number that depends on *t*, and can be represented as a rotating phasor in the complex plane, as shown here.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The phasor's projection on the real axis oscillates sinusoidally. It reaches a peak when parallel with the real axis, and crosses zero when parallel with the imaginary axis. Thus, a phasor with $\theta=90°$ represents a quantity that peaks 90 degrees after a phasor with $\theta=0°$.

Real and Imaginary Components

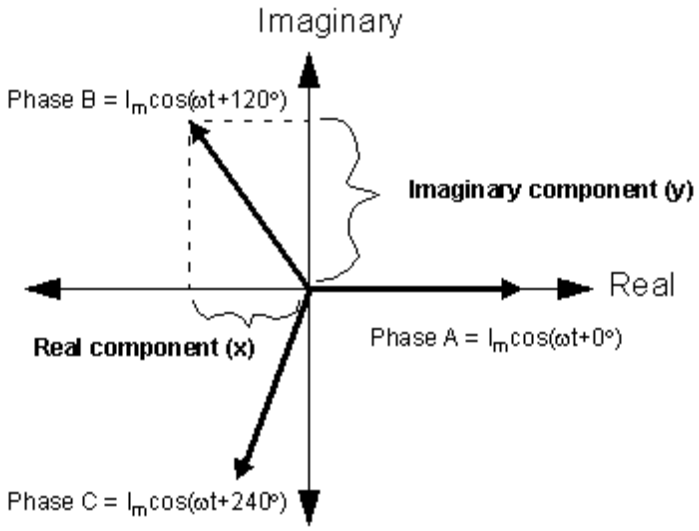
Maxwell expects you to enter magnitudes and phase angles when you specify voltages, currents, and other boundary or source quantities in models where time-varying fields are to be computed (that is, AC magnetic, AC conduction, and AC magnetic axial models). But when the magnitude and phase angle of a quantity are functions of position, it is easier to specify the functions in terms of real and imaginary components "x + jy" rather than in terms of magnitude and amplitude. Therefore, when you specify currents and boundary conditions as functions, the system expects you to describe the functions in terms of real and imaginary components.

The "x + jy" description of a phasor indicates that the phasor is the sum of two components — a sinusoid that peaks at $\omega t = 0°$ and a sinusoid that peaks at $\omega t = 90°$.

- The "x" component of the phasor is called the real component and can be represented by a phasor that lies on the real axis of the complex plane.
- The "y" component is called the imaginary component and is represented by a phasor that lies on the j-axis of the complex plane.

The real and imaginary components of a three-phase system are shown below:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The real and imaginary components are related to the magnitude and phase of a sinusoid in this way:

$$I_m = \sqrt{x^2 + y^2}$$

$$\theta = \operatorname{atan} \frac{y}{x}$$

Boundary Conditions

Boundary conditions define the behavior of the electric or magnetic field at object interfaces or edges of the problem region. They are always necessary in order to insure the uniqueness of the electromagnetic field calculation. They can also be used to do the following:

- Simulate structures that are magnetically isolated, electrically insulated, or electrically isolated.
- Set the electric or magnetic potential at a surface to a constant value or a function of position, in order to define the behavior of the electric or magnetic field on that surface
- Simulate the field patterns that would exist in a structure while modeling only part of it. To do this, you can define planes of symmetry where electric or magnetic fields are either tangential to or normal to the surface. Additionally, you can define planes of symmetry where the field on one surface matches the magnitude and direction (or opposite direction) of the field on another surface.
- Simulate the field patterns produced by thin resistive layers on conductors (DC conduction solver) or eddy currents with very tiny skin depths in conductors (AC Magnetic solver), without having to explicitly draw, assign materials to, or solve for fields inside the objects in question.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Available boundary types in Maxwell 2D are listed in the following table. Select from the table to see more information about a specific boundary type or for information on how to apply it in a specific problem type.

Boundary Condition	Applying in Problem Type
Vector Potential	Magnetostatic
	AC Magnetic
	Transient
Symmetry	Magnetostatic
	Electrostatic
	AC Conduction
	DC Conduction
	AC Magnetic
	Transient
Balloon	Magnetostatic
	Electrostatic
	AC Conduction
	DC Conduction
	AC Magnetic
	Transient
Impedance	AC Magnetic
Resistance	DC Conduction
Independent/Dependent	Magnetostatic
	Electrostatic
	AC Conduction
	DC Conduction
	AC Magnetic
	Transient

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Vector Potential

The Vector Potential boundary is available for use in Magnetostatic, AC Magnetic, and Transient solver type designs. Use Vector Potential boundaries to set the magnetic vector potential, $\mathbf{A}_Z$, to a constant value on a boundary.

The potential can also be defined as a function of position using math functions. Normally, this type of boundary condition is used to specify the potential of outer boundaries. It can also be used to set the interface between two objects to a potential, modeling the presence of a very thin conductor between the objects. They are sometimes called Dirichlet boundaries.

The behavior of the magnetic field on a Vector Potential boundary depends on whether you define a constant or functional potential on the boundary. Remember that the magnetic vector potential, **A**, is defined to be a field that satisfies the equation

$$\nabla \times \mathbf{A} = \boldsymbol{B}$$

Because the magnetostatic field solver assumes that **A** has a z-component only and **B** lies in the xy-plane, the relationship of **B** to **A** is given by the following:

$$\boldsymbol{B} = \frac{\partial \boldsymbol{A_z}}{\boldsymbol{dy}}\hat{\boldsymbol{x}} - \frac{\partial \boldsymbol{A_z}}{\boldsymbol{dx}}\hat{\boldsymbol{y}}$$

If $\mathbf{A}_Z$ is constant along a horizontal boundary, the partial derivatives of $\mathbf{A}_Z$ with respect to x will be zero – forcing **B** to have an x-component only, and be tangential to the boundary. Likewise, if $\mathbf{A}_Z$ is constant along a vertical boundary, the partial of $\mathbf{A}_Z$ with respect to y will be zero – forcing **B** to have a y-component only and again indicating that the field will be tangential.

In general, the magnetic field will be tangential to any boundary on which $\mathbf{A}_Z$ has been set to a constant.

If the potential is a function of position, the partial derivatives of $\mathbf{A}_Z$ with respect to x and y will not necessarily be zero. It all depends on what type of math function was used to specify the potential. Thus, **B** may not be tangential to the boundary and some flux will cross it.

In AC magnetic problems, the magnetic vector potential is a time-varying quantity in the form:

$$\boldsymbol{A_Z}(t) = \boldsymbol{A_m}\, \boldsymbol{cos}(\omega t + \theta)$$

where A_m is the magnitude of the potential and θ is its phase angle — its offset from a pure cosine wave. Therefore, when specifying $\mathbf{A}_Z$ on a boundary, you must enter both its magnitude and phase. The magnitude and phase of the potential can also be defined as a function of position using math functions.

Vector Potential Boundaries in Axisymmetric Models

In axisymmetric models, **A** is assumed to have only a Φ-component and **B** is assumed to lie only in the rz-plane. The relationship between $\mathbf{A}_\Phi$ and **B** is given by

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\boldsymbol{B} = \frac{1}{r}\left[-\frac{\partial}{\partial z}\left(rA_{\Phi}\right)\hat{r} + \frac{\partial}{\partial r}\left(rA_{\Phi}\right)\hat{z}\right]$$

Because equipotential lines of $r\mathbf{A}_{\Phi}$ in axisymmetric models coincide with the lines of magnetic flux, you must specify values or functions of $r\mathbf{A}$ (not **A**) when setting value boundaries.

Symmetry

The symmetry boundary is available for use in Magnetostatic, AC Magnetic, Transient, Electrostatic, AC Conduction, and DC Conduction solver type designs. A symmetry boundary models a plane of symmetry in a structure. Use this type of boundary condition to take advantage of geometric symmetry and electrical symmetry in a structure. Doing so enables you to reduce the size of your model — allowing you to conserve computing resources. Two types of symmetry boundaries — **Odd** and **Even** — are available for use.

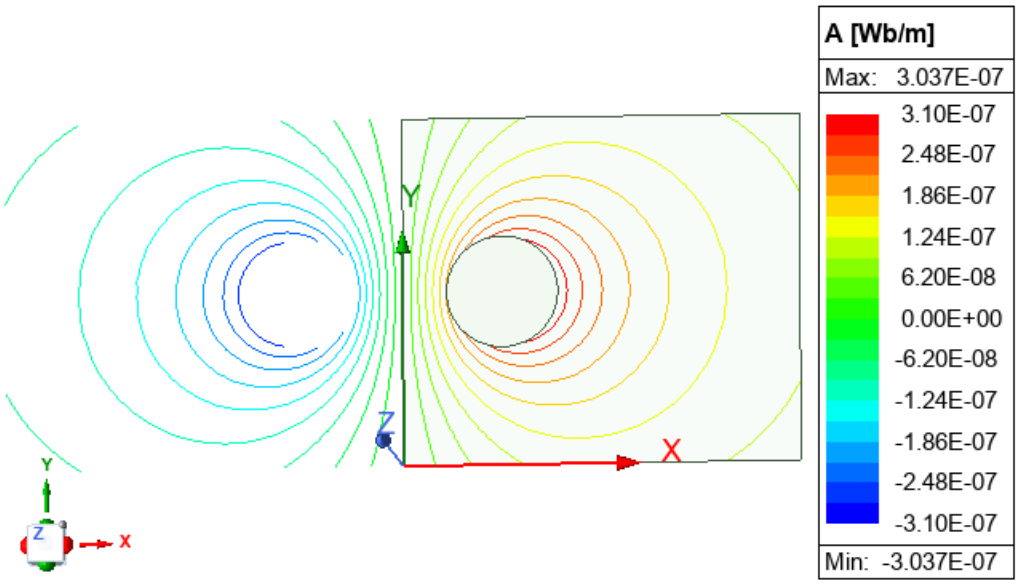
Odd Symmetry

An odd symmetry boundary models a structure in which the signs (positive or negative) of all currents, voltages, or charges on one side of a symmetry plane are the opposite of those on the other side. In magnetic field problems, the magnetic field is tangential to this type of boundary; while in electric field problems, the field is perpendicular to the boundary and equipotential lines are tangential to the boundary.

To define an odd symmetry boundary for magnetic field problems, the simulator sets the selected edge to a vector potential boundary with a magnetic vector potential of zero — acting as a magnetic mirror to the model. For electric field problems, the simulator sets the selected edge to a voltage of zero.

For instance, the plane of symmetry shown below is modeled by an odd symmetry boundary: the direction of the current flow in the conductor on the left side of the symmetry plane is the opposite of the current flow in the conductor on the right side of the plane (the side that is modeled):

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



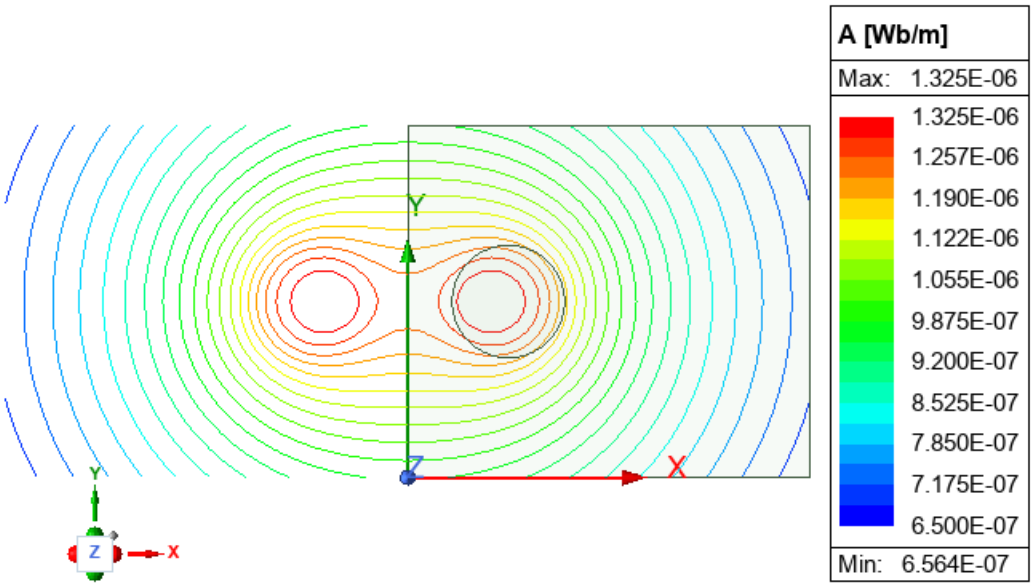
Even Symmetry

An even symmetry boundary models a structure in which the signs (positive or negative) of the currents, voltages, or charges on one side of a symmetry plane are the same as those on the other side. The magnetic field is perpendicular to this type of boundary; while the electric field is tangential to this type of boundary, and contours of equal potential are perpendicular to it.

To define an even symmetry boundary, the simulator sets the selected edge to a Neumann boundary.

The result of an even symmetry boundary is shown below. The plane of symmetry can be modeled by an even symmetry boundary because the direction of the current flow in the conductor on the left side of the symmetry plane is the same as that of the current flow in the conductor on the right side of the plane (the side that is modeled):

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Balloon

The balloon boundary is available for use in Magnetostatic, AC Magnetic, Transient, Electrostatic, AC Conduction, and DC Conduction solver type designs. Balloon boundaries model the region outside the drawing space as being nearly “infinitely” large – effectively isolating the model from other sources of current or magnetic fields. Visualize the **background** object as extending to infinity along the edges identified as balloon boundaries.

For magnetic solution types, the magnetic vector potential, $\mathbf{A}_Z$ or $\mathbf{A}_\Phi$, goes to zero at infinity. The lines of magnetic flux are neither tangential to nor normal to a balloon boundary.

For electric field solution types, two types of balloon boundaries are available:

Charge Models the case where the charge at infinity matches the charge in the solution region, forcing the net charge to be zero. Physically, this represents an electrically insulated system. This is the default for Electrostatic solutions.

Voltage Models the case where the voltage at infinity is zero. Physically, this represents an electrically grounded system. In most cases, the results will be very similar to those produced with the **Charge** option; however, the charge at infinity may not exactly match the charge in the drawing region.

The E-field is neither tangential to nor normal to a balloon boundary.

> **Note:** The Balloon boundary condition creates a background geometry that can be used in the Fields Calculator. See Fields Calculator's Geometry Command for more details.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Impedance

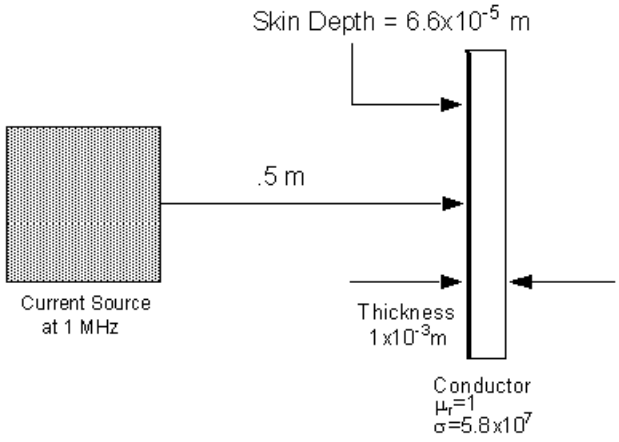
The impedance boundary is available for use in AC Magnetic solutions. Impedance boundaries allow you to simulate the effect of induced currents in a conductor without explicitly computing them. The ohmic loss due to induced currents is computed from the tangential components of the H-field along the impedance boundary — the surface of the object that you are interested in.

Use this boundary condition for models where the following conditions occur:

- The skin depth in the conductor of interest is less than two orders of magnitude smaller than the dimensions of the structure. In models like this, the Maxwell 2D's mesher may not be able to create a fine enough mesh in the conductor to compute eddy currents.
- The magnetic field decays much more rapidly inside the conductor in the direction that is normal to the surface than it does in directions that are tangential to the surface.
- The AC current source is relatively far away from the surface where eddy currents occur, compared to the size of the skin depth.

The object itself must be excluded from the solution region by making the object a perfect conductor. When drawing the geometry, make the surface along which eddy currents are to be computed an outer surface of the problem region. Then, when defining boundaries, assign an impedance boundary to this surface. By entering the conductivity, σ, and the relative permeability, μ_r, of the object, you specify the skin depth of induced eddy currents. The simulator uses this skin depth when computing the electromagnetic field solution. It assumes that the H-field falls off exponentially inside the conductor.

For instance, suppose you want to compute eddy current losses in the conductor next to the current source shown below.



If AC current is passing through the current source at a frequency of 1 MHz, the skin depth in the conductor is given by the following relationship:

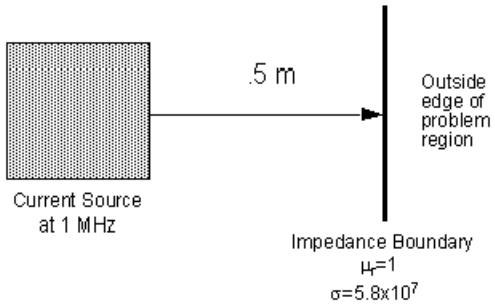
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\delta = \sqrt{\frac{2}{\omega\sigma\mu_r\mu_0}}$$

where

- $\omega = 2\pi f = 2\pi \times 10^6 = 6.28 \times 10^6$ radians/second
- $\sigma = 5.8 \times 10^7$ siemens/meter
- $\mu_r = 1$
- $\mu_0 = 4\pi \times 10^{-7}$ henries/meter

Substituting these values into this equation, the skin depth is found to be 6.6×10^{-5} meters. Because this is much smaller than the thickness of the conductor and the surface where currents are induced is relatively far away from the current source, an impedance boundary can be used to model the induced currents in the conductor, as shown below. The conductor itself is not included in the model; instead, the outside boundary of the model is moved to the inside surface of the conductor. This outside surface is defined as an impedance boundary, using the conductivity and permeability specified previously.



After generating a solution, you can compute the ohmic loss for the surface using the plane calculator and plot the loss density on the boundary.

For impedance boundaries, ohmic loss is given by

$$P = \sqrt{\frac{\omega\mu_0\mu_r}{8\sigma}} \int_{Sur} \boldsymbol{H}_t \bullet \boldsymbol{H}_t^* dS \quad \left(Watts\right)$$

where

- ω is the angular frequency, which is equal to $2\pi f$.
- σ is the conductor's conductivity in siemens/meter.
- μ_r is the conductor's relative permeability.
- μ_0 is the permeability of free space, which is equal to $4\pi \times 10^{-7}$ H/m.
- $\boldsymbol{H_t}$ is the tangential component of **H** on the impedance boundary.
- $\boldsymbol{H_t^*}$ is the complex conjugate tangential component of **H** on the impedance boundary.

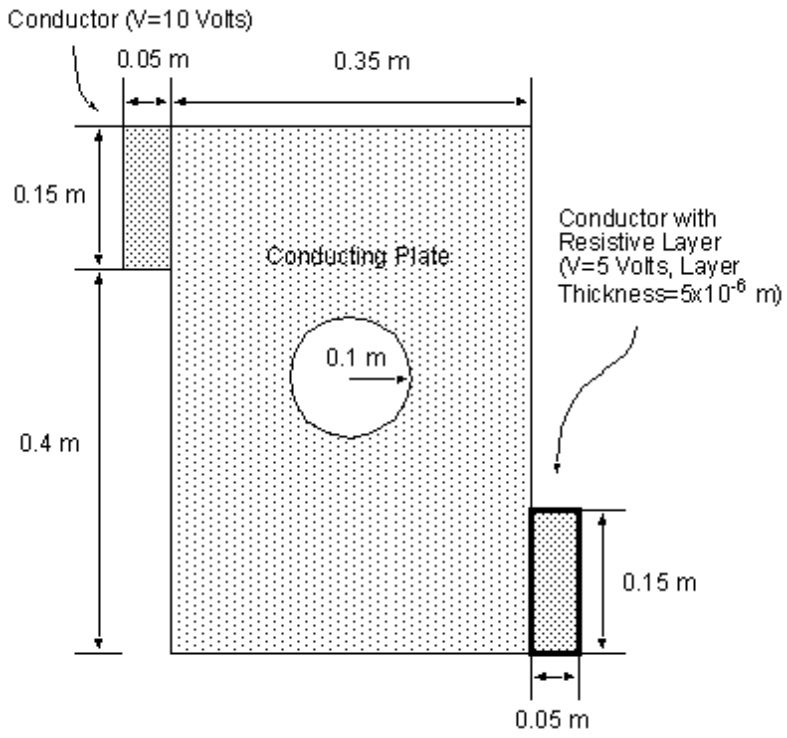
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: Keep in mind that an impedance boundary approximates the effect of eddy currents acting at a shallow skin depth; it does not directly compute them. In general, the fields modeled using an impedance boundary will closely match the field patterns that would actually occur in the structure. However, the field patterns may be different at discontinuities in the surface such as corners.

Resistance

The resistance boundary is available for use in DC Conduction solutions. A resistance boundary models a very thin layer of resistive material (such as that caused by deposits or oxidation on a metallic surface) on a conductor at a known potential. Use this boundary condition when the resistive layer's thickness is much smaller than the other dimensions of the model.

For instance, in the following example, the resistive layer on the conductor is $5x10^{-6}$ meters thick. Because this is four orders of magnitude smaller than the dimensions of the model, use a resistance boundary on the conductor to avoid having to create a very thin object modeling the layer — which could cause problems when the Maxwell generates a mesh for the model and solves for its conduction currents.



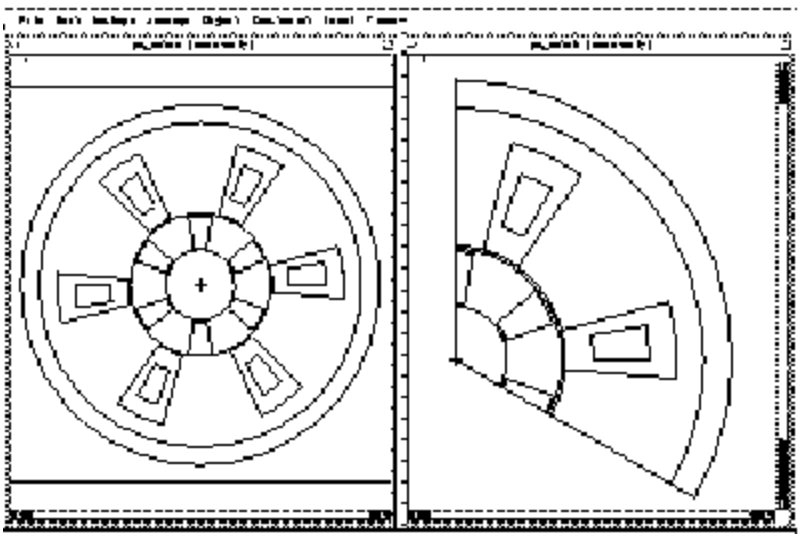
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specify the thickness and conductivity of the resistive material, and the potential of the conductor. Apply resistance boundaries only to the outside edge of the problem space. The external region touching a resistance boundary is assumed to be a PEC (perfect conductor) with a known potential value that is specified in the resistance boundary setup. For the example above, if the PEC region is included in the design model, solve inside should not be selected to exclude it from the FEM (finite element method) problem domain.

Independent/Dependent

Independent/Dependent boundaries are available for use in Magnetostatic, AC Magnetic, Transient, Electrostatic, AC Conduction, and DC Conduction solver type designs. Matching boundaries allow you to take advantage of periodicity in a structure. For example, the following figure shows the cross-section of a DC motor. The field in such a motor repeats itself every 120 degrees; that is, the field pattern in one third of the motor matches the magnitude and direction (or the opposite of the direction) of the field pattern in the other two thirds.

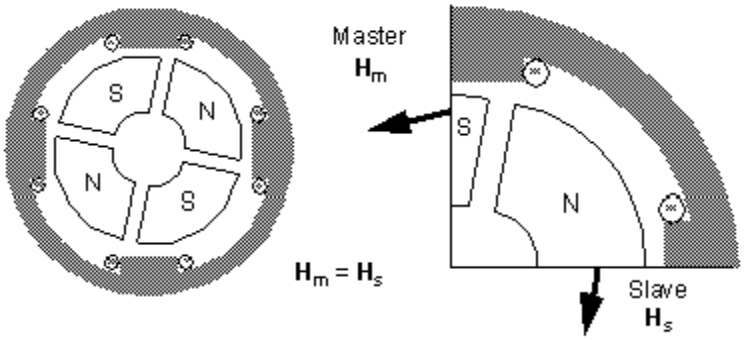
Matching boundaries force the magnetic field at each point on one boundary (the “dependent” boundary) to match the magnetic field at each corresponding point on the other surface (the “independent” boundary). Modeling one third of the structure allows you to make efficient use of the available computing resources:



To define matching boundaries, you must define both an independent matching boundary and a dependent matching boundary.

The condition that needs to be enforced, as illustrated in the following figure, is that the magnitude of the magnetic field at each point on the dependent boundary surface must match the magnetic field at each corresponding point on the independent boundary surface. The field on the dependent boundary must point in either the same direction or in the exact opposite direction as the field on the independent boundary:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



One-quarter of a periodic structure (DC motor) modeled using matching boundaries.

Note that a Vector Potential (Dirichlet), Neumann or symmetry boundary cannot be used to simulate periodicity because the magnetic field is not necessarily either perpendicular or tangential to periodic surfaces. For example, in the quarter model shown above, the magnetic field is exactly perpendicular to the bounding surfaces only when the gap separating the permanent magnets is perfectly horizontal or vertical. For all other positions of the rotor, matching boundaries are required to take advantage of symmetry.

2D Sources/Excitations

To compute fields for a structure, you must define a source of charge, voltage, current, or electric or magnetic fields for your model. Assign at least one object or edge as either a source (such as a current, charge, or voltage) or a value boundary.

Permanently polarized or magnetized materials also act as sources of charge or magnetic field (respectively). If you do not identify some type of source, the Maxwell 2D will not be able to generate a solution.

The field quantities computed by each solver — and the required electromagnetic sources — are given in the following table:

Field Solver	Sources	Field Computed	Derived Field Quantities
Electrostatic	Voltages charges charge density floating	Φ	**E**, **D**
Magnetostatic	DC current current density	$\mathbf{A}_Z$ (XY models), $\mathbf{A}_\Phi$ (RZ	**H**, **B**

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Field Solver	Sources	Field Computed	Derived Field Quantities
		models)	
AC Magnetic (Eddy Current)	AC currents current density	$\mathbf{A}_Z(\omega t)$ (XY models), $\mathbf{A}_\Phi(\omega t)$ (RZ models)	$\mathbf{J}_Z(\omega t)$ (XY models), $\mathbf{J}_\phi(\omega t)$ (RZ models), $\mathbf{H}(\omega t)$, $\mathbf{B}(\omega t)$,
DC Conduction	DC voltages	Φ	**E**, **D**, **J**
AC Conduction	AC voltages	$\Phi(\omega t)$	$\mathbf{E}(\omega t)$, $\mathbf{J}(\omega t)$
Transient	Transient voltages and currents through External Circuit connection. Current; current density; coil; end connection	$\mathbf{A}_Z$ (XY and RZ models)	**H**, **B**

where

- **A** is the magnetic vector potential.
- **H** is the magnetic field.
- **B** is the magnetic flux density.
- Φ is the electric potential.
- **E** is the electric field.
- **D** is the electric flux density.
- **J** is the current density.

These quantities are phasors in AC simulations.

2D Voltage Sources

Voltage sources are available for the Electrostatic, DC Conduction, AC Conduction and Transient solver types. Voltage sources specify the total DC voltage (electric potential), or the magnitude and phase of the AC voltage on a conductor. Voltages can be defined as constants or as functions; however, the voltage is assumed to be uniform over the source.

Voltage sources may also be specified on a selected edge or edges.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DC Voltage Source

This type of source specifies the total DC voltage (electric potential) on a conductor. Voltages can be defined as constants or as math functions; however, the potential on a conductor is constant over the entire conductor. Note that conductors that touch should be set to the same voltage or defined as a single voltage source, since their potentials are identical.

DC voltage may also be specified on an edge or edges. Voltages can be defined as constant or as functions of position (for instance, to model a specific distribution of potential on the surface of a dielectric).

AC Voltage Source

AC voltage sources are used in AC conduction, and Transient models. All voltages in AC models are time-varying quantities in the form:

$$\mathbf{V}(t) = \mathbf{V}_m \cos(\omega t + \theta)$$

where V_m is the magnitude of the voltage and θ is its phase angle — the offset of the current from a pure cosine wave. Therefore, when specifying a voltage, you must enter both its magnitude and phase.

2D Current/Current Density

Current sources are available in the Magnetostatic, AC Magnetic (Eddy Current), and Transient solver types. This type of source specifies the total DC current, or the magnitude and phase of the AC current on a conductor. Currents can be defined as total current, current density, or as functions.

Current sources may also be specified on a selected edge or edges. Edge currents can be defined as total surface current or a surface current density. In the case of edge currents, eddy current effects are not modeled because all currents are surface currents.

Generally,

- If the total surface current is specified, the current density is assumed to be uniform.
- If the current density is specified, you may define a uniform current density or one that varies as a function of position to model specific distributions of current on the surface.

DC Current Source

This type of source specifies the DC current flowing in a conductor. You can set either the total current or the current density flowing in the object.

- If total current is specified, the current density is assumed to be uniformly distributed throughout the object.

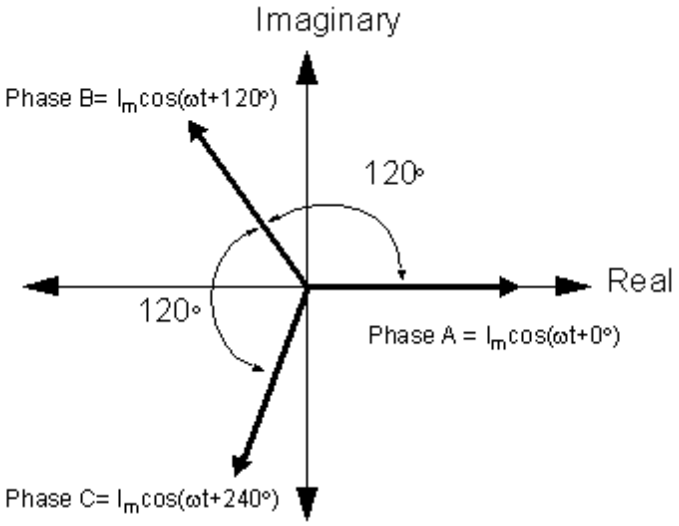
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If current density is specified, you may define a uniform current density or one that varies as a function of position.

AC Current Source

Remember, all AC currents are time-varying quantities in the form $I = I_m \cos(\omega t + \theta)$, where I_m is the magnitude of the current and θ is its phase angle — the offset of the current from a pure cosine wave. Therefore, when specifying a current or current density, you must enter both its magnitude and phase.

- In a single-phase system, time t = 0 is usually chosen so that the phase angle, θ, is zero — that is, the current peaks at t = 0.
- In multi-phase systems involving currents that are out of phase with each other, time t = 0 is usually chosen so that one current has a phase angle equal to zero. For example, phase angles in a three-phase system could be assigned as shown here:



Active vs Passive Conductors

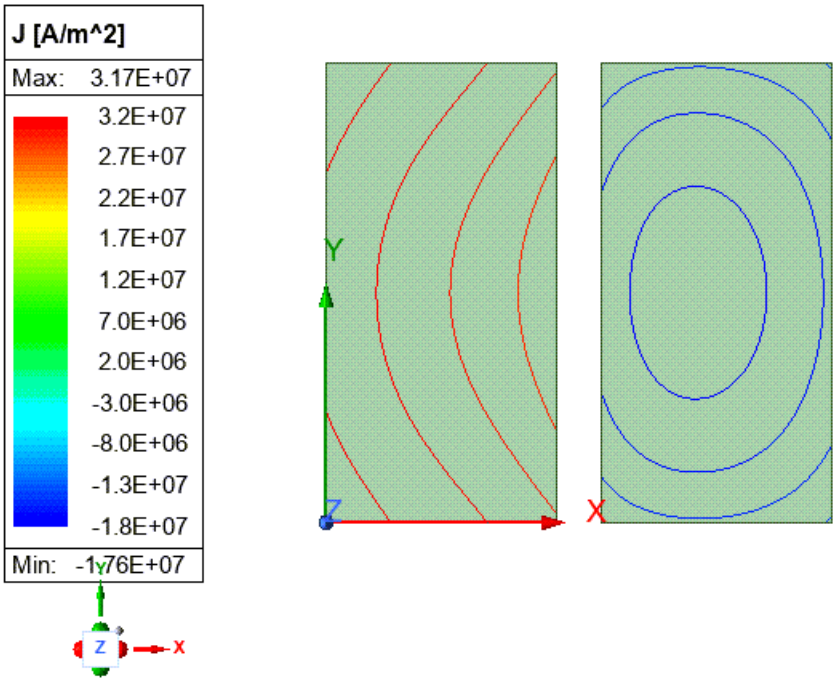
In addition, the conductors in an eddy current model can be divided into two groups:

- “Active” conductors. These conductors are connected to an external current source. Their total current is constrained to the value you specify.
- “Passive” conductors. These conductors are not connected to an external source, but current may be induced in them. Treat any conductor in which the current is constrained to zero (an open circuit) as being connected to a zero-amp current source.

Active and passive conductors are shown in the figure below. In this simple transformer model, the coil on the left is an active conductor carrying 1500 amps of current. The coil on the right is a

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

passive conductor in which current is induced by the oscillating magnetic field. The total current is plotted.



Passive Conductors

Passive conductors can have eddy and displacement currents flowing through them, but have no component of source current. Two types of passive conductors may be defined:

- To define a passive conductor modeling a short circuit, assign a conducting material to the desired object. Do not assign source current to it. There are no constraints on the eddy and displacement currents flowing in this type of passive conductor. For Cartesian models, visualize this type of conductor as being infinitely long and eventually looping back on itself. For axisymmetric models, visualize this type of conductor as a conducting ring that carries no source current.
- To define a passive conductor modeling an open circuit, assign a solid current source with a magnitude and phase of zero to it. Current may be induced in it, but the net current is constrained to zero amps. In Cartesian models, visualize this type of conductor as an infinitely long conducting rod with no return path for current. In axisymmetric models, visualize this type of conductor as a conducting ring with a gap in it.

Current Sources for Touching Conductors

Conductors whose surfaces touch are assigned sources as follows:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If they are not assigned the same material, these conductors must be defined as a parallel source. Otherwise, they will behave as if they are separated by a thin layer of insulating material.
- If they are assigned the same material, these conductors may be defined either as a parallel source or as grouped conductors assigned a solid source.

This distributes current appropriately across the surfaces of the conductors.

Current Sources for Perfect Conductors

This describes the case in which all current in a perfect conductor flows only on the surface of the conductor. Magnetic fields cannot penetrate this type of conductor. You can only specify the total DC current when defining a perfect conductor as a current source.

A perfect current source specifies the magnitude and phase of the AC current flowing through a perfect conductor. All currents in perfect conductors are surface currents, simulating the conductor's behavior at very high frequencies. You can only specify the magnitude and phase of the total current.

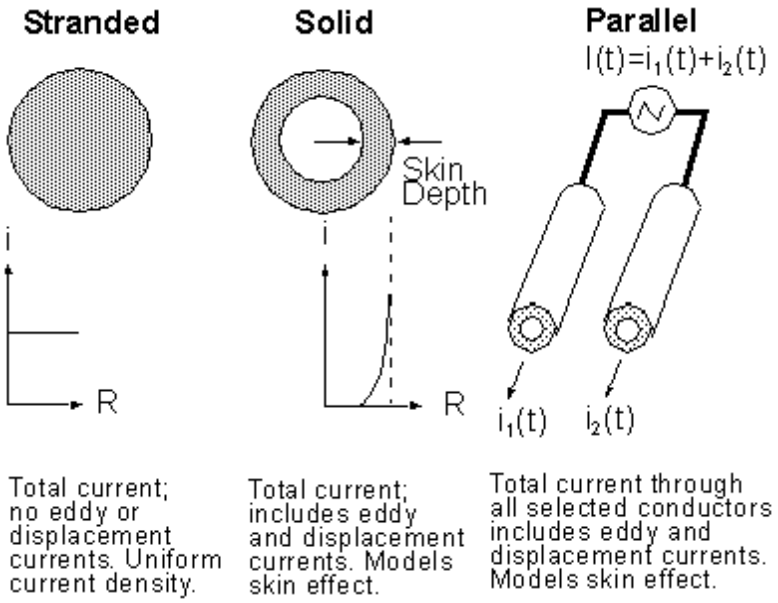
Solid, Stranded, and Parallel Current Sources

These types of sources specify the magnitude and phase of the AC current flowing through a conductor. Solid current sources model eddy and displacement currents in a solid conductor. The amount of eddy current and displacement current — as well as the amount of source current — are included in the total current you specify.

- Stranded current sources model current as being carried on strands within a conductor. They can be used to model conductors made up of many individual insulated turns, all small enough so that eddy currents can be neglected. Eddy currents and displacement currents are not computed inside the conductor. Either the total current or the current density may be specified. A uniform current density is assumed throughout the conductor, unless a functional current density is defined.
- Parallel current sources connect two or more conductors in parallel to an outside source. The total current flowing through *all* selected conductors (including eddy and displacement currents) is specified. However, the current flowing through individual conductors in the parallel group is unconstrained and its value is computed during the solution.

The differences between each type of AC current source are shown below:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For solid and parallel current sources, the current you specify is the total current in the conductor:

$$I_{Total} = I_{Source} + I_{Eddy} + I_{Displacement}$$

where:

- I_{Total} is the total current flowing through the source. It satisfies Ohm's law with the potential seen by the source.
- I_{Source} is the current due to the potential difference generated by the external source. It is the current that the source would supply if you reduced the potential difference by the back EMF produced by the eddy and displacement currents in the conductor.
- I_{Eddy} is the eddy current induced in the conductor due to time-varying magnetic fields penetrating the conductor.
- $I_{Displacement}$ is the displacement current due to time-varying electric fields in the conductor. It becomes significant only at very high frequencies.

For stranded current sources, the current you specify is the total source current (or source current density), $I_{Total} = I_{Source}$. Eddy current and displacement current effects are neglected.

2D Charge/Charge Density

This type of source specifies the total charge or charge density on a non-conducting object.

- If the total charge is specified, charge is assumed to be uniformly distributed throughout the interior of the object.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If a constant value for the charge density is specified, charge is assumed to be uniformly distributed throughout the object. The charge density can also be specified as a function of position to model a distribution of charge that varies inside the object.

This type of source may also specify the charge on a selected edge or edges. It is used primarily to assign surface charges to non-conductors. The “surfaces” being referred to are those created by extending the edge in the z direction (Cartesian models) or revolving it around the z-axis (axisymmetric models). Specify either the total charge or the charge density.

If the total charge is specified, charge is assumed to be evenly distributed on the selected surface.

If the charge density is specified, you can define either a uniform charge density or one that varies as a function of position — to model specific distributions of charge on the surface.

The electrostatic field simulator computes the electric potential on the edge during the solution.

2D Floating

This type of source specifies the total charge on a conductor, identifying it as a floating conductor. Charge is assumed to be evenly distributed on the object’s surface. Its value can be defined as a constant or as a function of position; however, charge is distributed over a conductor so that the electric potential is constant throughout the conductor. Because of this, the E-field is equal to zero in this region and no solution is computed inside the conductor.

External Circuit Connections

You may define transient current sources and transient voltage sources using the **Maxwell Circuit Editor**. This allows you to define an external circuit netlist which can be connected to the 2D model to act as a source in the time domain.

To define an external connection, the general procedure should be followed:

1. Create sheets objects to be used as terminals.
2. Assign the **Coil Terminal** excitation to the sheet objects.
3. Create a **Winding**, provide a unique name for the winding, and specify **External Circuit** as the winding type. Select either **Solid** or **Strand** as the type of conductor for the external connection.

   > **Warning:** You cannot mix solid and stranded sources in the external circuit.

4. Choose **Winding**, and add the **Coil Terminals** to the winding using the **Add Terminals** dialog box.
5. Launch the **Maxwell Circuit Editor** from the Windows Start menu.
6. Add a **Winding** element to the circuit with the same name as given to the winding in the Maxwell design.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Add additional circuit element to the schematic to create the external source circuitry for the winding.

   > **Note:** The transient solver accepts only the following subset of elements available in Schematic Capture: resistors, inductors, capacitors, diodes, switch on voltages, switch on currents, current sources, voltage sources, current meters, and voltage meters.

8. **Export** the circuit to a netlist, and **Save** and **Exit** the **Maxwell Circuit Editor**.
9. In the Maxwell design, use the **Edit External Circuits** command to **Import** the netlist.

The external circuit is now ready for use in the Maxwell design.

References for Electric or Magnetic Potential

You must specify a reference for electric scalar potential or magnetic vector potential that Maxwell 2D can use when computing fields. To do so, assign one of the following boundary or source types to at least one surface in your model:

- Value boundary
- Voltage source
- Odd symmetry boundary
- Balloon boundary

If you do not set a reference for electric or magnetic potential, the model is not uniquely defined and an error message appears when you try to generate a field solution. This problem usually occurs when you set up:

- Electrostatic problems that contain only charge sources. The electrostatic field solver requires that a reference voltage be defined in order to compute the electric potential (and from it, the electric field) in the problem region.
- Magnetostatic and AC magnetic problems that contain only current sources. These solvers require that a reference value of $\mathbf{A}_Z$ or $r\mathbf{A}_\Phi$ reference value be set in order to compute the magnetic vector potential (A_Z) and from it, the magnetic field in the problem region.

End Connections in 2D Transient and AC Magnetic Solvers

Choose this command to assign an end connection to a group of solid objects. This causes all objects in the group to be connected electrically in parallel using a finite resistance and inductance between adjacent objects.

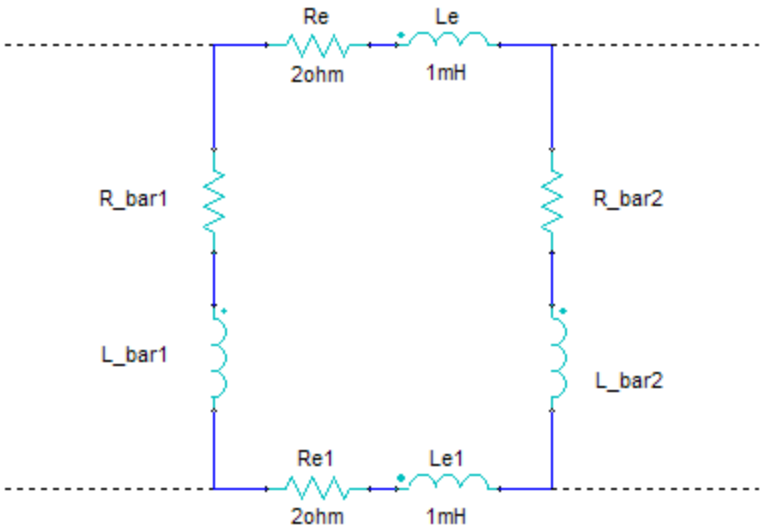
End connections are primarily used in passive conductors (with no source current assigned) when modeling cylindrical squirrel cage induction motors.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For evenly distributed conductors, the End Resistance, R_e, and the End Inductance, L_e, are assigned between the ends of each conductor pair.

Note:

- R_e and L_e are the values on one end of the squirrel cage as shown in the schematic below. Therefore, in the Maxwell 2D application, you would enter 2 ohm and 1 mH for this squirrel cage.
- The current reported as EndConnection1 is the current in the first bar located counter-clockwise from the X axis.
- All objects included in End Connection automatically have **Eddy Effects** enabled.



For unevenly distributed conductors, the user supplied resistance and inductance values are scaled such that the supplied values are used directly for the conductor pair with the smallest angle between them. All other conductor pairs are assigned values that are scaled up according to the following equations:

$$r_{eij} = r_e(\theta_{ij}/\theta_{smallest})$$

$$L_{eij} = L_e(\theta_{ij}/\theta_{smallest})$$

Where θ_{ij} is the angle between the conductor *i* and conductor *j*.

Functional Boundaries and Sources

Functional boundaries and sources have defined by math functions and are used

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- to model distributions of charge, current, or voltage that vary as a function of position.
- to model external fields that vary as functions of position.
- to define voltage, current, charge, or boundary values as variables to be used in a parametric sweep.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

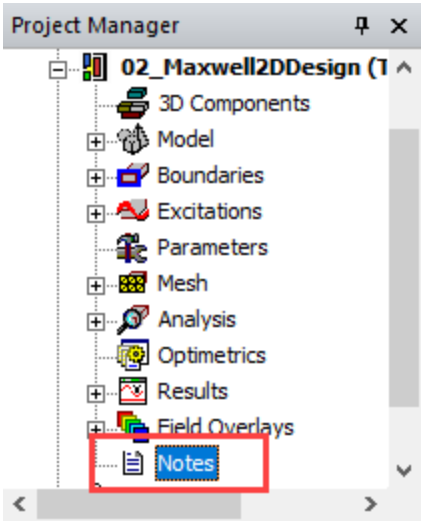
30 - Maxwell Examples

Maxwell provides numerous examples for customer use.

- Maxwell 2D and 3D examples are stored in the **Examples\Maxwell** subdirectory within the **AnsysEM** installation directory.
- RMxprt examples are stored in the **Examples\RMxprt** subdirectory within the **AnsysEM** installation directory.

Maxwell 2D and 3D Examples

Maxwell 2D and 3D example designs are stored in the \\Examples\Maxwell subdirectory within the **AnsysEM** installation directory. Detailed information for each design is included in the **Notes** in the design's Project Manager tree.

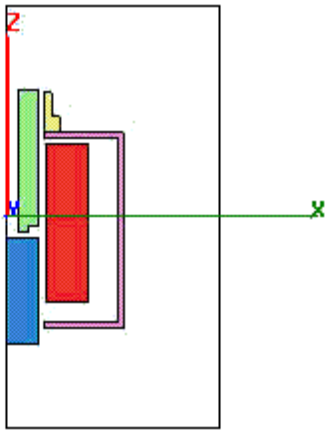


> **Note:** The **Ex_MS_4_ECE_Linear_Movement.aedt** and **WirelessCharging.aedt** designs cannot be used on Linux platforms because they contain TwinBuilder designs. The TwinBuilder application is not supported on Linux.

The designs below, found in the \\Examples\Maxwell subdirectory, are those modeled in the Maxwell *Getting Started Guides*.

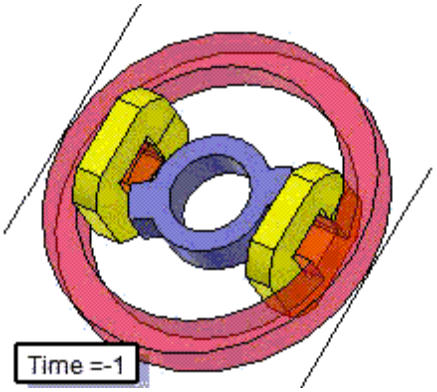
\\Examples\Maxwell\Actuators\ Solenoid.aedt	The solenoid model shown is a 2D axisymmetric simulation of a 3D actuator that demonstrates rotational symmetry about a central axis. The problem is a

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



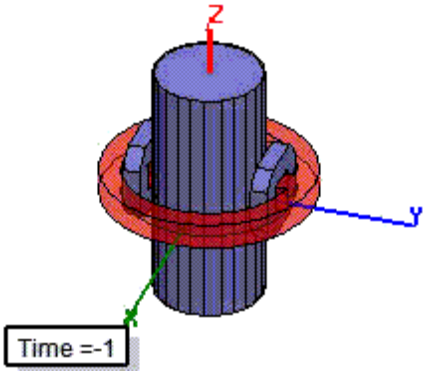
magnetostatic model that uses parametric analysis to determine the force on the core as various locations along the length of the stroke. This is a nonlinear problem incorporating both nonlinear steel and a permanent magnet. This problem is used in *Getting Started with Maxwell: A 2D Magnetostatic Solenoid Problem*.

\\Examples\Maxwell\Actuators\ Rotational_actuator.aedt



The rotational actuator is another magnetostatic example usable for the parametric study of the torque and inductance characteristics vs. position. Possible variation(s) of the setup can include using ferromagnetic materials with different magnetic (nonlinear) characteristics and the respective impact on torque and inductance characteristics. This model is used in Getting Started with Maxwell: Designing a Rotational Actuator.

\\Examples\Maxwell\Actuators\ Rotational_Act_TR.aedt

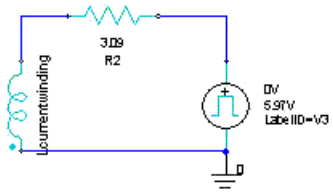


This model is a modified version of the Rotational Actuator model listed above. It is configured with a motion band for rotating the inner armature. It is a transient problem using an external circuit to drive the coils with a pulse voltage. The results demonstrate the torque, position, and current versus time as the inner armature object moves with time. This model is used in the Getting Started with Maxwell: Transient Problem guide.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

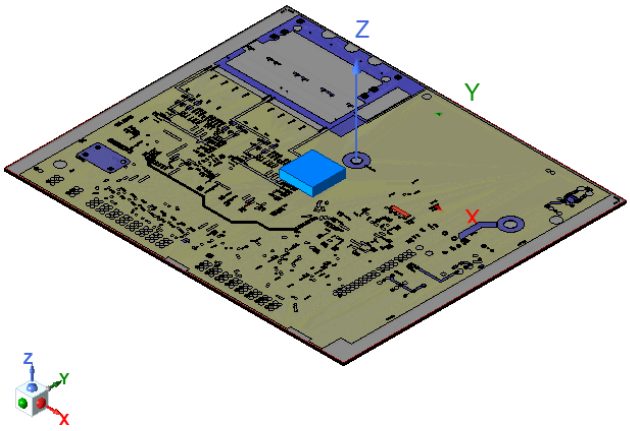
\\Examples\Maxwell\Actuators\ ExternalCircuit.aedt

This project is the Maxwell Circuit Editor model used in the transient rotational actuator problem to drive the coils with a pulse input.



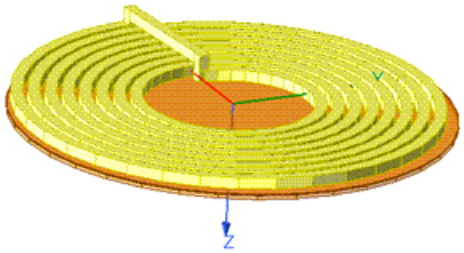
\\Examples\Maxwell\ECAD integration for PCBs\ SNN_ECAD_Maxwell_withMagnet.aedtz

The project archive is the complete model used to simulate the effect a permanent magnet has on the PCB board. If you would like to build the model, the starter archive is located at \\Help\Maxwell. This model is used in Getting Started with Maxwell: Maxwell Integration with ECAD.



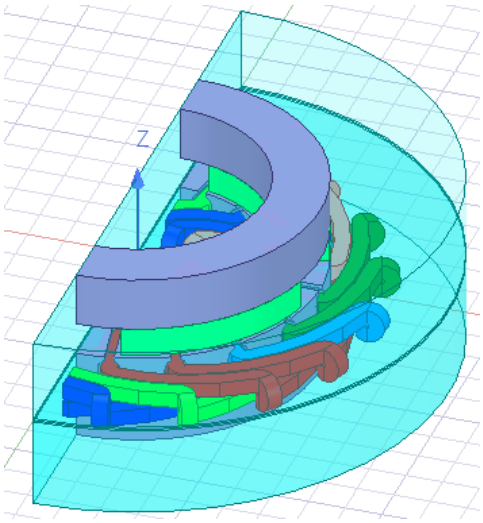
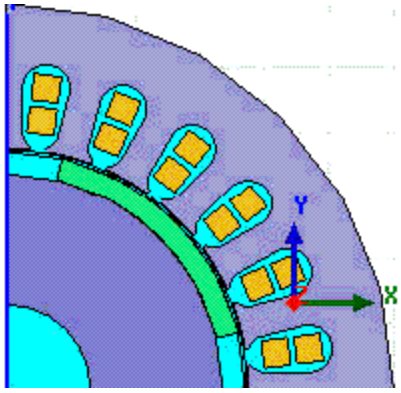
\\Examples\Maxwell\General\AC Magnetic\ CoilwithDisk.aedt

This project is a coil with an iron disk that can be used to model eddy current with skin effects. This model is used in the Getting Started with Maxwell 3D: AC Magnetic Problem guide.

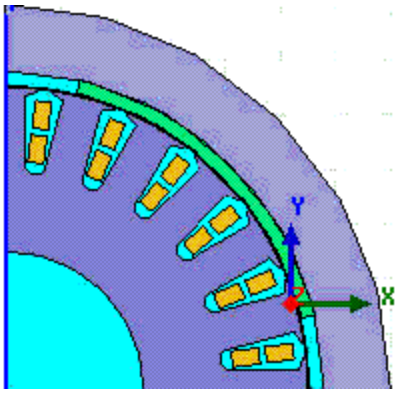


RMxprt Examples

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

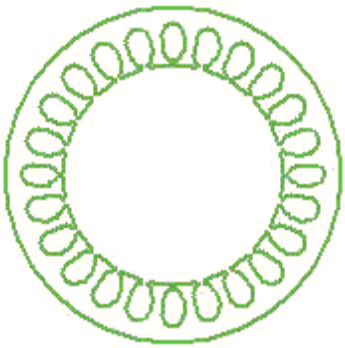
\RMxprt\afpm **afassm_1.aedt** **afassm_2.aedt** **afbldc_1.aedt** **afbldc_2.aedt** **afbldc_3.aedt** **afbldc_4.aedt**	Axial Flux, Permanent Magnet Generic Rotating Machines • 3D representation of **afbldc_2.aedt** is shown 
\RMxprt\assm **assm-1.aedt** **assm-2.aedt** **assm-3.aedt** **assm-4.aedt** **assm-5.aedt** **assm-6.aedt**	Adjustable Speed Synchronous Machine • ASSM can be used in Motor Mode or Generator Mode, this example is in motor mode 
\RMxprt\bldc **QuickStart.aedt** **ws-1.aedt** **ws-2.aedt** **ws-3.aedt** **ws-4.aedt** **ws-5.aedt** **ws-6.aedt**	Brushless Permanent Magnet DC Motor • ws-1 to ws-5 are examples of rotor PM Pole Type 1-5 as defined the "Rotor" section • ws-6 is an example of an outside rotor as shown

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

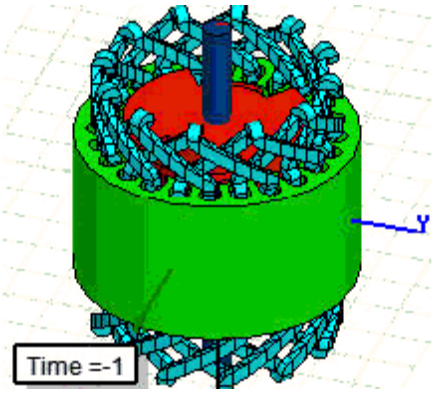


\RMxprt\cpsg\cpsg_1.aedt

Claw Pole Synchronous Machine



3D Representation is shown.



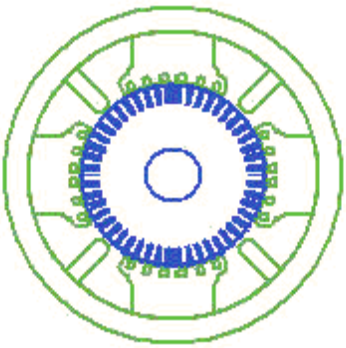
\RMxprt\dcm

z_500.aedt
zf_500.aedt

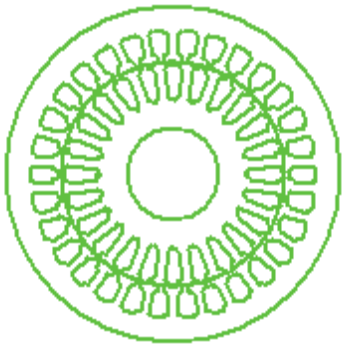
DC Machines

- these do not have any permanent magnets

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

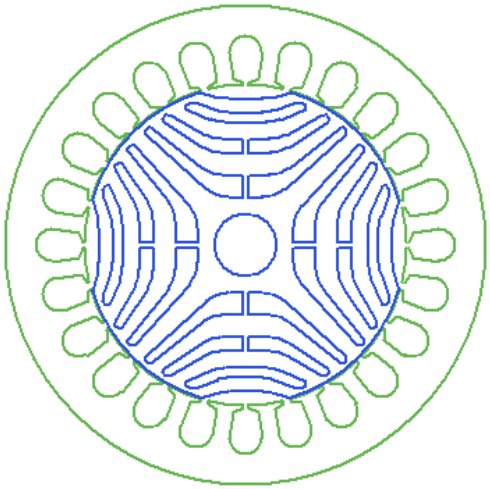


\RMxprt\dfig	Generic Rotating Machines
dfig_1.aedt **dfig_2.aedt**	**dfig_2.aedt** is shown.

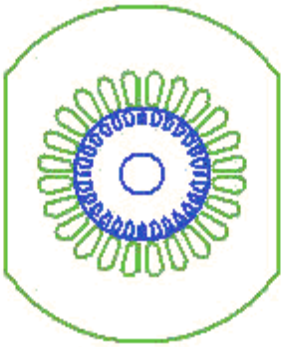


\RMxprt\grm	Generic Rotating Machines
AxialCageRotor.aedt	• SynRRotor is shown
CageRotor.aedt	
SalientPole.aedt	
StepMotor.aedt	
SynRRotor.aedt	

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

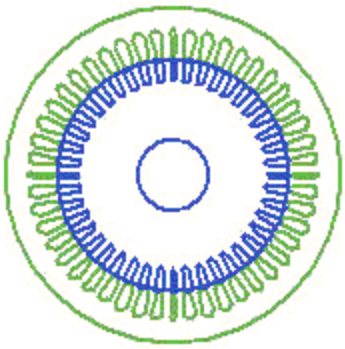


\RMxprt\hysm	Generic Rotating Machines
ipm_HysRotor.aedt	
ipm_SolidRotor.aedt	
\RMxprt\indm1\3hp.aedt	Single Phase Induction Motor



\RMxprt\indm3	Three Phase Induction Motor
tsyde-22.aedt **ylew-95.aedt** **yz200-6.aedt** **yz200-24.aedt** **yzd132-4.aedt** **yzd132-4G.aedt** **yzd132-8.aedt**	• These are 22, 95, 6, 24, 4, and 8 pole motor variations • 8 pole machine is shown.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



\RMxprt\ipm

ipm_1.aedt

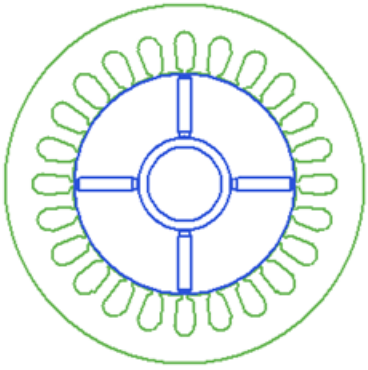
ipm_2.aedt

ipm_3.aedt

ipm_5.aedt

ipm_6.aedt

Generic Rotating Machine (Interior Permanent Magnet); example ipm_1 is shown.

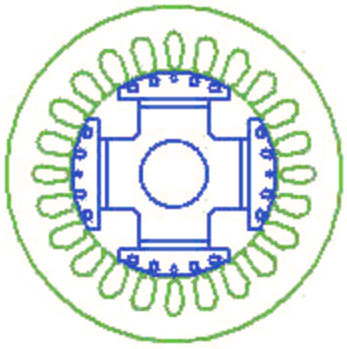


\RMxprt\lssm

sm-1.aedt
sm-2.aedt
sm-3.aedt
sm-4.aedt
sm-5.aedt
sm-6.aedt
sm-7.aedt
sm-8.aedt

Line Start Permanent Magnet Synchronous Motor

- sm-1 to sm-8 represent the 8 rotor pole types available
- sm-8 is shown.

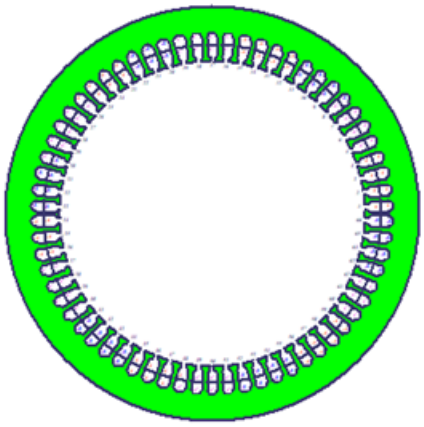


\RMxprt\manual

Example of Asymmetric Winding, p=6, y=11

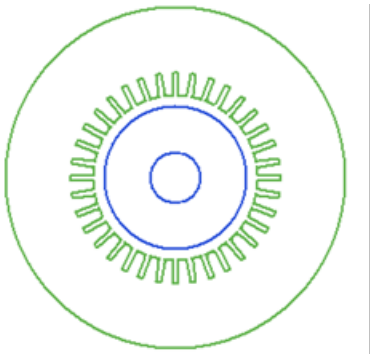
Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ASSM_4p50Hz550W.aedt
AssymetricWinding.aedt
BLDC_
4p1500rpm550W.aedt
CPSG_
4p50Hz550VA.aedt
DCM_
4p1100rpm500kW.aedt
DCM_
4p3000rpm500kW.aedt
IndM1_2p60Hz3hp.aedt
IndM3_6p50Hz11kW.aedt
LSSM_4p50Hz550W.aedt
NSSM_
2p3000rpm125MVA.aedt
PMDC_
2p2400rpm75W.aedt
PMSG_4p50Hz550W.aedt
QuickStart.aedt
SRM_8-
6p1500rpm550W.aedt
SynM3_
6p50Hz538kW.aedt
UniM_
2p12500rpm100W.aedt



\RMxprt\nssm

nssm-1.aedt



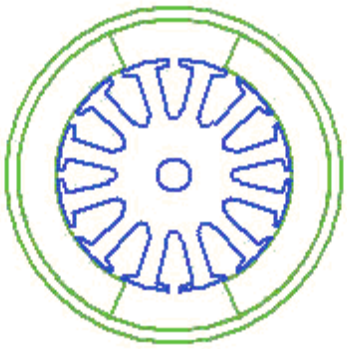
\RMxprt\pmdc

stndj-1.aedt

Permanent Magnet DC Motor

- These are brush type motors

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

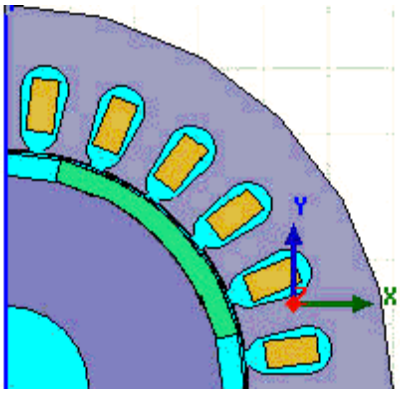


\RMxprt\pmsg

pmsg-1.aedt

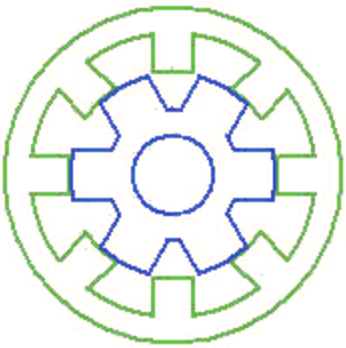
Adjustable Speed Synchronous Machine

- ASSM can be used in Motor Mode or Generator Mode, this example is a generator



\RMxprt\srm\srm-1.aedt

Switched Reluctance Motor



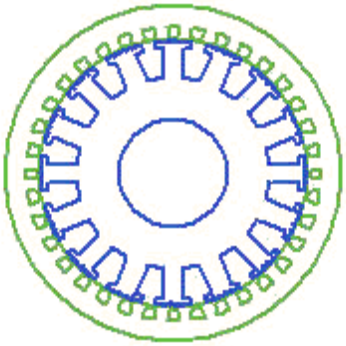
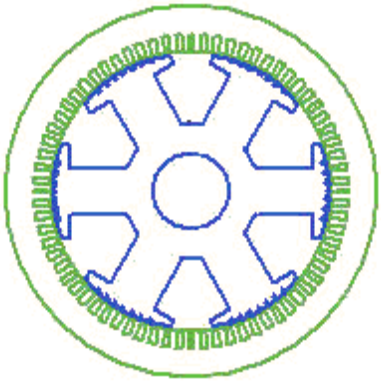
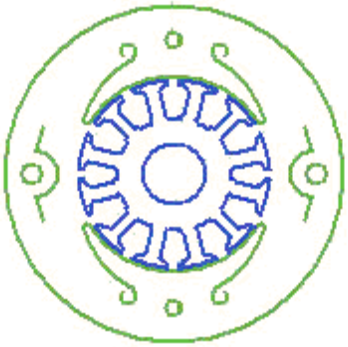
\RMxprt\syng3

bjs7-6-1.aedt
bjs10-61.aedt

Three Phase Synchronous Machine

- This can be either motor or generator mode, these examples are generators

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

bsm.aedt **g2-5.aedt**	• Model g2-5 is shown 
\RMxprt\synm3 **Bjs7_mot.aedt**	Three Phase Synchronous Machine • This can be either motor or generator mode, this example is a motor 
\RMxprt\unim\ac120_i.aedt	Universal Motor Example 

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31 - Maxwell Terminology

This section defines the terminology used in the Ansys Maxwell® help topics. Terms are listed in alphabetical order.

Glossary: A

advanced nonlinear options

For the magnetostatic solution type, the advanced nonlinear options allow you to perform more sophisticated simulations of the magnetostatic field. (Nonlinear materials are materials in which the magnetic flux density varies nonlinearly with the magnetic field.)

anisotropic materials

Anisotropic materials exhibit different values of the respective material property in different directions of the three-orthogonal coordinate system used to define it. Any of the material properties can be anisotropic: electric conductivity, dielectric permittivity, magnetic permeability. Mathematically, tensors are used to express the respective anisotropic properties.

Glossary: B

B-H curve

In nonlinear materials, when a material has a permeability that varies with the flux density, a B-H curve is used to describe the material's nonlinear behavior. The B refers to the magnetic flux density. The H refers to the magnetic field intensity.

Glossary: C

capacitance

Capacitance is a quantity used to describe the ability of a pair of corresponding electrodes to store electric energy. In an equivalent formulation, the storage of energy corresponds to charge accumulation on the respective armatures while a voltage is established between them. Mathematically, capacitance is the positive defined ratio between the charge accumulated on one of the armatures and the voltage between the two armatures.

cascaded windows

Cascaded windows are a set of partially overlapping windows stacked on top of each other.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

coil terminals

A coil terminal is used to apply excitations to coils. For example, it can be a 2D sheet object obtained via a section command (for closed coils that are totally contained within the region where the field problem solution is being computed), or it can be materialized by the intersection between the problem region and open-ended conductors. It is assumed that the current flow is perpendicular to the coil terminal.

conduction path

The conduction path is a sequence of conductors (objects with non-zero conductivity) through which current flows.

conductivity

Conductivity (electric) represents the ability of a material to allow electric current to flow through it.

conductor

Conductors are, in general, objects with non-zero value of conductivity. The way such objects are treated by the software depends upon the user-defined conductivity threshold value.

cost function

In an optimization setup, a cost function is calculated based on goal values specified for one or more solution quantities. The chosen Optimetrics engine automatically changes the design parameter values to fulfill the objective of the optimization. The cost function can be based on any quantity that can be computed directly by Maxwell or by the field calculator.

current density terminals

This refers to any terminal used to specify the current density of an object.

cutplane

A cutplane is any plane that cuts through the model. Cut planes can cut through the model at any angle and may be moved to a new position to observe a difference in plotted fields.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Glossary: D

damping (linear motion)

A value for damping for linear motion can be calculated by the windage and friction loss in Watts divided by the speed in m/sec (at which those losses are defined); this is Damping with resulting units of (W / $(m/sec)^2$) or (N*m * sec/m).

damping (rotational motion)

A value for damping for rotational motion (for electric machines) can be calculated by the windage and friction loss in Watts divided by the speed in rad/sec (at which those losses are defined); this is Damping with resulting units of (W / $(rad/sec)^2$) or (N*m * sec/rad).

design variation

A single combination of variable values that is solved during a parametric or optimization setup.

dielectric loss tangent

The dielectric loss tangent is a material property characteristic for lossy dielectrics, i.e. dielectrics which exhibit power losses.

Dirichlet boundary

The Dirichlet boundary (or value boundary) allows you to prescribe the value of the function (solution) on the respective surface of the solution domain where it is applied. Alone or together with other applied boundary conditions, it ensures the uniqueness of the field solution calculated by Maxwell.

Glossary: E

eddy current

Eddy currents are the currents due to the electric field induced by the time varying magnetic field in conductors.

eddy effect

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The eddy effect refers to the current flow induced by the time-varying magnetic field in conductors. It can be turned on or off depending on the characteristics of the application. Turning the eddy effect off in conductors (when their effects are estimated to be negligible) can save significant computing resources.

electric coercivity

This refers to the electric field that exists when the electric flux is zero.

electric retentivity

The electric retentivity refers to the value of the electric flux density when the electric field is zero.

electric flux

Electric flux is defined as the integral of the electric flux density vector over a specified surface. According to Gauss's law, the electric flux calculated for a closed surface is equal to the total electric charge enclosed by the respective surface.

electric susceptibility

This material property refers to a material's ability to become polarized.

executive parameters

Executive parameters are parameters of great practical significance that are calculated directly by the Maxwell solvers for all specified objects.

Glossary: F

finite element analysis

Finite element analysis (FEA) is a numerical method of solving physical domain partial differential equations (such as Maxwell's equations for electromagnetic fields) by which the solution domain is divided into a set of geometric entities such as tetrahedrons.

finite element mesh

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The finite element mesh for a Maxwell problem is the assembly of all tetrahedral finite elements used to discretize the solution space where the respective electromagnetic fields are calculated.

floating boundaries

In electrostatics, floating boundaries are similar to Dirichlet boundaries in that the entire surface is at a single potential, except that you need not specify the potential as a known a value. This is used to model conductors at unknown potentials.

floating conductor

Floating conductors are conductors at which the potential is unknown.

Glossary: G

Genetic Algorithm

Optimization method of the experiment tool with automatic parameter variation and target function determination.

goal

In an optimization setup, a goal is the value of a solution quantity that you want to be achieved during the optimization. A goal is represented as one row in the cost function table. Each cost function defined in an optimization setup must include at least one goal.

grid plane

The grid plane is the plane on which you want to display the grid in the active view window. Also called the drawing plane.

Glossary: H

hotkeys

Hotkeys are keystroke combinations that execute more complex commands without the need for accessing the commands in the menu bars or tool bar icons.

Glossary: I

imaginary permeability

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This material property refers to the imaginary component of the permeability of a material. See **permeability**.

imaginary anisotropic relative permeability

This material property refers to the imaginary component of the anisotropic relative permeability of an object. See **anisotropic materials**.

imaginary relative permeability

This material property refers to the imaginary component of the relative permeability of a material. See **relative permeability**.

impedance

Impedance represents the characteristic of an electric circuit element to impose a certain dependency between the voltage between a pair of terminals and the corresponding AC current flowing between the two terminals.

independent generator

The generator supplies electrical power to a load inductor. For an AC generator, the resistance and inductance of the load inductor are determined by the rated output power, the rated voltage, and the rated power factor. For a DC generator, the resistance of the load resistor is determined by the rated output power and the rated voltage.

inductance

Inductance represents the property of a magnetic circuit to relate its magnetic flux to the current which produced it. It is the expression of the property of magnetic devices to store magnetic energy.

infinite bus

When the generator is connected to an infinite bus of the power system, the terminal voltages remain constant and the output power is determined by the output currents.

insulating boundary

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

An insulating boundary is a convenient way of preventing the current flow across the boundary. This boundary condition is placed between conductors and allows you to avoid creating small separating spaces between conductors for insulation purposes.

insulator

An insulator refers to a material that does not allow the current to flow through it (typically with zero electric conductivity). The way such objects are treated by the software depends upon the user-defined insulation threshold value.

isotropic materials

Isotropic materials refer to materials with properties independent of the direction (orientation) of the axes of the coordinate system in which they are defined.

Glossary: L

lamination model

Lamination modeling allows you to specify a stacking factor and stacking direction, which represents the direction perpendicular to the plane of the lamination. This lamination model is an alternative way of specifying anisotropic behavior when using laminations.

line impedance

The line impedance of an object is the impedance of a single current loop.

LMB

This is an acronym for the left mouse button. The left mouse button is used to choose commands and select points, surfaces, and objects.

Lorentz force

The Lorentz force is the force on an object due to the currents in the magnetic field, neglecting forces due to the effects of material properties.

Lorentz torque

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Lorentz torque is the torque of an object due to the currents in a magnetic field, neglecting torques due to the effects of material properties.

lumped capacitance

Lumped capacitance is the capacitance between two conductors or groups of conductors.

Glossary: M

magnetic coercivity

This refers to the magnetic field that exists when the magnetic flux is zero.

magnetic flux

The magnetic flux is the total of all the lines of force in a magnetic field.

magnetic loss tangent

The magnetic loss tangent allows you to represent a dielectric that dissipates the power of a high-frequency magnetic field. The smaller the loss tangent, the less lossy the material.

magnetic retentivity

In nonlinear materials, the magnetic retentivity (or remanence) refers to the value of the magnetic flux density when the magnetic field is zero.

magnetic susceptibility

The magnetic susceptibility is a measure of the ability of the material to become magnetized.

matching boundaries

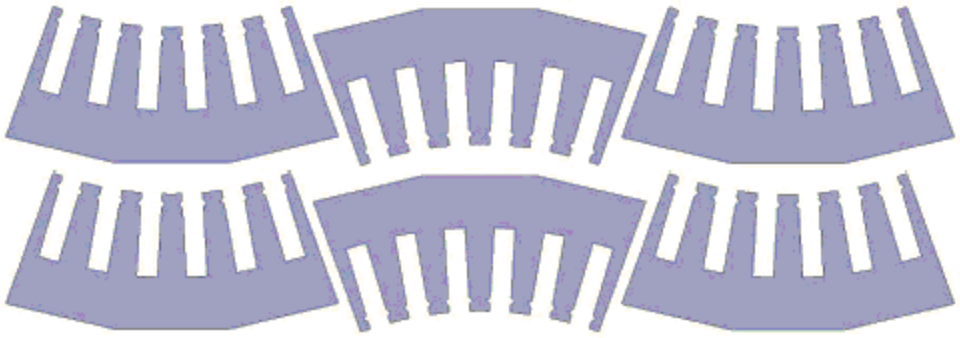
Matching boundary conditions force the field solution at one surface (the Dependent surface) to match the magnitude and orientation (or opposite orientation) of the field at the Independent surface.

material consumption

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Whole-circle laminations – Considers a minimum section of square “raw steel” from which the lamination could be cut. The calculation takes the rotor or stator diameter as the length of the square and calculates the weight based on the material density.

Lamination sectors – For large size machines, a lamination layer of the stator core may be laminated sector-by-sector, instead of a whole circle lamination. The number of Lamination Sectors indicates how many lamination sectors are needed to form a lamination layer. Lamination sectors are punched as shown below so that material consumption is minimized. There is a 3 mm allowance in the edge and between two sectors.



material coordinate system

The material coordinate system is the coordinate system used to define the vector or anisotropic material property of objects. Maxwell supports the following three material coordinate systems: Cartesian, cylindrical, and spherical.

mesh refinement

Mesh refinement refers to the process of successively breaking down large error finite elements into smaller ones for the purpose of continuously increasing the accuracy of the finite element calculation. Maxwell uses proprietary adaptive mesh refinement algorithms designed to automatically detect and refine the mesh in regions with the largest field error.

multi-processing

Using more than one processor to solve a Maxwell design. Requires a multiple processor license. Using multiple processors (if physically available on the computer) speeds up the matrix solution process phase. Thus, shorter solution times can be achieved with significant saves in absolute terms, particularly for large models of 100,000 finite elements or more. There are no negative side effects if the number of processors to be used is set to more than

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

are available on the respective computer. In this case, the actual number of available processors will be used in the matrix solution process.

Glossary: N

natural boundary

A natural boundary is a boundary condition at the interface between different objects. You do not need to apply this boundary condition yourself, as Maxwell enforces it internally. The natural boundary condition reflects the particular local form of Maxwell's equations at the interface between objects where there is an sudden change in the material properties. When crossing such a surface of discontinuity, Maxwell assumes the following behavior (in average):

- The normal component of B is continuous.
- The normal component of D has a jump equal with the local superficial charge density (on the discontinuity surface).
- The tangent component of H is continuous if there are no current sheets on the discontinuity surface (has a jump equal with the density of the current sheet if they are present on the discontinuity surface), and so on.

Neumann boundary

For magnetic problems with T-Omega formulation, the homogeneous Neumann boundary condition specifies that the normal component of the H field is zero. For magnetic problems with A-Phi formulation, the homogeneous Neumann boundary condition specifies that the tangential component of the H field is zero. If the outermost surfaces of the problem region are not assigned any boundary conditions, they automatically receive a Neumann boundary condition. Because of this, as a rule, the solution space (region size) should be chosen such that the respective region walls are not close to field sources (currents and/or permanent magnets).

nodes

Nodes normally refer to the vertices of the tetrahedral finite elements. They can also refer to the connection points of branches in an electric circuit (if external circuits are used to drive coils).

nominal design

The original model on which Optimetrics analyses are based.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

nonlinear materials

Nonlinear materials are materials with the constitutive relationship being nonlinear (such as is the case of the dependency between B and H for some magnetic materials).

Glossary: O

outer terminals

Typically, outer terminals are faces of 3D objects that lie on the surface of the problem region.

Glossary: P

parametric analysis

A parametric analysis consists of a sweep of one or more design or project variables for the purpose of analyzing one or more output quantities (Maxwell parameters or quantities calculated by the field calculator).

paste buffer

When you cut or copy an object, such as with the **Edit** commands, the object is stored in the paste buffer. The paste buffer itself is an area of resident memory that is erased and replaced with the newest item that has been cut or copied.

perfect conductor

Perfect conductors are idealized materials with very high conductivity (typically many orders of magnitude above the usual conductivity values). The actual value is influenced by the perfect conductivity threshold value you specify in the solve options.

permanent magnets

Permanent magnets are any materials that generate magnetic flux due to permanent magnetic dipoles in those materials.

permeability

Permeability is a material property characteristic of magnetic materials and relates magnetic flux density to magnetic field strength in objects with no permanent magnetization present.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

permittivity

Permittivity is a material property characteristic for dielectrics and relates electric flux density to electric field strength in objects with no permanent electric polarization present.

phasors

Phasors represent a conventional way of representing sinusoidal (AC) quantities such as voltages and currents. In a phasor representation, the amplitude of the signal relates to the length of the rotating vector (phasor) while the initial phase is the starting angle at time t = 0. The angular velocity of the phasor rotation relates to the frequency of the signal.

polyline

A polyline is an object composed of one or more straight or curved line segments.

postprocessing

Postprocessing is the phase of the analysis when information is extracted from the simulation. Typical results are 2D plots, contour and vector plots of field quantities, results from the field calculator, animations, *etc*.

press board (pressboard)

An optional material used above and below the lamination stack and as a spacer in manufacturing. In RMxprt, users can specify the press board thickness and whether it is magnetic or non-magnetic.

problem region

The problem region is the region where Maxwell calculates the field solution. This is an all-encompassing object with user-selected material properties and with surfaces that are used to apply the necessary boundary conditions (user-specified or default ones).

Glossary: R

relative permeability

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Relative permeability relates the absolute permeability of a magnetic material to the permeability of vacuum ($4\pi 10^{-7}$ H/m).

relative permittivity

Relative permittivity relates the absolute permittivity of a magnetic material to the permittivity of vacuum(8.85× 10-12 Farad/m).

resistance

Resistance represents the characteristic of a DC electric circuit element to impose a certain dependency between the voltage between a pair of terminals and the corresponding DC current flowing between the two terminals.

RMB

This is an acronym for the right mouse button.

RMBM

This is an acronym for the right mouse button menu. This menu becomes visible when you hold the right mouse button down while the cursor is in a view window. Depending on the module you are in, you may use this menu to select or deselect objects, or accept or cancel selections or values.

Glossary: S

section

This refers to the **Surfaces/Section** command which creates a cross-section of 3D objects with the XY, YZ, or XZ planes.

seeding

Seeding refers to the process of adding points to your finite element mesh to create a finer one.

self-inductance

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Self-inductance of an object or group of objects associates the current flowing in the respective object(s) with the own flux linkage.

skin depth

The skin depth of a material is the distance measured inside a conductor from the surface where the AC field (magnetic/electric) penetrating the conductor is reduced by a factor of 1/e (~ 1/2.71). As general rule is that at 5 skin depths from the surface, the AC fields penetrating inside conductors can be considered to be zero.

$$\delta = \sqrt{\frac{2}{\omega \sigma \mu_0 \mu_r}}$$

slot fill factor

The slot fill factor is the ratio between the cross-sectional area of all conductors in one slot and the entire slot area.

solver

Maxwell has a number of solvers allowing you to choose from electric, magnetostatic, eddy, and transient, depending upon the characteristics of the problem to be solved.

solver residual

The solver residual is an error term that measures the current accuracy of the solution process when using an iterative solver.

superposition

Superposition allows you to combine the results of field solutions, each corresponding to linear problems. Superposition should not be used when problems are nonlinear.

surface

A surface is an object face.

sweep

Sweep can refer to two different topics:

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- In the 3D Modeler, sweeping a face or object along a path or vector creates a new 3D object. The object you sweep need not be orthogonal to the path or vector.
- In the Executive Parameters module, a parametric sweep refers a series of variables.

symmetry boundary

Symmetry boundaries model the planes of symmetry in a problem.

Glossary: T

terminal

As a terminal, you can use an exact cross-section of the conduction path, to apply an eligible excitation.

tile window

Existing windows, regardless of their contents, can be tiled either horizontally or vertically.

toolbar

A toolbar contains icons which can be used to shortcut commands.

Glossary: V

value boundary

See **Dirichlet boundary**.

variable sweep definition

A set of variable values within a range that Optimetrics drives Maxwell to solve when a parametric setup is analyzed. A parametric setup can include one or more sweep definitions.

vector functions

Vector functions are used to specify certain vector type properties. For instance, magnetization can be defined using vector properties in a chosen, previously defined coordinate system.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

view window

In the 3D Modeler, the view window is the window in which you create your model. The view window gives a 3D view of the model. In this window, you can assign materials to objects, specify boundary conditions, set up any executive parameters, or post process data or fields in the model.

virtual force

Virtual force is the force calculated on an object or group of objects using a special mathematical algorithm that contains the effects of a virtual motion along a generalized coordinate (such as x, y, z). The movement does not actually take place; it is virtual.

virtual torque

Virtual torque is the torque calculated on an object or group of objects using a special mathematical algorithm that contains the effects of a virtual rotation around a coordinate axis (such as x, y, z). The movement does not actually take place; it is virtual.

visibility

Visibility refers to whether or not an object is displayed. You can show or hide an object using the **View** > **Active View Visibility** commands.

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Index

.

.amat file 13-72

#

2D AutoCAD files, exporting 3-144

3D AC conduction solver

theory 28-79

3D modeler window 2-95

history tree 2-97

commands 2-111

controlling the view 2-100

organizing 2-99

selecting objects 2-106

upgrading modeling kernel 2-112

3D models

changing background 8-2

clip planes 8-52

curved surface 8-51

fitting 8-37

hiding objects 8-39

lighting 8-56

modifying view 8-1

orientation 8-18

panning 8-8

rendering solids 8-47

rendering wireframes 8-47

rotating 8-25

showing objects 8-41

spinning 8-25

user interface options 8-3

zooming 8-9

3D structures

managing 13-53

3D transient solver

A-Phi Formulation 28-95

flux linkage 28-94

A

aborting an analysis 18-240

aborting analyses 26-102

AC conduction solver 29-1

admittance 28-84 , 29-35

admittance matrix 28-85

theory 29-34

ACT extensions 2-44, 2-158

adaptive analysis

percent refinement per pass 15-6

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

setting up 15-5

Adaptive Pass, Plotting Versus 24-87

adaptive passes, completed and remaining 24-4

adjustable-speed permanent-magnet synchronous motors

general data 26-238

general procedure 26-237

stator data 26-240, 26-363, 26-387

winding type 26-243

adjust-speed synchronous machine, general data 26-239

adjust-speed synchronous machines, transistor drop 26-239

admittance 28-84 , 28-85 , 29-35

Ampere’s Law 29-11

analyses

re-solving 26-102

starting 21-1 , 26-102

stopping 26-102

analysis

aborting 18-240

distributed 18-9

parametric (large scale DSO) 18-28

remote 18-1

analysis configurations

in the registry 3-94

analyze all 22-3

ANFV4, Translations of Padstacks to 13-69

animations 24-22

field 24-27

anisotropic materials

about 28-128

and ports 28-132

conductivity tensor 28-130

electric loss tangent tensor 28-130

magnetic loss tangent tensor 28-131

permeability tensor 28-128

permittivity tensor 28-129

anisotropic tensors 6-37

anisotropy

conductivity 28-130

electric loss tangent 28-130

magnetic loss tangent 28-131

permeability 28-128

permittivity 28-129

anisotropy tensors

defining conductivity 6-34

defining dielectric loss tangent 6-30

defining magnetic loss tangent 6-50

defining relative permeability 6-27

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

defining relative permittivity 6-31
Ansys Discovery
integration with 7-163
link options 3-35
Ansys Electronics Desktop
batchoptions 2-125, 2-145, 2-154
files 3-1
getting started with 2-1
launching 2-5
menu bar 2-49
overview 2-11
projects 3-1
ribbon interface 2-11, 2-13, 2-159
running from a command line 2-125, 2-145, 2-154
running from Windows remote terminal 2-156
status bar 2-113
system requirements 2-5
user interface 2-11
window displays 2-17
Ansys HPC Platform Services Support for HPC Job Management 18-190
Ansys Product Improvement Program 2-7
Ansys SpaceClaim
integration with 7-171
Ansys technical support 1-12
Ansys Workbench
feedback iterator 19-42
integration with 19-1
material data transfer 19-35
archive
restoring 3-156
saving project as 3-153
arrows
modifying in plots 24-220
types in plots 24-220
AutoCAD Drawing Interchange Format files 3-144
auto-complete
variables and properties 2-120
auto-complete, variables and properties 2-120
auto-save 3-112
average force 28-67
axes, torque about 28-50
axisymmetric models
electric flux linkage 29-10
field simulation for 29-2
magnetostatic flux linkage 29-16

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

B

background color 8-2

balloon boundaries

charge at infinity 29-50

used as reference 29-63

voltage at infinity 29-50

Band UDPs 9-4

3D independent and dependent boundaries 9-13

creating a band 9-6

creating a tool 9-8

outer region 9-7

Band.cpp 9-284

Band.h 9-283

batchoptions 2-125, 2-145, 2-154

command line examples 3-90

desktop settings 2-139

directories and library paths 2-138

examples 2-135

file format 3-90

in the registry 3-94

override registry entry 2-145

temp directory 2-139

beta options 3-8

BH-curve 28-40

for permanent magnets 6-111 , 6-111

bondwires

drawing 7-77

Boolean operations 9-258

boundaries

duplicating with geometry 10-32

hiding 10-33

reprioritizing 10-36

setting default values 10-33

boundary conditions

eddy current 10-77

functional 29-64

brush data, DC motors 26-273, 26-378, 26-405

brushless permanent-magnet DC motors

available circuits 26-237, 26-308, 26-331

circuit type 26-238, 26-308, 26-331

general data 26-308, 26-311, 26-312, 26-313, 26-322, 26-325, 26-326, 26-332

general procedure 26-307

rotor pole data 26-254, 26-262, 26-356, 26-364, 26-388

pole embrace 26-254

stator data 26-311, 26-334

conductors 26-266, 26-313, 26-372, 26-398

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

end length adjustment 26-248, 26-269, 26-318, 26-350, 26-374, 26-400, 26-418

stator windings 26-266, 26-313, 26-372, 26-398

transistor drop 26-309

trigger angle 26-309, 26-332

C

calculated expressions, plotting 24-187

calculator

opening 24-242

capacitance

and electric field 29-5

calculating 29-8

in terms of charges and voltages 29-5 , 29-5

in terms of currents and time-varying voltages 29-7

short circuit 29-7

changing motor or generator machine type 26-31, 26-35

charge

and balloon boundaries 29-50

and reference voltage 29-63

charge density 29-3

Charge Standard Field Quantities, Surface 24-163

chip model analyzer (CMA) 24-395

ClawPoleCore UDPs 9-143

coil and terminals 9-145

inner core 9-145

magnet 9-147

clean stop 18-240, 22-7, 26-102

cleaning up solutions 24-21

clip plane 8-52

clip planes 8-55

clipboard

importing model from 7-244

options 3-50

closing

Property Window 2-41

Code implementation

creating primitives 9-267

mapping parameters 9-268

parameter validation functions 9-265

UDP parameters 9-263

validating parameters 9-266

coenergy

and virtual force 28-28 , 28-50 , 29-15

and virtual torque 28-29 , 28-51 , 29-16

coil terminals

ordering for A-Phi formulation 5-7 , 5-7

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

color scheme
changing 3-12
command properties
modifying 2-36
command-line syntax 2-125, 2-145, 2-154
commands
align faces 7-127
chamfer 7-298
copy 7-243
cut 7-247
fillet 7-296
heal 7-119
hide objects 8-2
pan 8-8
paste 7-243
perform explicit subtractions 7-129
project sheet 7-278
purge history 7-307
redo 3-162
remove edges 7-129
remove faces 7-129
rotate view 8-25
scale 7-258
separate bodies 7-295
show objects 8-2
simplify 7-112
spin view 8-25
stitch sheets 7-127
thicken sheet 7-269
undo 3-162
wrap sheet 7-270
zoom 8-9
commutator 26-273, 26-379, 26-405
commutator type
cylinder 26-273, 26-379, 26-405
pancake 26-273, 26-379, 26-405
complex numbers
admittance 28-84 , 28-85 , 29-35
impedance 29-23
phasor notation for 29-43
real and imaginary components 29-43
component
creating, editing 13-12
creation sequence 13-9
editor 13-11
libraries 13-1
parameters, reserved 13-24
Component CosimDefinition property 13-25
component libraries 7-171
Component properties, copying and pasting 27-15

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

components
creating, editing 13-11, 13-12
editing 13-12
editing definitions 7-226
editing properties 7-223
human bodies 7-233
inserting 7-205
legacy projects using 7-239
updating 3-161
updating definitions 7-232
viewing properties of 7-217
wireless power transfer 7-236
Components Tab 27-11
ConCoil UDPs 9-46
single concentric coil 9-52
conductance 29-32
in admittance matrices 29-34
conduction paths
recalculating 10-32
showing 10-31
conductions paths, verifying 10-31
conductivity 28-124
and skin depth 29-22
anisotropic 28-129 , 28-129
defining anisotropy tensors 6-34
conductors
parallel, explicitly defining (AC current source) 29-60
passive 29-59
skin depth in 29-22
stranded 29-36
touching, assigning AC currents to 29-59
configuration files
example uses 3-80
user options 3-82
using to set options 3-71
using UpdateRegistry command 3-75
constants
list of 3-192
convergence
in solution process 28-3
output variable 15-8
viewing output variable 24-5
convergence data
number of passes completed 24-4
output variable 24-5
plotting 24-5
coordinate system, effect on field simulation 29-2
coordinate systems
default planes 7-360

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

deleting 7-368

face 7-358

managing 13-53

modifying 7-366

moving to end 7-365

object 7-361

overview 7-349

relative 7-351

working 7-350

coordinates

assigning to objects 7-334

expressing 7-367

setting coordinate systems 7-349

copying and pasting objects 7-243

core loss

transient solvers 29-41

Core Region UDPs 9-15

inner core region 9-15

outer core region 9-15

Core regions 9-15

cost function

setting a goal 25-108

Coupling 17-1

ANSY Structural 17-13

Ansys Fluent 17-112

Ansys Structural 17-16

Ansys Thermal 17-3 , 17-16

HFSS 17-83

Magnetostriction 17-115 , 17-116

Twin Builder 17-107 , 17-110

coupling coefficient matrix 24-15 , 24-15

CPU time

viewing for solution tasks 24-5

vs. real time 24-5

creating

components 13-9, 13-11

new footprint 13-49

new padstack 13-58

new symbol 13-29

creating a quick report 26-148

creating motor or generator models 26-29

creating new projects 26-2

creating, custom report templates 24-154

cross-sections

creating 7-277

CupCoil UDPs 9-63

coil and terminals 9-65 , 9-72 , 9-74 , 9-75 , 9-77 , 9-79

curl, axisymmetric vs. cartesian 29-2

current

and capacitance 29-7

and inductance 29-12

and virtual force 29-15

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

constraints on passive conductors 29-59
instantaneous 29-24
relationship to current density (eddy current solver) 29-21
source 29-19
time varying, and inductance 29-14
current density 29-10
AC 29-26
DC (magnetostatic) 28-65 , 29-25
displacement currents 29-18
eddy current 28-65
eddy currents 29-18
in Ampere's Law 29-10
in stranded AC current sources 29-60
relationship to electric field (DC conduction) 29-29
source current 29-19
current distribution, effect on inductance and resistance 29-25
current flow
for DC conduction solver 29-33
for eddy current (AC magnetic) solver 29-27
current sources
AC
assigning to touching conductors 29-59
constraints on current in solid and parallel sources 29-60
parallel 29-60
perfect conductors 29-60
reference for computing A-field 29-63
solid 29-60
stranded 29-60
DC, reference for computing A-field 29-63
currents
and Lorentz torque 28-50
and virtual force 28-28 , 28-50
curves
equation-based 7-24
settings 3-44
tooltip 3-50
custom report templates, creating 24-154
customize
user tools 2-157
cut edge effects on core loss 6-72
cylinder commutator type 26-273, 26-378, 26-405

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

D

data markers 24-229

data tables

creating 26-137

dataset expressions 3-249

Dataset preview plot properties 3-221

datasets

adding 3-217

cloning 3-225

editing 3-223

exporting 3-228

importing 3-221

overview 3-216

plot properties 3-230

removing 3-229

using SheetScan 3-233

DC conduction solver 29-1

conductance 29-32

current flow 29-33

theory 29-30

when to use 29-32

DCMCore UDPs 9-91

commutating coil and terminals 9-102

commutating poles 9-101

frame 9-101

main poles 9-101

outer core 9-100

series coil and terminals 9-101

shunt coil and terminals 9-101

debug logging 2-160

Debye's model 28-132

defaults

setting for boundary values 10-33

setting for excitation values 10-33

defining

footprint handles 13-57

definition archives

exporting 13-3

importing 13-6

importing and exporting 13-3

definition layer, initial padstack 13-68

definition libraries 13-1

definitions, updating on the library 13-3

deleting

field overlay plots 24-232

delta H 28-128

delta temperature units 3-251

Demagnetization coefficient 24-169

Demagnetization Curve 6-122

derived field quantities, plotting 24-187

design area 2-95

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

design of experiments 25-154
Design Settings 4-5
Design Settings in RMxprt 26-35
designs
copying and pasting 3-152
deleting 3-160
design notes 3-163
in project tree 2-25, 26-7
inserting in project 4-1
nominal 25-1
read-only 3-160
setting up 4-1 , 26-149
updating components 3-161
DesignXplorer 25-231
desktop
menu bar 26-5
desktop configuration 3-8
targeted 3-8
Desktop Configuration 7-134
desktop-engine connection 3-20
dielectric loss tangent 28-125
defining anisotropy tensors 6-30
dielectrics, time constant 29-32
directories
options 3-16
system, user and personal 13-1
DiskCoil UDPs 9-204
lap coil and terminals 9-207
whole winding 9-207
DiskPMCore UDPs 9-208
all magnets 9-211
axial-flux core 9-210
one magnet 9-211
DiskSlotCore UDPs 9-199
axial-flux slot core 9-203
squirrel-cage winding 9-204
displacement current 29-19
in solid and parallel AC current sources 29-60
relationship to AC current density 29-18
relationship to AC electric field 29-60
display types, of reports 26-115
distributed analysis
command line options 18-69
configuration 18-10
optimal settings 18-26
overview 18-9
two-level distribution 18-27
divergences, axisymmetric vs. cartesian 29-2
documentation file
inserting 3-163

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DoubleCage UDPs 9-59
bars only 9-62
two rings only 9-61
whole double-cage winding 9-61
draw menu, symbol 13-31
drawing objects
polylines 7-20
DSO
large scale 18-28
command line syntax 18-37
configuration 18-46
monitoring 18-45
outputs 18-40
post-processing 18-42
prerequisites 18-30
troubleshooting 18-60
user interface 18-31
Duplicate operations 9-257
duplicating
boundaries 10-32
excitations 10-32

E

eddy current 29-18
and skin depth 29-22
effect on AC inductance and resistance 29-25
in solid and parallel AC current sources 29-60
relationship to AC current density 29-18
relationship to AC magnetic field 29-60
eddy current field simulation, magnetic field energy 28-56 , 28-58
eddy current solver 29-1
current flow 29-28
derivation of field equations 29-19
impedance 29-23
theory 29-17
virtual force 29-26
virtual torque 29-27
eddy currents, effect on AC inductance 28-65
Edge Loss Density 11-34
edges
creating objects from 7-290
detaching 7-276
flattening 7-298
moving 7-282
removing 7-129
rounding 7-296
selecting 7-324
Edit Components window 13-12
general tab 13-13
miscellaneous tab 13-14
Solver on Demand tab 13-17

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

terminals tab 13-16
Edit Libraries window 13-85
Edit Padstack Definition window 13-59, 13-64
edit symbol pins 13-41
editing
components 13-12
materials 13-72
editing an external circuit 27-92 , 27-92
Editing Pin Properties 13-40
editor
component 13-11
library 13-84
material 13-72
models 13-71
padstack 13-58
script 13-48
symbol 13-28
electric field 29-3
and capacitance 29-5
static 29-4
electric flux density 29-3 , 29-3 , 29-4
electric flux, flux linkage 29-10
electric potential
reference for 29-63
theory 29-3
electric transient analysis, setting up 15-88
electrostatic solver 29-1
capacitance 29-5
flux linkage 29-10
specifying reference for potential 29-63
theory 29-4
virtual force 29-8
when to use 29-32
encrypted
libraries 13-74, 13-76
passwords 13-76
encryption
3D components 7-188
EndCap UDP 9-242
energy
and impedance 29-24
and virtual force (electrostatic) 29-8
errors
disk space warning 3-17
errors,logging 2-160
even symmetry boundaries
behavior of magnetic field 29-49
in magnetostatic models 29-49
relationship to Neumann boundaries 29-49

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Excitations 29-55
duplicating with geometry 10-32
hiding 10-33
setting default values 10-33
exciter efficiency 26-303
executive module 29-1
export a circuit 21-1 , 21-4
exporting
2D AutoCAD files 3-144
2D model files 3-144
3D model files 3-145
datasets 3-228
definition archives 13-3, 13-3
field overlay plots 24-231
graphics files 3-147
hierarchical components 13-87
materials 6-94
options files 3-149
exporting a netlist 27-92 , 27-92
Exporting Hierarchical Components 13-87
Exporting Reports 24-151
exporting winding data 26-186
expressions
dataset 3-249
delta temperature units 3-251
intrinsic functions 3-246
operators 3-245
overview 3-245
piecewise linear functions 3-247
external tools, customize 2-157

F

faces
aligning 7-127
covering 7-275
creating objects from 7-289
detaching 7-276
moving 7-282
removing 7-129
selecting 7-319
uncovering 7-276
fast calculation-update 25-191
ferrite materials
adding to libraries 6-12
properties of 6-13
ferrites
about 28-127
and delta H 28-128
and lande G 28-127
and magnetic saturation 28-127
FFT 24-144
field overlay plots 24-162 , 24-180 , 24-187
modifying field quantities 24-198

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

modifying phase 24-158

on lines 24-229

scalar plots 24-190

vector plots 24-195

field plot animations 24-27

field plots

exporting 24-233

field quantities

phase angle 24-269

plotting basic 24-158

plotting derived 24-187

field simulation

eddy current (AC magnetic) 5-2

electrostatic 5-19

magnetostatic 5-2

field solutions

saving for a tuning analysis 25-221

fields calculator

complex vector electric field 24-270

general commands 24-267

input commands 24-256

output commands 24-281

overview 24-238

registers 24-253

scalar commands 24-270

stack commands 24-245

vector commands 24-276

fields quantities

inception voltage evaluation 24-183

plotting field line trace overlays 24-182

plotting noise vibration 24-177

plotting standard 24-159

plotting time averaged 24-174

file formats

.amat 13-72

.dsp 24-231

.dxf 3-144

.q3dx 26-3

.q3dxresults 26-3

files

about 3-1

exporting 3-144

importing 3-115

Q3D Extractor 26-3

fitting objects 8-37

flux density, electric flux density 29-4

flux linkage

electrostatic 29-10

magnetostatic 29-16

footprint

creating 13-49

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

editing existing 13-50

editor 13-49

load into project 13-50

Footprint Edit Menu 13-50

footprint handles, defining 13-57

Footprint View Menu 13-52

footprint, handles 13-57

force

virtual (Eddy Current) 29-26

virtual (electrostatic) 28-28 , 29-8

virtual (magnetostatic) 28-50 , 29-15

frequency

and impedance 29-23

and impedance matrix 28-62

and line impedance 28-64

and skin depth 29-22

frequency-dependent materials, and lossy dielectrics 28-132

Function library 9-258

Function pointers 9-254

functions

defining 3-252

intrinsic 3-246

list of 3-252

piecewise linear 3-247

selecting for a quantity 26-140

G

Gauss's Law 29-4

GDSII

importing 3-116

layer mapping files 3-119

tech file colors 3-120

generic rotating machine

field winding type 26-490

winding type 26-501

generic rotating machines

general data

output power 26-514

rated voltage 26-515

winding data

parallel branches 26-491, 26-503

wire diameter 26-492, 26-504

wire gauge 26-492, 26-504

wire wrap 26-492, 26-504

goal

setting weight of 25-108

goal weight, setting 25-108

GPU 18-23

gradients, axisymmetric vs. cartesian 29-2

Graphical objects, copy and paste properties 27-16

grid engine (GE) 18-113

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

grids
settings 3-45
grounded systems, modeling with balloon boundaries 29-50
grouped objects, and parallel current sources 29-59
grouping
options 3-34
groups
selecting 7-330

H

HairpinCoil UDPs 9-67
Helix UDPs 9-243
help 2-38
Help for Components 27-13
HFSS, mesh operations 12-1
hide selection 8-39
hiding
boundaries 10-33
excitations 10-33
field overlay plots 24-231
windows 2-17
hierarchical components, exporting 13-87
high performance computing
configuring Ansys Electronics for 18-75
diagnostics 18-265
grid engine (GE) 18-113
HPC Platform Services Support 18-190
integration 18-72
load sharing facility (LSF) 18-129
Microsoft Windows HPC 18-78
overview 18-1
portable batch system (PBS) 18-139
remote simulation management (RSM) 18-246
schedulers 18-74
SLURM Scheduler (Linux Utility) 18-147
terminology 18-73
third party schedulers 18-153
highlighting selections 3-30
history tree 2-97
commands 2-111
controlling the view 2-100
options 3-33
organizing 2-99
selecting objects 2-106
Upgrade Version 2-112
upgrading modeling kernel 2-112
Housing UDP 9-237
HPC
setting options 3-52

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

hysteresis loss 28-59

I

IFFT 24-144

imaginary quantities, representing with phasors 29-43 , 29-43

impedance

and AC inductance 28-62 , 29-23

and AC resistance 28-65 , 29-23

and AC self-inductance 28-64

and AC self-resistance 28-64

and frequency 28-62 , 29-23

and time-varying currents 28-61 , 29-23

and time-varying voltages 28-61 , 29-23

difference in AC and DC computations 29-25

matrix 28-61 , 29-23

import

file (symbol) 13-39

importing

data tables 3-139

datasets 3-221

definition archives 13-3, 13-6

files 3-115

materials 6-102

table arrays 3-140

Importing Plot Data 24-152

inductance

AC vs. DC 28-65 , 29-25

and impedance 28-63 , 29-24

in terms of magnetic flux and currents 29-12

in terms of voltages and time-varying currents 29-14

open circuit 29-14

initial

padstack definition layer 13-68

initial seed value, setting for statistical analysis 25-145

Inner core regions 9-15

inserting designs 4-1

insulated systems, modeling with balloon boundaries 29-50

interactive scheduler jobs

design types 18-69

DSO configuration 18-65

options 18-63

overview 18-62

intersecting objects 7-289

Intrinsic function declarations 9-254

IPMCore UDPs 9-153

all magnets 9-155

inner core 9-154 , 9-154

one-pole magnets 9-156

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

J

jobs

aborting 18-240

advanced submission options 18-210

exporting 18-216

importing 18-216

monitoring 18-230

multi-step submission 18-216

scripting 18-228

submitting 18-204

submitting using command line 18-151, 18-214

Windows to Linux 18-221

K

keyboard shortcuts

3D modeler 2-115

custom 2-118

desktop 2-114

general 2-113

report window 2-116

L

lamination modeling 6-61

lande G factor 28-128

LapCoil UDPs 9-38

3D open insulation 9-43

bended coil 9-44

single lap coil 9-42

tooth coil 9-45

whole lap winding 9-42

large scale DSO 18-28

command line syntax 18-37

configuration 18-46

monitoring 18-45

outputs 18-40

post-processing 18-42

prerequisites 18-30

troubleshooting 18-60

user interface 18-31

Large Scale DSO, technical notes 18-57

Layout Editor

vias and pin padstacks 13-67

legacy Maxwell projects, opening 3-102

legends

settings 3-46

length-based refinement

inside objects 12-12

on faces 12-10

libraries

changing locations 13-2

component 13-1

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

definition 13-1
encrypted 13-74
personal 6-96
search path precedence 13-9
system 6-96
user 6-96
library
components, RF vendor 13-72
editor 13-84
precedence 13-2
structures and usage precedence 13-2
vendor components 13-72
Library Editor 13-84
lighting 8-56
linear constraints 25-226
linear materials
adding to libraries 6-12
properties of 6-13
LinearMCore UDPs 9-212
core 9-214
core and coils 9-214
one coil 9-215
lines
field plots on 24-229
line-start permanent-magnet synchronous motors 26-342
defining motors 26-344
functionality 26-342
general data 26-345
general procedure 26-344
stator data 26-346
stator data, windings 26-348
stator windings 26-348
Linux remote spawn 3-59, 3-61
Litz Wire modeling 6-63
load sharing facility (LSF) 18-129
loading
a footprint 13-50
log of solution tasks 24-5
logging errors 2-160
Lorentz force 28-50
Lorentz torque 28-28 , 28-50
and current 28-50
and Lorentz force 28-50
and magnetic flux density 28-50
theory (magnetostatic) 28-50

M

magnetic coercivity, in permanent magnets 6-111 , 6-111
magnetic field
behavior on even symmetry boundaries 29-49
behavior on odd symmetry boundaries 29-48

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in Ampere's Law 29-10
relationship to electric field (eddy current solver) 29-21
static 29-10
magnetic field simulation, time-dependent 29-36
magnetic flux 29-10
and inductance 29-12
magnetic flux density 29-10
and Lorentz torque 28-50
in nonlinear materials 28-40
magnetic loss tangent 6-49 , 6-49 , 28-125 , 28-125
defining anisotropy tensors 6-50
magnetic retentivity, in permanent magnets 6-111 , 6-111
magnetic saturation 28-127
magnetic vector potential 29-10
reference for 29-63
magnetostatic solver 29-1
flux linkage 29-17
inductance 29-12
specifying reference for potential 29-63
theory 29-10
virtual force 29-15
virtual torque 29-16
Magnetostriction model 28-133
Manage Files 13-8
markers
settings 3-47
table settings 3-47
X-Y settings 3-48
material browser, accessing 10-1 , 10-176 , 11-70 , 26-66
material characteristics
magnetic loss tangent 6-49 , 28-125
magnetostatic, nonlinear permeability 28-40
nonlinear materials 28-40
permeability 28-123
material coordinate systems 6-60
material properties
anisotropic 6-15
changing units of 6-12
conductivity 28-124
delta H 28-128
dielectric loss tangent 28-125
lande G 28-128
magnetic loss tangent 6-49 , 28-125
magnetic saturation 28-127
permeability 28-123
permittivity 28-124
simple 6-15

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

materials
about ferrites 28-127
adding to library 6-12
anisotropic tensors 6-37
assigning 6-1 , 6-89
assigning properties 6-14
assigning to objects 10-1 , 10-176 , 11-70 , 26-66
copying 6-93
deleting 6-94
editor 13-72
exporting 6-94
filtering 6-95
functional properties 6-59
importing and converting 6-102
libraries 6-96
modifying attributes 6-90
nonlinear 28-40
properties as expressions 6-59
removing 6-94
reverting 12-53
searching 6-7
sorting 6-94
thermal modifiers 6-51
validating 6-93
variable properties 6-51
viewing attributes 6-90
viewing definitions 2-24
MATLAB optimization 3-11
matrices
admittance 28-84 , 28-85 , 29-35
capacitance 29-5
diagonal terms in 29-7
mutual capacitance 29-5
off-diagonal terms in 29-7
self-capacitance 29-6
sequence of solutions 29-8
conductance 29-32
and current 29-32
and voltage drops 29-32
impedance 28-61 , 28-61 , 29-23
and inductance 28-63
and resistance 28-64
diagonal terms in 28-61 , 29-23
matrix elements 28-62
monitoring the solution 28-63
mutual impedance 28-62 , 29-23
off-diagonal terms in 28-61 , 29-23
self-impedance 28-62 , 29-23
setting sources during solution 28-63
solution process 28-63 , 29-24
symmetry 28-63
theory 28-61 , 29-23

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

inductance 29-12

diagonal terms in 28-62 , 29-13

mutual inductance 29-14

off-diagonal terms in 28-62 , 29-12

self-inductance 29-13

maximum delta S, viewing results 24-5

Maxwell ACT Extensions 3-253

Maxwell's equations 29-20

Maxwell>Excitations>Conduction Paths>Recalculate Conduction Paths 10-32

Maxwell>Excitations>Conduction Paths>Show Conduction Paths 10-31

Maxwell>Excitations>Conduction Paths>Verify Conduction Paths 10-31

measuring modes 7-345

memory, used during solution 24-5

menu bar 2-49

menu bar, overview 26-5

menus

customizing 2-85

design-specific 2-64

draw menu (layout editor) 2-66

edit menu 2-53

edit menu (3D Layout) 2-53

edit menu (report window) 2-56

file menu 2-51

help menu 2-78

project menu 2-63

shortcut menus 2-81

tools menu 2-67

adding tools 2-70, 2-74

view menu 2-56

view menu (3D Layout) 2-58

view menu (basic) 2-57

window menu 2-76

menus, shortcut menus 26-5

mesh

region 28-13

size vs. accuracy 28-3

mesh generation

and surface approximation settings 12-46 , 28-11

mesh operations

defining 12-1 , 12-1

HFSS 12-1

model resolution 12-48

mesh operations, surface approximation overview 12-46 , 28-11

mesh refinement

defining mesh operations 12-1

length based 28-6

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

percent refinement per pass 15-6

meshing

deleting refinements 12-83

generating mesh without solve 12-53

mesh properties 12-86

operations list 12-84

plotting a mesh 24-296

reassigning operations 12-85

settings 12-2

statistics 24-17

TAU flex 12-54

meshing, detecting and addressing problems 28-115

message manager 2-28

action messages 2-30

automatically expanding 2-30

clearing messages 2-31

hiding 2-32

showing new messages 2-29

Message severity 9-254

Microsoft Windows HPC

commands 2-157

integration with 18-78

Minerva remote storage environment 3-165

model resolution, mesh operations 12-48

modeling kernel

upgrading version 2-112

models

analyzing 7-100

cleaning up history of 7-369

drawing 7-1

setting drawing plane 7-368

setting units of measure 4-4 , 7-1

Models Editor, using 13-71

monitor job 18-230

Motion Variables 2-27

mouse sensitivity 3-27

movement mode 7-338

MPI Vendor, selecting 18-76

multi-step job submission 18-216

multithreading 23-48

mutual capacitance 29-5

mutual impedance 28-62 , 29-23

mutual inductance 28-63 , 29-14

N

named expressions, plotting 24-187

names, legal for excitations and boundaries 10-38

netlist

syntax 13-25

netlist, exporting 27-92 , 27-92

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

network data explorer 23-1
cell filtering 23-21
exporting data 23-6
loading data 23-5
menu commands 23-17
modifying the display 23-32
overview 23-2
smoothing 23-20
Neumann boundaries, relationship to even symmetry boundaries 29-49
new projects, creating 26-2
non-cylindrical rotational motion 10-183 , 11-77
nonlinear materials 28-40 , 28-40 , 28-40
relationship of B and H in 28-40
NonSalientPoleCore UDPs 9-27
core with a Q-axis damper 9-30
core with a uniform-distributed damper 9-36
inner core 9-30
notes, saving with project 26-3

O

objects
assigning color to 7-240
assigning coordinates to 7-334
assigning transparency to 7-242
bondwires 7-77
boolean operations on 7-240
boxes 7-54
center-point arc 7-12
circles 7-29
cones 7-60
connecting 7-277
creating a list of 7-317
creating cross-section 7-277
creating from edges 7-290
creating from faces 7-289
creating from intersections 7-288
cylinders 7-49
deleting 7-245
drawing 7-3
duplicating 7-253
ellipses 7-32
equation-based curves 7-24
equation-based surface 7-42
fitting 8-37
generate history of 7-307
helices 7-67
hiding 8-39
imprinting 7-301
mirroring 7-251
modifying 7-239
modifying attributes 2-36

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

moving 7-247
non-model objects 7-82
offsetting 7-253
orientation of 7-251
planes 7-79
points 7-78
polygons 7-39
polyhedra 7-57
polylines 7-20
polylnes 7-19
purging history of 7-307
rectangles 7-36
rotating 7-249
scaling 7-258
segmented 7-80
selecting 7-309
selecting multiple types 7-328
separating bodies 7-295
setting deformation 6-107
showing 8-41
showing and hiding 8-2
solid 8-47
spheres 7-46
spirals 7-73
splines 7-16
splitting 7-291
straight lines 7-5
subtracting 7-286
sweeping 7-260
three-point arcs 7-10
tori 7-64
uniting 7-286
wireframe 8-47
Odd Symmetry (Flux Normal) 28-19
odd symmetry boundaries
behavior of magnetic field 29-48
in magnetostatic models 29-48
relationship to value boundaries 29-48
used as reference 29-63
ohmic loss, formula 28-59
old Maxwell projects, opening 3-102
open circuit 29-59
open circuit inductance 29-14
opening projects in RMxprt 26-2
opening the Fields calculator 24-242
opening, field overlay plots 24-231
operators
list of valid 3-245
optimetrics
design of experiments 25-154
linear constraints 25-226
link to DesignXplorer 25-231
optimization analysis 25-15

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

overview 25-1
parametric analysis 25-2
sensitivity analysis 25-128
statistical analysis 25-142
viewing results 25-229
optimization
choosing an optimizer 25-16
overview 25-15
setting up analysis 25-61
optimization analysis
Fast Calculation-Update 25-191
optimization setups
adding a cost function 25-108
setting a goal 25-108
options
3D modeler 3-24
advanced 3-38
Discovery 3-35
display 3-29
drawing 3-25
group 3-34
operations 3-26
snapping 3-27
3D user interface 8-3
animation 3-17, 3-43
auto-save 3-112
axis 3-45
beta 3-8
clipboard 3-50
clone 3-26
component 3-22
curve 3-44
digital 3-49
exporting as file 3-149
faceting 3-38
general 3-6
component libraries 3-22
default units 3-20
desktop configuration 3-8
desktop performance 3-17
directories 3-16
miscellaneous 3-11
project 3-10
remote analysis 3-20
user interface 3-12
grid 3-45
group 3-34
history tree 3-33
HPC and analysis 3-52
legend 3-46
marker table 3-47
model edit 3-26
model save 3-26
mouse sensitivity 3-27

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

notes 3-46
object appearance 3-32
object outline 3-32
object transparency 3-32
operation history 3-30
polyline 3-25
pre/post-processing 3-17
relative coordinate system 3-25
rendering 3-32
report headers 3-46
report markers 3-47, 3-48
report table 3-51
report update 3-17
Report2D 3-44
reporter 3-42
fields reporter 3-43
report setup 3-42
reports 3-50
RSM 3-20
selection 3-30
setting via configuration files 3-71, 3-75
significant digits 3-42
snap 3-27
stacked 3-49
UDM/UDP 3-38
options, Symbol Editor 13-30
Order Coil Terminals 5-7
Other Cores and Coils UDPs 9-199 , 9-199
Outer core regions 9-15
output variable, viewing convergence 24-5
output variables
adding 24-145
deleting 24-147

P

pad behavior, vias and pins 13-66
padstack
creating new 13-58
Edit Padstack Definition window 13-59, 13-64
Padstack Definition Translations 13-69
padstack editor 13-58
padstact
initial 13-68
pancake commutator type 26-273, 26-378, 26-405
parallel current sources
AC source current in 29-60
displacement current in 29-60
eddy current in 29-61
parameter
reserved 13-21
reserved component 13-24
statistics display 13-23

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

parameter data, viewing 24-16 , 24-19 , 24-19 , 24-20
parameter defaults
adding from components 3-206
converting to variables 3-206
defining 3-198
deleting 3-198
editing 3-198
overriding 3-198
passive conductors
displacement currents in 29-59
eddy currents in 29-59
password manager 2-124
passwords, setting up 13-76
percent error 15-6
percent refinement per pass, setting 15-6
perfect conductors, assigning AC currents to 29-60
permanent magnets
linear vs. nonlinear 6-111 , 6-111
nonlinear 6-111 , 6-111
permanent-magnet DC motors
brush displacement 26-274, 26-379, 26-406
brush voltage drop 26-273, 26-378, 26-405
commutator and brush data 26-273, 26-378, 26-405
brush pairs 26-273, 26-379, 26-405
brush width 26-273, 26-379, 26-405
commutator diameter 26-273, 26-378, 26-405
commutator insulation 26-273, 26-378, 26-405
commutator length 26-273, 26-378, 26-405
commutator type 26-273, 26-379, 26-405
mechanical pressure of brushes 26-273, 26-378, 26-405
general data 26-260, 26-413
rotor data 26-264, 26-423
rotor slots 26-423
permeability 15-13 , 28-123
anisotropic 28-128 , 28-128
nonlinear 28-40
of free space (Eddy Current) 29-22
of free space (Magnetostatic) 29-10
relative (Eddy Current) 29-22
relative (Magnetostatic) 29-10
permittivity 28-124
anisotropic 28-129 , 28-129

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of free space 29-3
relative 29-3
personal library 6-96
personal library directories 13-1
personallib 13-2
phase, modifying for field overlays 24-158
phasors
notation for 29-43 , 29-43
real and imaginary components 29-44
used to express current 28-61 , 29-23
used to express voltage 28-61 , 29-23
pin
locations, editing 13-41
padstacks and vias
editing 13-67
properties, editing 13-40
pin and net operations 13-69
PinToPin utility 24-396
placing components 27-13
planes 7-79
Plot Data
Importing 24-152
plots
2D rectangular 24-66
3D rectangular 24-70
3D rectangular bar 24-77 , 26-123
data tables 24-88
deleting field overlays 24-232
hiding 24-231
limit lines in 24-115
modifying field quantities 24-198
named expressions 24-187
opening field overlays 24-231
rectangular contour 24-89
saving field overlays 24-231
spectral domain 24-144
time domain from frequency 24-144
y markers 24-100
plotting
basic quantities 24-158
derived quantities 24-187
Versus Adaptive Pass 24-87
Plotting Standard Field Quantities
Current 24-163
Dielectric Loss 24-164
Direct Current Density 24-163
Edge Loss 24-163
Electric Field Intensity 24-164
Electric Flux Density 24-163
emloss 24-163

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Field Energy 24-163

Hysteresis Loss 24-164

Magnetic Co-energy 24-163

Magnetic Field 24-163

Magnetic Flux Density 24-163

Ohmic Loss 24-164

Surface Current Density 24-164

Total Loss 24-164

Vector Potential 24-163

Voltage 24-164

PMCore UDPs 9-147

all magnets 9-151

inner or outer core 9-149

one magnet 9-151 , 9-152

PMDamperCore UDPs 9-157

all magnets 9-162

damper 9-161

inner core 9-160

one magnet 9-163

points 7-78

polar plots

creating 3D 26-132

Pole Cores and Coils UDPs 9-80

pole embrace (DC motors) 26-254

PolygonHelix UDPs 9-243 , 9-243

left-handed 9-248

right-handed 9-248

port

create, footprint editor 13-53

remove, footprint editor 13-53

portable batch system (PBS) 18-139

ports, and anisotropic materials 28-132

post processing

overview of options 24-1 , 26-103

viewing convergence data 24-2

viewing parameter data 24-16 , 24-19 , 24-19 , 24-20

viewing profile data 24-5

Post-processing Data, updating 24-151

potential 29-4

precedence, library usage 13-2

primary sweep

modifying the variable 26-140

specifying for 2D rectangular plots 24-67 , 26-115

specifying for 3D rectangular plot 26-117, 26-132

specifying for data tables 26-137

Primitives, copy and paste properties 27-16

printing 3-164

priority of boundaries, changing 10-36

pro, premium, enterprise licensing 3-8

problem region 28-13

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

profile information, viewing 24-5
progress window 2-32
project manager window 2-21
project tree
automatically expanding 2-23
field overlays 24-158
Field overlays 24-177
field plot folders 24-158
Fields overlays 24-159 , 24-174 , 24-182 , 24-183
fields plot folders 24-159 , 24-174 , 24-177 , 24-182
overview 2-22
visualization options 3-12
project, loading footprint into 13-50
projection view 8-59
projects
about 3-1
archiving 3-153
copying and pasting 3-152
creating new 3-2, 3-104
deleting 3-160
design notes 3-163
example projects 3-172
exporting 3-144
opening 3-99
renaming 3-153
saving 3-110
validating 3-149
working with 3-1
Projects
creating new 26-2
default names 26-27
managing 26-27
opening legacy Maxwell 3-102
saving notes 26-3
properties
auto-completing 2-120
Properties
copying and pasting 27-15
defining for components 13-21
pin, editing 13-40
properties dialog box 2-41
general and symbol tabs 2-43
opening 2-42
parameter values tab 2-42
property displays tab 2-43
properties window 2-34
automatically opening 2-35
modifying command properties 2-36
modifying object attributes 2-36
opening 2-35
showing and hiding 2-36
property display setup, symbol 13-46

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Property Window for Schematic and Layout Editors 2-37

Property window, closing 2-41

purge history 7-307

Q

quantities

plotting S-parameter 24-130 , 26-147

quick report 26-148

R

RacetrackSlotCore UDPs 9-25

outer core 9-27

RAM, used during solution 24-5

range functions 24-135

read-only designs 3-160

recalculating conduction paths 10-32

rectangular plots

creating 2D 26-115

creating 3D 26-117

rectangular stacked plots, creating 2D 24-67

RectHelix UDPs 9-249

left-handed 9-251

right-handed 9-251

redo command 3-162

reference for voltage and magnetic vector potential 29-63

regions

drawing 7-83, 7-86, 7-92

registry entry

overriding 2-145

relative permeability 28-123

and skin depth 29-22

defining anisotropy tensors 6-27

relative permeability (Magnetostatic) 29-10

relative permittivity 28-124 , 29-3

defining anisotropy tensors 6-31

release

obtaining information about 2-80

remote analysis

overview 18-1

troubleshooting 18-4

remote simulation management (RSM) 18-246

remote spawn (Linux) 3-59, 3-61

Report Data 24-150

report templates, creating 24-154

report window 2-96

reports

adding traces 26-138

creating 2D rectangular plots 26-115

creating 2D rectangular stacked plots 24-67

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

creating 3D polar plots 26-132
creating 3D rectangular plots 26-117
creating a quick report 24-38
creating data tables 26-137
creating quick reports 26-148
custom templates 24-62
data tables 24-88
display types 26-115
exporting 24-151
file formats 24-153
importing plot data 24-152
modifying 24-50
modifying data in 26-114
modifying properties 24-56
overview 26-114
perform FFT 24-136
perform TDR 24-141
selecting a function 26-141
sweeping variables 26-140
updating 24-151
user defined documents (UDDs) 24-367
user defined outputs (UDOs) 24-322
zooming and fitting 24-56
reprioritizing, boundaries 10-36
reserved component parameters 13-24
resistance
AC vs. DC 28-65 , 29-25
and conductance 29-32
and impedance 28-64 , 29-23
in eddy current problems 28-64
re-solving a problem 26-102
results window 2-96
reverting to initial mesh 12-53
RF vendor, library components 13-72
RMxprt
changing the machine type 26-35
general procedure 26-29, 26-31, 26-35
setting up a model 26-29
RMxprt projects 26-2
rotor pole diagram 26-298
run commands 2-125, 2-145, 2-154

S

SalientPoleCore UDPs 9-80
coil and terminals 9-86
damper 9-86
inner or outer core 9-85
poles or yoke 9-88 , 9-90
save options 3-8
saving fields
for a tuning analysis 25-222

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

saving, field overlay plots 24-231
scalar field plots
creating 24-190
schedulers
Ansys RSM 18-246
GE 18-113
HPC Platform Services Support 18-190
integration using IronPython 18-182
LSF 18-129
Microsoft Windows HPC 18-78
overview 18-74
PBS 18-139
proxy interfaces 18-171
selecting 18-203
SLURM 18-147
terminology 18-73
testing custom integration 18-179
testing integration 18-177
third party 18-153
schematic, placing components 27-13
script editor 13-48
Search Tab 27-13
secondary sweep
modifying the variable 26-140
specifying for 3D rectangular plot 26-117, 26-132
seed value 25-145
seeding the mesh 28-5
selecting
edges 7-324
faces 7-319
groups 7-330
multi select 7-328
objects 7-309
objects behind objects 7-331
showing selected objects 8-41
submodels 7-330
vertices 7-327
selecting a machine type 26-2
self-capacitance 29-6
self-impedance 28-62 , 29-23
self-inductance 28-62 , 29-13
sensitivity analysis 25-128
choosing variables 3-213
setting up designs 4-1 , 26-149
setting up projects 4-1 , 26-149
settings
surface approximation 12-42
Settings, Design 4-5 , 26-35
setup link dialog 15-98
setups, solution 15-1 , 26-91

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ShadedPoleCore2 UDPs 9-102
coil and terminals 9-106
core with poly holes 9-110
jointed shaded conductor and terminals 9-108
outer shaded pole core 9-105
separated shaded conductor and terminals 9-108
SheetScan 3-233
checking monotonicity 3-241
defining characteristic curve 3-238
exporting data 3-244
importing characteristic data 3-242
loading datasheets 3-236
short circuit 29-59
short circuit capacitance 29-7
shortcut menus 2-81
3D modeler 2-82
project manager 2-83
shortcut menus, overview 26-5
showing conduction paths 10-31
significant digits
setting 3-42
simulations
aborting 2-33
changing solution priority 22-6
monitoring 22-5
monitoring queued 22-5
re-solving 26-102
re-solving after modifying design 22-8
re-solving after thermal feedback 22-8
running 22-1
running multiple 22-3
starting 21-1 , 26-102
stopping 2-33, 22-7, 26-102
single-phase induction motors
defining the motor 26-211
general data 26-212
rotor data 26-226, 26-229
rotor slots 26-227
stator data 26-215
inner diameter 26-226
outer diameter 26-226
slot type 26-226
slots 26-226
stator slots 26-216
wire diameter 26-221
wire gauge 26-221, 26-223
stator windings 26-218
skin depth
and conductivity 29-22
and eddy currents 29-22

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and frequency 29-22

and magnetic field 29-22

and permeability 29-22

effect on AC resistance 28-65 , 29-25

skin depth-based refinement, creation of layers 28-10

Slot Cores and Coils UDPs 9-16

SlotCore UDPs 9-16

inner or outer slot cores 9-20 , 9-20

SLURM Scheduler 18-147

solid current sources, AC source current, displacement current and eddy current 29-60

solid sources 29-60

solution data

deleting 24-21

viewing 26-103

viewing convergence data 24-2

solution process

tetrahedra used during 24-5

viewing memory used during 24-5

viewing profile data 24-5

viewing tasks performed 24-5

solution settings

percent refinement per pass 15-6

setting up adaptive analyses 15-5

setting up electric transient analyses 15-88

setting up transient analyses 15-46

specifying 15-3 , 26-91

solution setups

adding 15-1 , 26-91

solutions

aborting 22-7

after modifying the model 26-102

changing priority 22-6

monitoring 22-5

re-solving 26-102, 26-102

re-solving after modifying design 22-8

re-solving after thermal feedback 22-8

setting sources to compute impedance 28-63

solving 22-1

starting 21-1 , 26-102

stopping 26-102

Solver 5-1

Eddy Current 5-2

Electrostatic 5-1

Magnetostatic 5-2

Solver On Demand Tab 13-17

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

solvers
3dtransientaphi 5-4
eddy current 5-2
electrostatic 5-19
magnetostatic 5-2
material properties for 5-23
transient 5-4
solving 21-1 , 26-102
inside an object 6-7
Solving Inside 6-6
Solving on Surface 6-6
Source code files 9-270
Band.cpp 9-284
Band.h 9-283
UDPBase.cpp 9-273
UDPBase.h 9-270
source current
in passive conductors 29-59
relationship to AC current density 29-18
sources
AC conduction
required 29-55
Voltage 29-55
DC conduction
required 29-55
Voltage 29-55
eddy current
Current 29-55
parallel current 29-60
passive 29-59
perfect current 29-60
required 29-55
solid current 29-60
stranded current 29-60
electrostatic
Charge 29-55
Floating 29-55
required 29-55
Voltage 29-55
functional 29-64
magnetostatic
Current 29-55
required 29-55
transient, required 29-55
S-parameters, plotting quantities 24-130 , 26-147
spinning the view 8-31
Square Stator Yoke for Stepper Motors 9-178
Square Yoke UDPs 9-175
SquirrelCage UDPs 9-55
bars only 9-58
two rings only 9-58

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

whole squirrel cage winding 9-58

SRMCore UDPs 9-121

coil and terminals 9-126 , 9-127 , 9-127

core and all coils 9-125

inner or outer core 9-124

stacking direction 6-63

statistical analysis 25-142

choosing variables 3-214

statistical setups

setting the initial seed value 25-145

status bar 2-113

steady-state conditions 29-31

StepMCore UDPs 9-166

coil and coil terminals 9-173

inner or outer core 9-170

inner rotor core 9-172

stator core with non-uniform pole pitch 9-174

stopping an analysis 26-102

stranded conductors 29-36

stranded current sources, AC source current in 29-60

submit job 18-204

surface approximation 12-42

guidelines for setting 12-47 , 28-12

overview 12-46 , 28-11

surfaces

curved 8-51

equation-based 7-42

susceptance 28-85 , 29-35

Sweep functions 9-255

sweep variables in reports

modifying values 26-140

sweeping

along a path 7-264

along a vector 7-262

around an axis 7-260

draft types 7-266

faces along normal 7-266

objects 7-260

sweeps

orphaned 3-209

variable 3-209

switched reluctance motors

defining reluctance motors 26-330

general data 26-331

stator coil data 26-335

Symbol draw menu 13-31

symbol editor 13-28

Symbol Editor options 13-30

symbol grid, setup window 13-47

Symbol Menu 13-37

symbol property display setup 13-46

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

symbol, creating new 13-29

symbol, editing existing 13-30

symbol, import file 13-39

Synchronous reluctance machines 9-129

SynRMCore UDPs 9-129

different barriers 9-136

inner core 9-135

syslib 13-2

system directories 13-1

system library 6-96

system resources

changing solution priority 18-270

requirements 2-5

T

TAU flex meshing 12-54

technical support 1-12

temperature-dependent BH curves 6-29

templates for reports, creating 24-154

temporary directory

configuration file format 3-88

setting 3-86

using UpdateRegistry command 3-89

Terminals Tab, Edit Components window 13-16

tetrahedra

setting number added each pass 15-6

used during solution 24-5

theory

AC magnetic field (eddy current) simulation 29-17

DC conduction field simulation 29-29

DC magnetic field (magnetostatic) simulation 29-10

electrostatic field simulation 29-4

thermal feedback 22-8

three-phase induction motors

defining rotor slots 26-205

defining rotor vents 26-205

defining stator conductors 26-195

defining stator windings 26-195

defining the motor 26-190

general data 26-191

friction and wind loss 26-191, 26-212, 26-237, 26-260, 26-308, 26-331, 26-345, 26-362, 26-386, 26-413

motor speed 26-208, 26-230, 26-256, 26-275, 26-303, 26-326, 26-341, 26-359, 26-381, 26-408, 26-425

motor voltage 26-208, 26-230, 26-256, 26-275, 26-303, 26-326, 26-341, 26-359, 26-381, 26-408, 26-425

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

output power 26-208, 26-231, 26-257, 26-276, 26-303, 26-326, 26-341, 26-359, 26-381, 26-408, 26-425

type of load 26-208, 26-230, 26-256, 26-275, 26-303, 26-326, 26-341, 26-359, 26-381, 26-408, 26-425

winding connection 26-208

rotor data 26-203, 26-207, 26-253, 26-256, 26-275, 26-303, 26-322, 26-325, 26-339, 26-340, 26-355, 26-358, 26-369, 26-381, 26-395, 26-407, 26-425

end ring 26-206, 26-229

stator data 26-192

conductor length adjustment 26-195, 26-243, 26-266, 26-284, 26-313, 26-335, 26-348, 26-372, 26-398, 26-416

conductors 26-195

inner diameter 26-192, 26-203, 26-215, 26-240, 26-253, 26-261, 26-264, 26-281, 26-294, 26-311, 26-322, 26-334, 26-339, 26-355, 26-363, 26-370, 26-395, 26-414, 26-423

outer diameter 26-192, 26-203, 26-215, 26-240, 26-253, 26-261, 26-264, 26-281, 26-294, 26-311, 26-322, 26-334, 26-339, 26-355, 26-363, 26-370, 26-387, 26-395, 26-414, 26-423

parallel branches 26-195, 26-243, 26-284, 26-313, 26-335, 26-348, 26-367, 26-416

slot type 26-193, 26-203, 26-241, 26-264, 26-282, 26-311, 26-346, 26-414

slots 26-192, 26-203, 26-240, 26-264, 26-282, 26-311, 26-334, 26-339, 26-346, 26-414

stator slots 26-194

windings 26-195

wire diameter 26-198, 26-247, 26-268, 26-288, 26-317, 26-337, 26-349, 26-367, 26-374, 26-399, 26-418

wire gauge 26-198, 26-247, 26-268, 26-288, 26-317, 26-337, 26-349, 26-367, 26-374, 26-399, 26-418

wire wrap 26-197, 26-247, 26-267, 26-288, 26-316, 26-337, 26-349, 26-367, 26-373, 26-399, 26-417

three-phase inductions motors, stator data, winding type 26-195, 26-243, 26-266, 26-284, 26-313, 26-372, 26-398, 26-416

three-phase non-salient synchronous generators, general data 26-430

three-phase non-salient synchronous

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

machine
stator data
stator slots 26-432
stator skew 26-433
three-phase synchronous generators
exciter efficiency 26-303
friction loss 26-280
general data 26-281
rotor pole data 26-298
rotor pole, diagram 26-298
rotor winding data 26-299
parallel branches 26-300
winding type 26-299
wire wrap 26-300
stator data 26-281
stator slots 26-283
stator ducts 26-284, 26-433
stator skew 26-284
stator winding, end clearance 26-195, 26-243, 26-266, 26-284, 26-313, 26-348, 26-372, 26-398, 26-416
time
real vs. CPU 24-5
viewing for solution tasks 24-5
time constant, for electrostatic and DC conduction simulation 29-32
time domain plots from eddy current solution 24-144
time-varying quantities
real and imaginary components 29-44
representing with phasors 29-43
Toolkit command 24-363
torque
Lorentz (magnetostatic) 28-50
virtual (Eddy Current) 29-27
virtual (electrostatic) 28-29
virtual (magnetostatic) 28-51 , 29-16
traces
adding blank 26-140
adding characteristics to 24-107
adding data markers 24-98
adding to reports 26-138
editing properties 24-94
overview 24-92
removing 24-113 , 26-139
replacing 26-139
y markers 24-100
Traces dialog box 26-114
TransCoil UDPs 9-220
one leg coil 9-223
three legs 9-223

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TransCore UDPs 9-216
three legs only 9-219
two yokes 9-220
Transform operations 9-256
transien solver
core loss 29-41
transient analysis, setting up 15-46
Transient GPU 18-241
transient motion, restrictions 29-35
transient simulation 29-35
transient solid conductors 29-37
current sources 29-38
voltage sources 29-38
translating legacy projects 3-102
translational motion equations 29-39
Translations, of Padstacks 13-69
TransTwist UDPs 9-226
core and all twist coils 9-230
twist insulator 9-232
twist wire and terminals 9-231
tuning 3-212, 25-214
two-level distribution
guidelines 18-27

U

UDPBase.cpp 9-273
UDPBase.h 9-270
UDPs
band 9-4
bands and regions 9-4
ClawPoleCore 9-143
coding 9-252 , 9-252
intrinsic structures 9-252
ConCoil 9-47
core region 9-15
CupCoil 9-63
DCMCore 9-91
DiskCoil 9-204
DiskPMCore 9-208
DiskSlotCore 9-199
DoubleCage 9-59
EndCap 9-242
HairpinCoil 9-68
helix 9-243
Housing 9-237
intrinsic head files 9-252
IPMCore 9-153
LapCoil 9-39
LinearMCore 9-212
machine parts, other 9-233
NonSalientPoleCore 9-27
other cores and coils 9-199
PMCore 9-147
PMDamperCore 9-157

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

pole cores and coils 9-80
PolygonHelix 9-243
PressPlate 9-234
RacetrackSlotCore 9-25
RectHelix 9-249
SalientPoleCore 9-81
sample coding 9-263
ShadedPoleCore2 9-102
slot cores and coils 9-16
SlotCore 9-17
source code files 9-270
Square Yoke 9-175 , 9-178
SquirrelCage 9-55
SRMCore 9-121
StepMCore 9-166
SynRMCore 9-129
TransCoil 9-220
TransCore 9-216
TransTwist 9-225
UnivMCore 9-116
UPMCore 9-180
VentCore 9-191
VentSlotCore 9-22
VPMCore 9-185
WaveCoil 9-52
undo command 3-162
units of measure
setting default 3-20
universal motors
defining motors 26-362
functionality 26-360
general data 26-363
general procedure 26-362
UnivMCore UDPs 9-116
all coils 9-119
coil and terminals 9-120
outer universal motor core 9-118
Updating post-processing data 24-151
Updating Reports 24-151
UPMCore UDPs 9-180 , 9-181
ducts 9-184
magnets 9-182
user defined documents (UDDs) 24-367
user defined outputs (UDOs) 24-322
user directories 13-1
user library 6-96
user tools, customize 2-157
user-defined models (UDMs) 7-156
user-defined primitives (UDPs) 7-131
userlib 13-2
Using Definition Libraries 13-1
Using the Component Editor 13-11

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using the Library Editor 13-84
Using the Material Editor 13-72
Using the Padstack Editor 13-58
Using the Script Editor 13-48
Using the Symbol Editor 13-28
U-Type IPMCore 9-181
UV isolines 3-30

V

validation check 3-149, 26-44
value boundaries
  relationship to odd symmetry boundaries 29-48
  used as reference 29-63
variables
  adding from components 3-206
  and properties, auto-complete 2-120
  auto-completing 2-120
  constants 3-192
  copying and pasting list 3-215
  design-level 3-192
    arrays 3-201
    converting to parameter defaults 3-206
    defining 3-193
    editing 3-193
    overriding 3-193
    parameter defaults 3-198
  fixed vs. non-fixed 3-209
  for output convergence 15-7
  in sensitivity analysis 3-213
  in statistical analysis 3-214
  intrinsic 3-190
  modifying value of a fixed 25-225
  optimizing 3-211
  output 24-145
  overview 3-179
  project-level
    adding 3-181
    arrays 3-186
    deleting 3-181
    editing 3-181
  sweeping 24-122
  tuning 3-212, 25-214
vector field plots
  creating 24-197
  modifying arrows 24-220
Vendor Library Components, components, vendor library 13-72
VentCore 9-195
VentCore Notches 9-198
VentCore Shaft 9-197
VentCore UDPs 9-191 , 9-195 , 9-197 , 9-197 , 9-198

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VentCore Vent Ducts 9-198
VentSlotCore UDPs 9-22
inner or outer cores 9-25
verifying conduction paths 10-31
vertices 7-327
vias
and pin padstacks
edit in layout 13-67
vias and pins, pad behavior 13-66
view options
spin 8-31
view navigation options 2-117
view options
rotate 8-25
viewing data markers 24-229
virtual force
and current 28-28 , 28-50 , 29-15
and energy 29-8
and magnetic coenergy 28-28 , 28-50 , 29-15
and virtual work (Electrostatic) 28-28 , 29-8
and virtual work (magnetostatic) 28-50 , 29-15
instantaneous 29-26
theory (eddy current) 29-26
theory (electrostatic) 28-28 , 29-8
theory (magnetostatic) 28-50 , 29-15
time-averaged value of 29-26
virtual torque
and magnetic coenergy 28-51 , 29-16
and virtual force (electrostatic) 28-29
and virtual force (magnetostatic) 28-51 , 29-16
instantaneous 29-27
theory (eddy current) 29-27
theory (electrostatic) 28-29
theory (magnetostatic) 28-51 , 29-16
time-averaged value of 29-27
virtual work
and force (electrostatic) 28-28 , 29-8
and force (magnetostatic) 28-50 , 29-15
and torque (electrostatic) 28-29
and torque (magnetostatic) 28-51 , 29-16
visibility
of field overlay plots 24-231
of mesh on field plots 24-221
visualization options
for boundaries 10-33
for excitations 10-33
voltage
and inductance 29-14

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and balloon boundaries 29-50
and capacitance 29-5 , 29-5
reference 29-63
time-varying 29-7
voltage sources
AC, used as reference 29-63
DC, used as reference 29-63
VPMCore UDPs 9-185 , 9-187
ducts 9-190
magnets 9-189
V-Type IPMCore 9-187

W

WaveCoil UDPs 9-52
single wave coil 9-55
whole wave winding 9-55
winding data, exporting 26-186
window displays
3D modeler 2-95
ACT extensions 2-44
message manager 2-28
progress window 2-32
project manager 2-21
properties window 2-34
report window 2-96
results window 2-96
status bar 2-113
windows
auto-hiding 2-17
docking 2-17
moving and resizing 2-17
showing and hiding 2-17

Z

zooming 8-9

Ansys Electromagnetics Suite 2025 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.